Karpilovsky V.S., Kriksunov E.Z., Malyarenko A.A., Mikitarenko M.A., Perelmuter A.V., Perelmuter M.A., Fedorovsky V.G., Yurchenko V.V.

SCAD Office.

Implementation of SNiP in Computer-Aided Design Applications

Fifth edition, revised and enlarged

Electronic version

Kyiv• 2024

ISBN 978-617-7031-98-6

Karpilovsky V.S. et al.

SCAD Office. Implementation of SNiP in Computer-Aided Design Applications / Karpilovsky V.S., Kriksunov E.Z., Malyarenko A.A., Mikitarenko M.A., Perelmuter A.V., Perelmuter M.A., Fedorovsky V.G., Yurchenko V.V., –: SCAD SOFT, 2024. – 671 p.

ISBN 978-5-903683-32-1

This book is intended for the users of **SCAD Office**, and contains the description of **WeST**, **Kristall**, **ARBAT**, **KAMIN**, **Monolit**, **COMET**, **Decor**, **Pasternak**, **Cross**, **Slope**, and **ZAPROS**. **Kristall**, **ARBAT**, and **KAMIN** enable to perform structural analysis and various checks of members of steel, reinforced concrete, masonry, and reinforced masonry structures. **Monolit** enables to design in-situ ribbed floors. **COMET** is used to check and design the most common types of joints of steel bar structures used in civil and industrial engineering. **DECOR** enables you to perform structural analysis and various checks of members and joints of timber structures. **WeST** enables to determine loads and actions on building structures and can be useful both when using general-purpose computing systems, and such programs as **Kristall** and **ARBAT**. **CROSS** enables to calculate Winkler coefficient using the results of geological surveys. **Pasternak**, **Slope** and **ZAPROS** enable to perform the analysis of the action and behavior of soils.
Most of the described programs are based on the requirements of the design codes (SNiP, SP, DBN, EN(EC3)).
This book can be interesting for students of the respective specialties and for the developers of similar software.

Reviewers: D.Sc., prof. Gordeev V.N.; D.Sc., prof. Permyakov V.A.

Approved by the Academic Board of the V. Shimanovsky Ukrainian Institute of Steel Construction.

TABLE OF CONTENTS

Preface

The software package for designing steel, reinforced concrete and masonry structural members was developed by the ***SCAD Soft*** company: I. Belokopytova, I. Gavrilenko, S. Girenko, V. Karpilovsky, E. Kriksunov, P. Licman, M. Mikitarenko, A. Perelmuter, M. Perelmuter, D. Rud, A. Sementsov, V. Yurchenko.

We are grateful to A. Varanitsa (Vinnitsa), I. Vishnitsky (TEP, Moscow), B. Vorobyev (Petrostroysystem, Saint Petersburg), O. Kabantsev (MGSU, Moscow), O. Kalosha (Minskproject, Minsk), L. Katsnelson (Central Research Institute for Steel Construction, Moscow), A. Korablev (IN RE, Vilnius), V. Kretov (NIISK, Kiev), V. Kulikov (Moscow Engineering Design Institution), N. Mosina (SCAD Soft, Moscow), T. Prokhorova (Industrial Construction Design Institution, Moscow), S. Reshetnikov (Krasnodarproject, Krasnodar) for critical remarks and suggestions that contributed to the development of the software package.

We are also very grateful to I. Laykina for her patience and work invested in preparing the book for publication.

1. General Information

1.1 Principles of Development

Programs described here are based on the requirements of the codes for designing steel, reinforced concrete, masonry, and timber structures. These applications were developed because **SCAD** [1] is intended mostly for analyzing systems as a whole and does not operate with the detailed information on the designs which is required during the design process.

Kristall, **ARBAT**, **Decor**, **COMET**, **ZAPROS**, and **ComeIn** enable to perform various checks of the bearing structural members for compliance with the current codes. They work autonomously, but they are related to **SCAD** by the approach to the problem and some common functions. These programs were not developed for obtaining design documentation, they only enable to perform the detailed checks of the given structure (it should be noted that design codes — SNiP — were developed as verification systems).

Bearing in mind that these programs can be used not only by an experienced designer, but also by a beginner engineer, the developers tried to create such software products that could prevent the user from skipping any of the numerous checks given in the design codes. In particular, the set of designs is dictated, among other things, by the fact that only the designs with all the requirements completely defined by the design codes are considered. If at least one of them could not be accurately determined (for example, the check of stability of in-plane bending of an angle in **Kristall**), then such designs were either excluded completely or the list of possible modes of the programs was limited for them. In this sense, the programs act as a thorough expert.

Thus, the set of main ideas underlying the development of programs can be formulated as follows:

- the user should be sure that the program will conduct an exhaustive and rigorous examination in accordance with the requirements of the design codes and only the provisions of the codes will be implemented in it, and other (possibly very reasonable) design ideas and procedures will not be applied;
- the user should not have to search for the basic reference and regulatory information, it must be available in the program database;
- the user should be given the opportunity to analyze the results of the examination in detail and the right to make decisions on changing and improving the design.

Monolit enables you to design monolithic ribbed slabs, and unlike other programs, it generates a set of design documentation in the result. As the initial data, this program can use the results of selection of the reinforcement in the elements of reinforced concrete structures, obtained using either SCAD or other programs. Design rules implemented in the program are taken from several sources. These are mainly SNiP, design manuals and guides, but some solutions that do not contradict the codes are suggested by experienced designers.

WeST enables to determine loads and actions on building structures and can be useful both when using general-purpose computing systems, and such programs as **Kristall**, **ARBAT**, and **ComeIn**. This is the reason it took an “honorable first place” in this book.

Auxiliary programs intended for preliminary determination of design parameters also include **CROSS**, which enables to calculate Winkler coefficient using the results of geological surveys.

1.2 Direct and Inverse Problems

Design codes are developed as a system of checks of the known design, i.e. they solve the problem of structural assessment rather than the problem of its synthesis. **Kristall**, **Comet** and **ARBAT** enable to solve both these problems — assessment task and selection task. However, the latter problem (selection) is solved in a limited formulation as a purposeful search through the list of possible designs.

Thus, when designing steel structures in **Kristall** sections with the lowest steel consumption are selected from the specified assortment of profiles.

The described approach to the selection (synthesis) of the cross-section leads to designs that for one reason or another (design considerations, unification, etc.) may not satisfy the designer. He can adjust the design proposed by the program and perform its verification in the check mode.

1.3 Load-bearing Capacity Area of Sections and Structural Assessment

Standard requirements (conditions of strength, general and local stability, slenderness, etc.) to a certain design section of a structure can be written in the form of a certain system of inequalities, each of which depends functionally on the values of internal forces $\vec{S} = \{S_1, S_2, ..., S_n\}$ that can arise in the considered section from the action of the design combinations of loadings:

$$\mathbf{\Phi}\left(\vec{S}, \vec{R}\right) \le 1;$$

or

$$\begin{cases} f_1\left(\vec{S}, \vec{R}\right) \le 1; \\ f_2\left(\vec{S}, \vec{R}\right) \le 1; \\ \quad ... \\ f_j\left(\vec{S}, \vec{R}\right) \le 1; \\ \quad ... \\ f_m\left(\vec{S}, \vec{R}\right) \le 1; \end{cases}$$

where n is the total number of possible internal forces in the section; m is the number of inequalities that describe the standard requirements; f_j is a function of principal variables that implements j-th check; $\vec{R}$ are generalized resistances.

Based on the values of functions $K_j = f_j\left(\vec{S}, \vec{R}\right)$, we can introduce the concept of a **utilization factor of restrictions**, and thus represent the analysis criterion as:

$$K_{\max} = \max\left\{K_j \,|\, j = 1, ..., m\right\} \le 1,$$

where K_j is the left-hand side of the design inequality $f_j\left(\vec{S}, \vec{R}\right) \le 1$,

where all required analyses are included. The value of K_j itself will define a reserve of strength, stability, or another regulated quality parameter available for a particular element (joint, connection, cross-section etc.). If the requirement of the design code is met excessively, then K_j is equal to the relative value of exhaustion of the design requirement (for example, K_j = 0,7 corresponds to the reserve of 30%). If the design requirements are not met, the value of $K_j > 1$ indicates a violation of a requirement, i.e. it describes a degree of overloading.

Each standard requirement $f_j\left(\vec{S}, \vec{R}\right) \le 1$ defines a certain area $\mathbf{\Omega}_j$ in the n-dimensional space of internal forces, and the intersection of all areas $\mathbf{\Omega}_j$ forms the load-bearing capacity area of the section $\mathbf{\Omega}$ in terms of the considered codes (Fig. 1.3-1). The maximum utilization factor of restrictions for each point of the load-bearing capacity area of the section is $K_{\max} = \max\left\{K_j \,|\, j = 1, ..., m\right\} \le 1$.

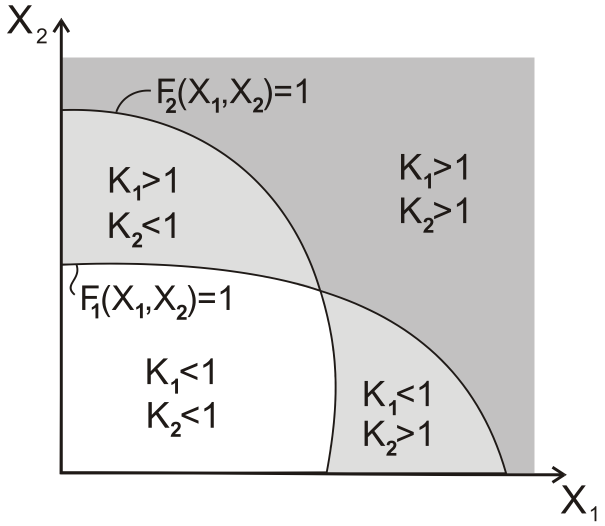


Figure 1.3-1. Formation of the load-bearing capacity area in the two-dimensional space of internal forces

Regulatory restrictions are usually written as follows:

$$\varphi_j\left(\vec{S}\right) \le \psi_j\left(\vec{R}\right),\ (j = 1, \ldots, m),$$

where φ_j, ψ_j – functions of the main variables which implement the *j*-th check.

However, not all these requirements can or need to be rewritten in the form:

$$f_j\left(\vec{S},\vec{R}\right) \le 1,\ (j = 1, \ldots, m), \text{ where}$$

$$f_j\left(\vec{S},\vec{R}\right) = \frac{\varphi_j(\vec{S})}{\psi_j(\vec{R})},$$

i.e. **in the form of the ratio of the left-hand side of the standard inequality to its right-hand side**.

To illustrate this let's consider formula (8.78) of SP 63.13330 as an example:

$$T \le T_0\sqrt{1-\left(\frac{M}{M_0}\right)^2}.$$

If the following ratio was calculated as the value of the utilization factor of this standard check:

$$\frac{T}{T_0\sqrt{1-\left(\frac{M}{M_0}\right)^2}} \le 1,$$

then in the design situation when $M > M_0$, we would obtain a root from a negative number in the denominator, and would not be able to obtain a quantitative estimate of the excess of the bearing capacity, which is avoided by transforming the formula (8.78) as follows:

$$\left(\frac{T}{T_0}\right)^2 + \left(\frac{M}{M_0}\right)^2 \le 1.$$

Another example is the formula (8.10) of SP 63.13330

$$N \cdot e \le R_b \cdot b \cdot x(h_0 - 0{,}5x) + R_{sc} \cdot A'_s\ (h_0 - a'),$$

for which the ratio of the left- and right-hand sides can not be used as a factor, because the height of the compressed area x depends on the longitudinal force.

The numerical value of the factor (the value of the utilization factor of restriction) **is a measure of how fully the bearing capacity of the structural member is used (or how much it is exceeded)**, and as a result it allows the designer to make a correct decision about the type of the necessary design modification **and only!** For example, it hardly makes sense to select another steel grade with a greater design strength in the case when the stability check turned out to be critical.

It should be specially emphasized that there may be no direct proportionality between the value of the factor and the values of the forces appearing in the standard check, as there may be no direct proportionality between the value of the factor and the values of the geometric properties of the cross-sections of the structural elements. This is due to the **non-linearity of the standard checks**, in particular, the non-linearity of the function of the buckling coefficient, etc.

All values of the K_j factors obtained by analysis are given in the **Factors Diagram** dialog box (Fig. 1.3-2) or in a full report document. The respective dialog boxes display the value of K_{max} – the maximal (i.e., the most dangerous) among the detected K_j values and indicate the type of check (e.g., strength, stability) in which this maximum took place.

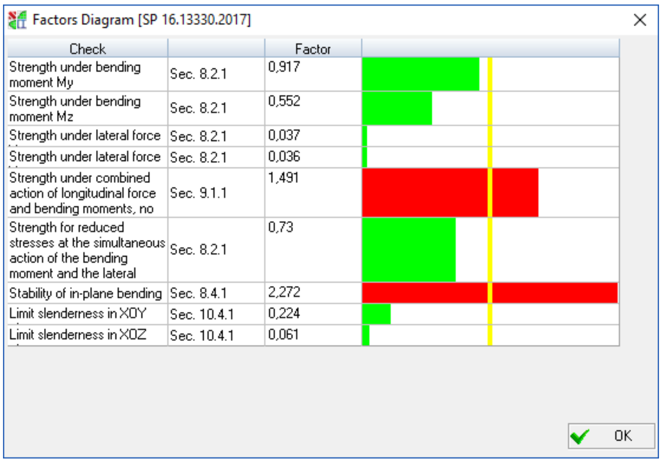


Figure 1.3-2. An example of the **Factors Diagram**

The data given in the diagram of factors enable the designer to make a correct decision on the type of necessary structural modifications. For instance, it hardly makes sense to increase the design strength of steel if the stability check turned out to be the critical one.

1.4 Generation of the Load-Bearing Capacity Area and Its Properties

ARBAT, **Kristall**, **Décor,** and **ComeIn** enable to generate the load-bearing capacity areas for sections of load-bearing bar elements of reinforced concrete, steel, timber, and masonry structures, respectively, and **Comet** enables to generate them for joints of steel structures.

Sections of bar elements, where six internal forces (longitudinal force, bending moments, transverse forces, and torque) can appear under the load, have a load-bearing capacity area in the form of a six-dimensional geometric object which is very difficult to analyze. The best way to display the load-bearing capacity area of sections is by performing its orthogonal projection onto a certain plane (pair) of internal forces. The generation of a two-dimensional projection of the load-bearing capacity area of a section is performed according to the algorithm given below.

The user selects a pair of internal forces (for example, a pair “longitudinal force N – bending moment M_y”), in the coordinate system of which an orthogonal projection of the load-bearing area will be generated. The remaining internal forces in the section (M_z, Q_y, Q_z, M_x) are fixed at a certain level (they are specified by the user or take zero values). At a certain fixed value of the ratio $e = M_y/N$, the point closest to the origin is sought, where a certain inequality from the system $\mathbf{\Phi}(\vec{S},\vec{R}) \le 1$ takes a limit value $f_j(\vec{S},\vec{R}) = 1$. This point belongs to the boundary of the two-dimensional orthogonal projection of the load-bearing capacity area of the section.

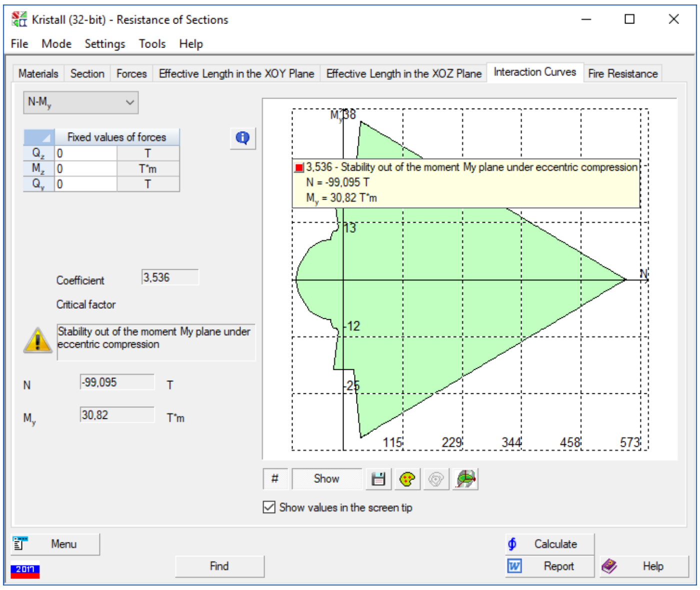


Figure 1.4-1. Interactive mode of exploring the load-bearing capacity area of the section

This method of generating a load-bearing capacity area assumes that the area is starlike, which is a hypothesis, on the one hand, and a restriction of the software implementation, on the other hand.

Moreover, the generated area is an interactive tool for communicating with the user. Using your cursor, you can examine a two-dimensional projection of the area. A certain set of internal forces corresponds to each position of the cursor. Their values are displayed in the respective fields. Depending on the change in the position of the cursor (changes in the respective pair of internal forces), the maximum value of the utilization factor of standard restrictions-inequalities $K_{\max}$, corresponding to these forces is output, as well as the type of the inequality for which it is calculated (Fig. 1.4-1). Clicking the right mouse button on the area enables to see the entire list of performed checks and the values of utilization factors of restrictions $K_j (j = \overline{1,m})$ for the set of internal forces that corresponds to the cursor position. Moreover, the following operations with the mouse can be performed in the program:

- scrolling the wheel changes the scale of the image;
- clicking and holding the middle mouse button (wheel) while dragging the cursor moves the image.

One of the most important properties of the load-bearing capacity area is its convexity. It should be noted that it is the convexity of the load-bearing capacity area of the cross-section that gives us the right to limit ourselves in the linear calculation to the checks of this section for the action of only those combinations of internal forces in the cross-section for which the extreme (minimum or maximum) values are characteristic. The positive result of such checks automatically means that all other conceivable combinations of loads will be acceptable.

The absence of the convexity property of the load-bearing capacity area of the considered section can lead to many unpleasant consequences related to the fact that, traditionally, evaluating unfavorable combinations of internal forces, engineers either do not consider some actions at all (in the case when they have a unloading effect) or take them fully into account. This rule is entirely valid for a convex load-bearing capacity area, while for a nonconvex area a combination with intermediate (not extreme) values of internal forces can turn out to be an unfavorable one.

1.5 Load-bearing Capacity Area of Sections as a Tool for the Analysis of Codes

Several thousand calculations are performed during the computer-aided generation of the load-bearing capacity area of a section, which is apparently the largest check of the considered section. Moreover, the shape of the load-bearing capacity area of the section as well as the character of its boundaries in many cases enables to perform a more detailed analysis of the requirements of the codes than it can be done in other ways. The analysis of the shape of the area enables to check the consistency and completeness of the standard requirements. In this case, it is easy to identify the inconsistency of certain provisions of the codes, in particular the non-smoothness of the transition between the approximations used.

SP 16.13330 Steel Structures

For example, let's consider the design code for steel structures SP 16.13330. We will generate the load-bearing capacity area for a cross-section in the form of a symmetric welded I-beam with a 400×10 mm web and 200×10 mm flanges made of steel with the design strength R_y = 2050 kg/cm^2. The effective length of the bar in both principal planes of inertia is 600 cm, the service factor and the importance factor are taken as γ_c = 1,0 and γ_n = 1,0. The load-bearing capacity area $\mathbf{\Omega}_{SNiP}$ of this section in accordance with the codes SP 16.13330 is shown in Fig. 1.5-1.

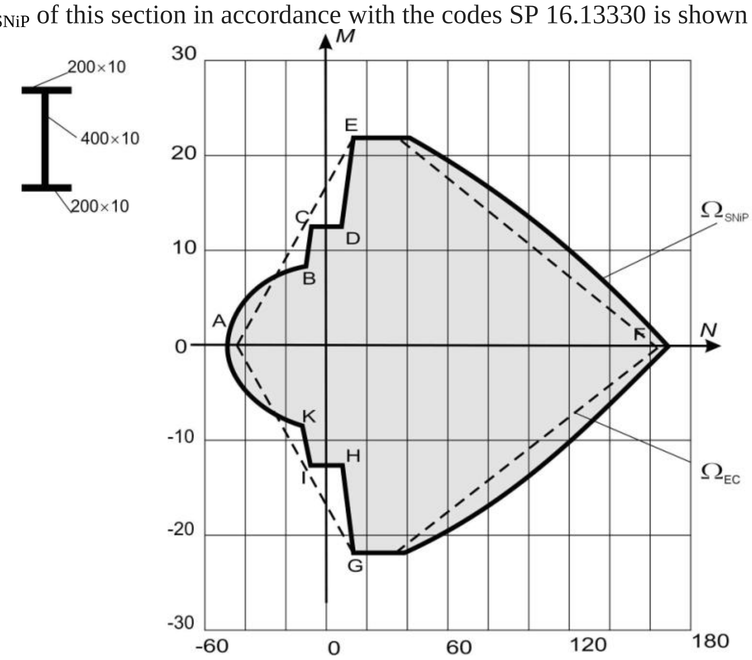


Figure 1.5-1. Load-bearing capacity area of a steel section:
$\mathbf{\Omega}_{SNiP}$ – according to SP 16.13330; $\mathbf{\Omega}_{EC}$ – according to EN 1993-1-1:2005

The boundary of the load-bearing capacity area $\mathbf{\Omega}_{SNiP}$ on the *DEFGH* section is defined by the strength condition under the combined action of tension and bending, on the *CD* and *IH* sections – by the condition of stability of in-plane bending, and on the *IKABC* section – by the condition of stability out of the bending moment plane.

The non-convexity of the boundary of the load-bearing capacity area $\mathbf{\Omega}_{SNiP}$ on the *IKABC* section is related to the change in the type of dependence of the coefficient *c* on the value of the relative eccentricity *m*. This coefficient is included in the condition for checking the stability out of the plane of bending of a bar under bending and compression. The graphs $c = \boldsymbol{c}(m_x)$ for three values of the effective length of the considered bar out of the bending plane are shown in Fig. 1.5-2, which shows a characteristic break at the value of the relative eccentricity m_x

= 10, where the function $\boldsymbol{c}(m_x)$ changes from a linear to a hyperbolic one. The indicated break corresponds to the points *K* and *B* of the load-bearing capacity area $\boldsymbol{\Omega}_{SNiP}$ (Fig. 1.5-1).

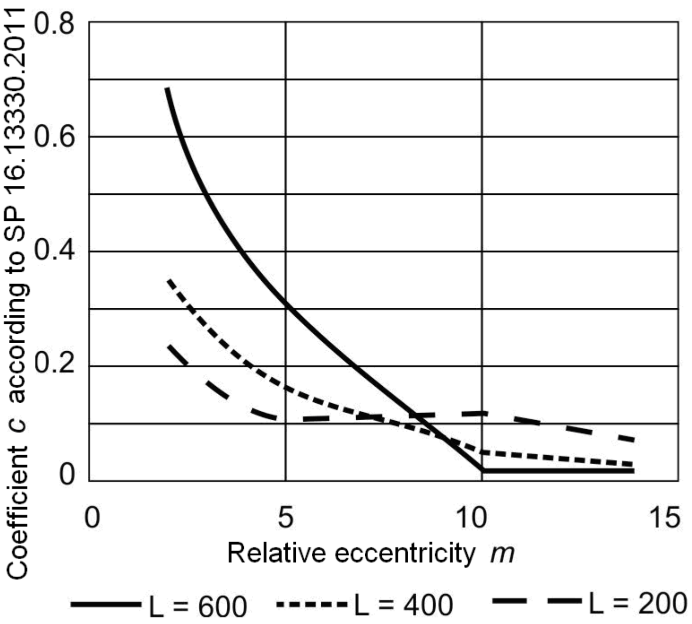


Figure 1.5-2. Dependence of the coefficient *c* on the relative eccentricity *m*

It should be noted that the nonconvexity of the *IKABC* section of the load-bearing capacity area $\boldsymbol{\Omega}_{SNiP}$ does not appear when the element has small out-of-plane slenderness, in spite of the fact that the break in the curve $c = \boldsymbol{c}(m)$ does not disappear, but for such design cases the condition of stability out of the bending moment plane is not determinative.

The configuration of the *CDE* and *IHG* sections of the load-bearing capacity area $\boldsymbol{\Omega}_{SNiP}$ is determined by the codes specifying that the stability of in-plane bending of a bar should be checked only at the values of the relative eccentricity $m_x > 20$, when the stability check of such a bar has to be performed as for a flexural member. Sections of the boundary *DC* and *JK* of the load-bearing capacity area $\boldsymbol{\Omega}_{SNiP}$ correspond to these values (Fig. 1.5-1).

EN 1993-1-1:2005. Eurocode 3. Design of Steel Structures

The dashed line in Fig. 1.5-1 shows the load-bearing capacity area $\boldsymbol{\Omega}_{EC}$ of a cross-section calculated according to the requirements of EN 1993-1-1:2005. The load-bearing capacity area of this section is convex, because the section operates within the limits of elastic deformation of steel.

Fig. 1.5-3 shows the load-bearing capacity area of a steel I-beam with a 800×10 mm web and 360×20 mm and 240×20 mm flanges generated in accordance with the requirements of EN 1993-1-1:2005. Here the nonconvexity of the load-bearing capacity area of the section is related to the requirements of EN 1993-1-1: 2005, concerning the classification of sections. In the stress state corresponding to the appearance of compressive stresses in one of the flanges, the section ceases to be classified as a section of the 2nd class (plastic deformations of steel, calculation using the plastic moment of resistance) and passes into the third class of sections (elastic deformations of steel, calculation using the elastic moment of resistance). The jumps AB and CD correspond to these transitions. A similar jump-like change in the boundary of the load-bearing capacity area can occur at the transition of the section from the 3rd class to the 4th one.

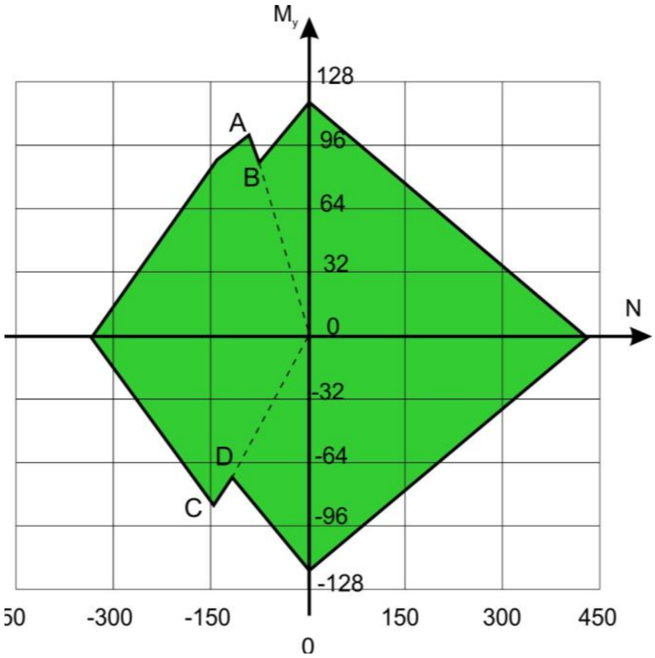


Figure 1.5-3. Load-bearing capacity area of a steel section according to EN 1993-1-1

SP 63.13330 Reinforced Concrete Structure

Let's consider a rectangular reinforced concrete section 700 mm × 200 mm, reinforced by 2 rebars ∅ 28 mm. The areas of the load-bearing capacity of this section corresponding to two variants of the standard crack resistance requirements are given in Fig. 1.5-4 in the "longitudinal force N – bending moment M_y" axes. The area $\mathbf{\Omega}_1$ is generated taking into account the regulatory restriction of the crack opening width, and the area $\mathbf{\Omega}_2$ is generated taking into account the prohibition of crack formation. It should be noted that both methods of the standard check of reinforced concrete sections for crack resistance are semi-empirical. Let's consider the causes of the non-convexity of the load-bearing capacity area of a reinforced concrete section.

The jump *AB*, which is typical for both areas of the load-bearing capacity $\mathbf{\Omega}_1$ and $\mathbf{\Omega}_2$, is defined by the postulate of the theory of reinforced concrete, which provides for the immediate exclusion from the operation of the tension area of concrete when tensile stresses appear in it.

The jump *BC* is typical only for the load-bearing capacity area of the reinforced concrete section $\mathbf{\Omega}_2$, generated taking into account the prohibition of crack formation. In this case, the area $\mathbf{\Omega}_2$ sharply narrows, which is quite expected and, moreover, it acquires a very sophisticated shape, due to a new method of checking reinforced concrete elements for crack resistance, which appeared in the 2004 edition of the codes.

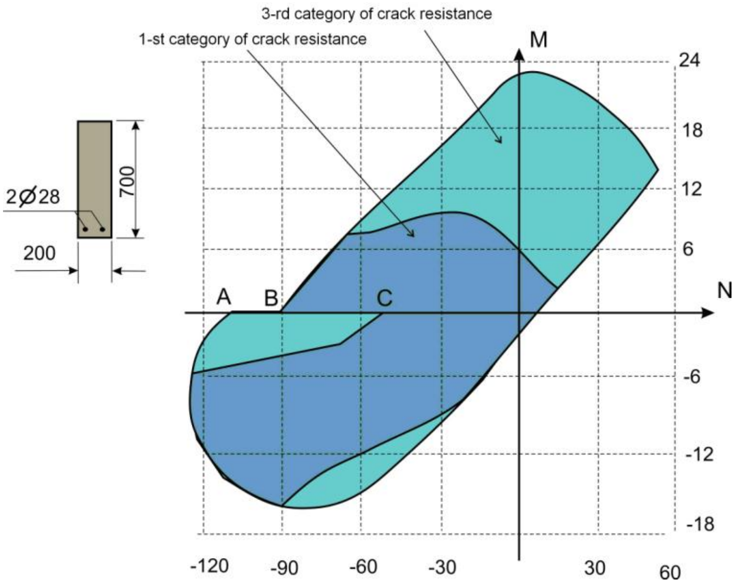


Figure 1.5-4. Load-bearing capacity areas of a reinforced concrete section according to SP 63.13330:
Ω_1 – restriction of the crack opening width (3-rd category of crack resistance);
Ω_2 – prohibition of crack formation (1-st category of crack resistance)

Another instructive example is given in Fig. 1.5-5, which shows the load-bearing capacity area of a reinforced concrete section calculated according to a non-linear deformation model without taking into account (variant *a*) and taking into account (variant *b*) the random eccentricity. The eccentricity is obviously taken into account "unphysically" in the variant given in the codes, since there is no physical model that would violate the smoothness of the boundary of the load-bearing capacity area. It is specified in the codes that for a statically indeterminate system the value of the eccentricity of the longitudinal force with respect to the center of gravity of the reduced section e_0 should be taken equal to the eccentricity value e obtained from the static analysis but not less than a certain fixed value of the random eccentricity e_a; and for a statically determinate system, the eccentricity e_0 should be taken equal to the value (e + e_0). In both cases, when the eccentricity values e are small, an eccentrically compressed element is considered, and the eccentricity e_0 is never small.

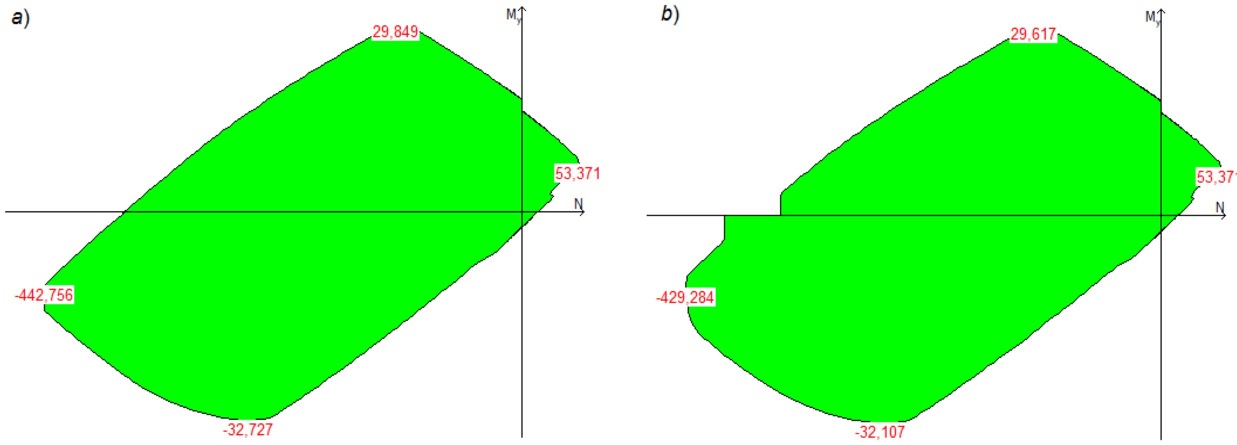


Figure 1.5-5. Effect of random eccentricity on the shape of the projection of the load-bearing capacity area of a reinforced concrete section

SP 64.13330.2011 Timber Structures

Fig. 1.5-6 shows the load-bearing capacity area of a timber section (3 m long pine bar with a rectangular cross-section 500×80 mm). The jump at the transition from the area $N > 0$ to the area $N \leq 0$ is defined by the formulations of Sec. 6.14 and 6.20 of the codes, which provide the stability check requirements only for elements under bending or under bending and compression. The codes do not consider this check for bar elements under tension and bending. However, it is obvious that if the tensile force values are very small, this check must be performed. In this case, the boundary of the load-bearing capacity area of the section would pass along the dotted curve in Fig. 1.5-6.

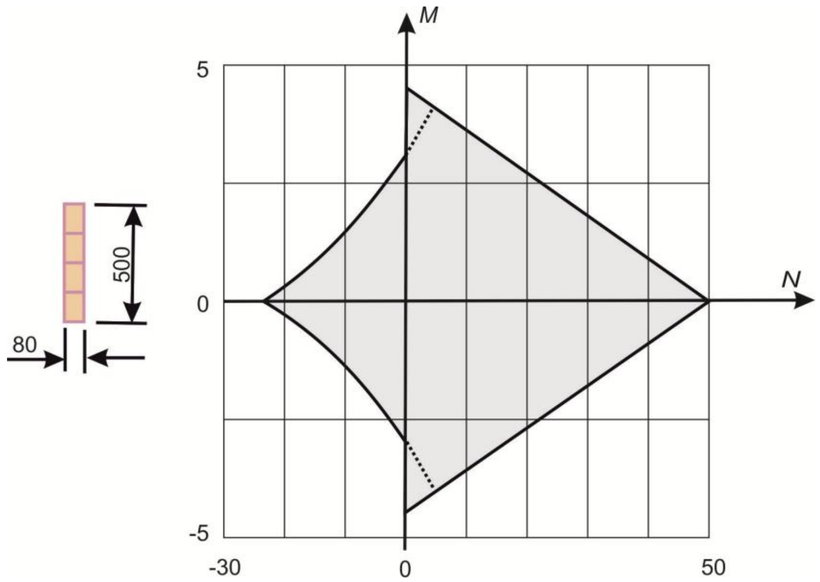


Figure 1.5-6. Load-bearing capacity area according to the codes for timber structures

SNiP II-22-81. Masonry and Reinforced Masonry Structures

Similar problems related to the non-convexity of the bearing capacity area arise also at the analysis of the behavior of elements of masonry and reinforced masonry structures. The load-bearing capacity area of the sections of bar elements of masonry and reinforced masonry structures can be non-convex. Masonry structures, like reinforced concrete ones, resist tension and compression in different ways. In this regard the possibility of transition to various limit state models is implemented stepwise in the design codes.

In particular, in accordance with Sec.4.20 of SNiP II-22-81, the analysis of the unreinforced masonry for shear is performed as follows:

- if the eccentricity of the longitudinal force does not go beyond the section core, — taking into account the entire cross-sectional area A;
- if the eccentricity exceeds the section core size, — taking into account only the compression area A_c.

As a result, a break occurs on the boundary of the bearing capacity area which leads to the loss of convexity (point *Q* in Fig. 1.5-7, *a*).

a b

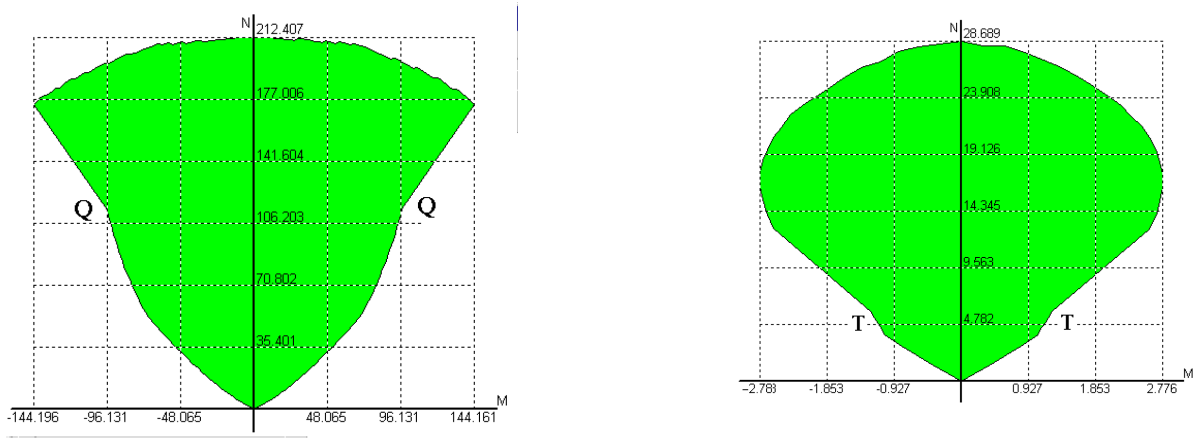


Figure 1.5-7. Load-bearing capacity areas of masonry structures (example)

Another factor leading to the non-convexity of the load-bearing capacity area of masonry structures is related to the instruction of Sec.5.3 of the mentioned SNiP II-22-81, which requires to perform a check for the masonry joint opening only at the values of the eccentricity of the longitudinal force with respect to the center of gravity of the section $e_0 > 0.75y$. As a result, a break occurs at the point *T* (Fig. 1.5-7, *b*).

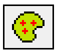
The number of examples could be increased, but the ones given here indicate a very real situation when the load-bearing capacity area can turn out to be non-convex. As the analysis shows, in many cases there are some inconsistencies in the formulation of requirements for the bar elements of load-bearing structures, due most likely to insufficient adjustment of the formulations themselves.

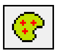
The origin of such inconsistencies is related to the fact that the traditional approach based on the manual calculation generated all sorts of “simplifications”, which allowed to skip some checks or to replace the general case with a certain particular case (as at $m < 20$ for steel elements under bending and compression).

Moreover, the use of “logical switches” that change rules without an exact physical basis leads to an abrupt change in the algorithm, as in the classification of cross-sections in the Eurocode. Modern technologies enable to detect such inaccuracies and determine ways to improve the codes.

1.6 Load-bearing Capacity Area as a Tool for Analyzing Loading Conditions

Dangers related to the nonconvexity of the load-bearing capacity area point to the necessity of analyzing the closeness of the set of the specified combinations of forces to the section of the boundary of the load-bearing capacity area, where the property of nonconvexity is manifested.

An almost identical analysis can be performed using additional tools provided by **SCAD Office**. The following buttons are provided for the analysis of problems related to the non-convexity of the load-bearing capacity area: . They enable to perform the following operations:

 – if the forces are specified, clicking this button will draw the entire set of given combinations of forces as points corresponding to the projections of the variants of combinations of forces on the plane of the selected pair of forces (Fig. 1.6 -1).

 – drawing a convex hull of the points specified above, i.e. an entire set of points which may result from a linear combination of specified forces, including their incomplete values.

Despite the fact that these combinations were not subjected to a direct check, in the case when the convex hull of loadings does not leave the load-bearing capacity area of the section, it can be ensured that the various loadings combined from the basic ones are not dangerous.

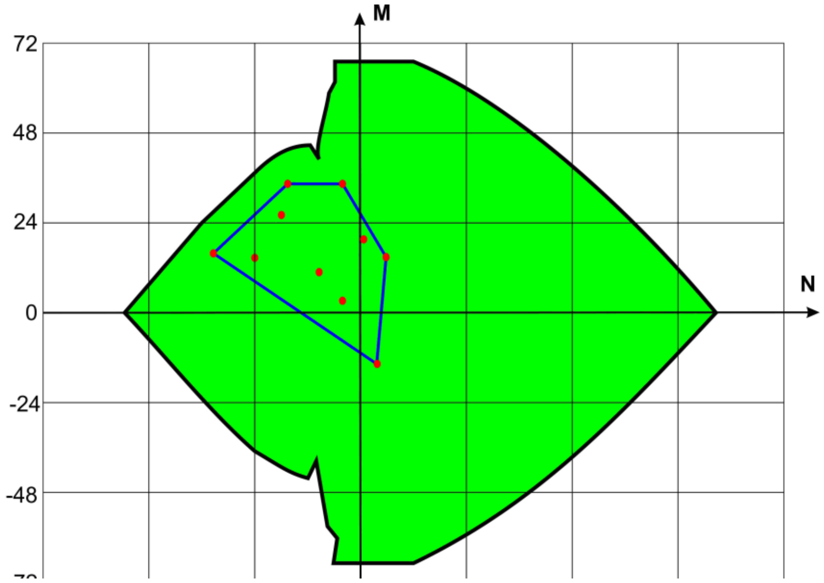


Figure 1.6-1. Given (basic) loadings and their convex hull, combined with the load-bearing capacity area of the section

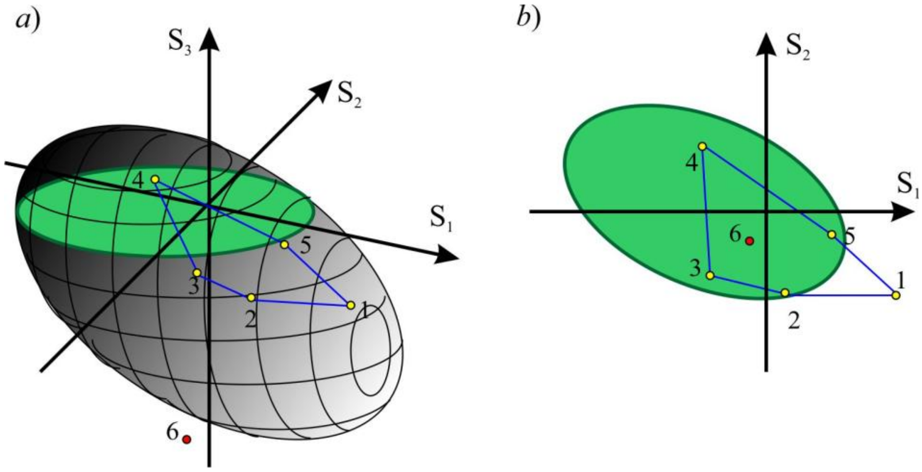


Figure 1.6-2. Illustration of possible design situations

It should be noted that the considered mechanism is a powerful tool for analyzing loading conditions, but it should be used carefully.

The load-bearing capacity area of a section is a body in the six-dimensional space of internal forces (with the coordinates N, M_y, Q_z, M_z, Q_y, T). The load-bearing capacity area is a section of the given body by a plane, and the point corresponding to the set of forces is the projection onto this plane. If the point corresponding to a certain set of forces lies within the load-bearing capacity area, it does not follow from this that all the requirements of the codes are satisfied. This is due, for example, to the fact that when generating the load-bearing capacity area (interaction curves), the restrictions of the slenderness are ignored (these restrictions do not depend on forces), and a situation may occur when the point belongs to the load-bearing capacity area of the section (lies in the "green area"), but the calculation will show $K_{max} > 1$.

When operating only with two-dimensional orthogonal projections of the load-bearing capacity area, it is possible to "see" the projections of some points (combinations of internal forces) belonging to this area (for which the maximum utilization factor of restrictions does not exceed one $K_{max} < 1$) as displayed outside the projection boundary of the load-bearing capacity area (as, for example, point 1 in Fig. 1.6-2). This can happen in the case when the **Seismic** checkbox is checked for the specified set of combination of internal forces, and as a result the service factor $m_{cr} > 1$ was used in the calculation. And the generation of the load-bearing capacity area (interaction curves) is performed without taking into account factor m_{cr}. A more complicated situation can arise when analyzing the interaction curves for reinforced concrete structures, since different sets of forces (points) may have different coefficients of duration.

There may also be an erroneous "vision" of a different kind, when the projection of the point lies within the boundaries of the projection of the load-bearing capacity area, and the point itself does not belong to the area (see, for example, point 6 in Fig. 1.6-2).

In order to identify such situations, the projections of the points in which the utilization factor of restrictions exceeds one, are displayed in red on the projections of the load-bearing capacity area, otherwise they are displayed in green.

1.7 Interaction Surfaces

In most programs the *Interaction Curves* tab has a button , which enables to generate a surface in 3D where K_{max}=1 for the triple of internal forces and moments selected by the user (for example, M_y-M_z-N). Clicking this button invokes the dialog box (see Fig. 1.7-1), where you can select which force corresponds to a certain coordinate axis (XYZ).

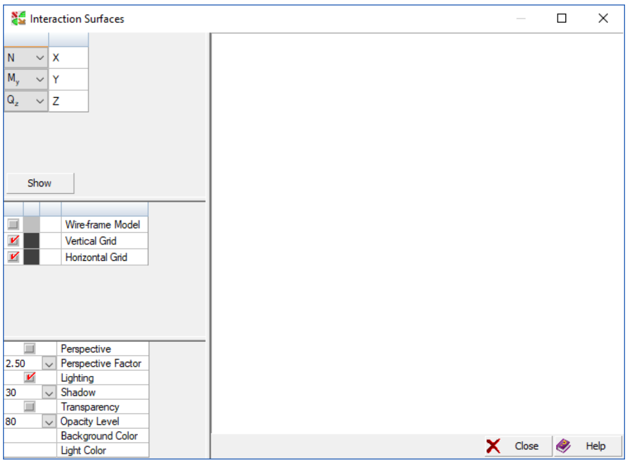


Figure 1.7-1. *The* **Interaction Surfaces** *dialog box*

Clicking the **Show** button will start the calculation which can be aborted by clicking the **Cancel** button. Once the calculation is completed, the image of the interaction surface will appear (see Fig. 1.7-2)

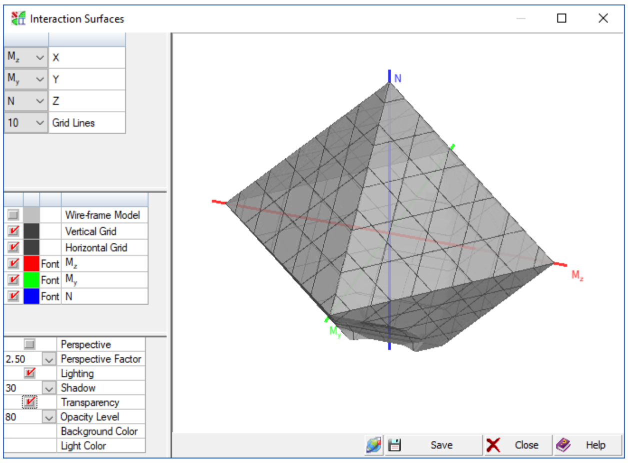


Figure 1.7-2. Interaction surface

Controls in the left side of the window enable to

- select the density of the mesh applied to the surface;
- change the colors used for drawing (as well as the fonts for the names of the axes);
- not to display vertical and/or horizontal meshes on the surface;
- set the transparency and/or lighting mode;
- display geometric perspective and select the distortion factor;
- change the background color and the color of the lighting.

In order to change the color or font, double click on the respective cell and select the parameters.

The **Wireframe model** checkbox enables to obtain an image of the surface in the form of a wireframe (see Fig. 1.7-3).

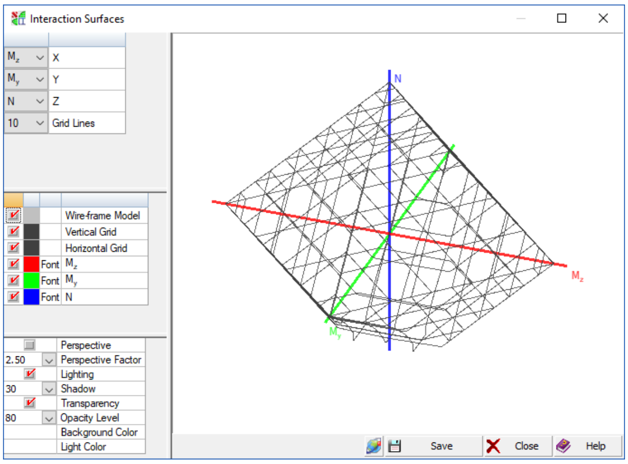
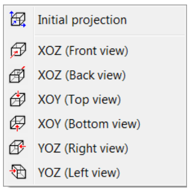


Figure 1.7-3. Interaction surface (wireframe model)

Right-clicking invokes the context menu where you can select a projection:



The following operations with the mouse can be performed in the program:

- scrolling the wheel changes the scale of the image;
- clicking and holding the middle mouse button (wheel) while dragging the cursor moves the image;
- clicking and holding the middle mouse button (wheel) together with *Shift* or *Ctrl* key while dragging the cursor rotates the surface (it is also possible to use button for switching to the rotation mode).

The **Save** button enables to save the forces for which K_{max}=1 in a text file (this file can later be imported into other programs for further analysis).

1.8 Safety Codes

All design modes require data on the importance factor according to the selected safety codes (GOST 27751-88, GOST 27751-2014, DBN B.1.2-14-2009, …).

The following importance factors are used in the analysis:

- ultimate limit state importance factor γ_n;
- serviceability limit state importance factor;
- importance factor for accidental/special design situations (it is also used in the fire resistance analyses).

The factor γ_n can be selected from a drop-down list in accordance with the characteristic of a structure (it can be of *special*, *essential* or *limited* importance) or entered by the user (in non-standard situations).

DBN B.1.2-14-2009 provides three importance factors (for the ultimate limit state, for the serviceability limit state and for accidental situations). The user can specify each of these three factors. GOST 27751-88 provides only one factor, which is used in all design situations. If GOST 27751-2014 is selected as a safety code, the user-defined factor γ_n is used in the ultimate limit state analyses. The following value is assumed for the serviceability limit state analyses γ_n=1.0 (but not greater than γ_n for the ultimate limit state in order to avoid a situation in which the characteristic values turn out to be greater than the design ones). And the following value is used for the analysis of special design situations γ_n =1.0 (see Sec. 5.5 of SP 296.1325800.2017).

1.9 Seismicity

You can select the design codes regulating the construction in seismic regions in the main window of **Decor**, **ARBAT**, and **Kristall**. Moreover, you can check the **Seismic** checkbox when specifying forces in some modes, for example, *Resistance of Sections*, *Truss Member* (**Kristall** or **Decor**), *Bolted Connections*, *Friction Connections* (**Kristall**). In this case the service factor m_{cr} specified in the seismic codes will be additionally used in the calculation.

1.10 References

[1] V.Karpilovsky, E.Kriksunov., A.Maliarenko, M.Mikitarenko, A.Perelmuter, M.Perelmuter, SCAD Office System SCAD. — M: ASV Publishing House, 2007.— 592 p.

2. Interface and Controls

2.1 Controls

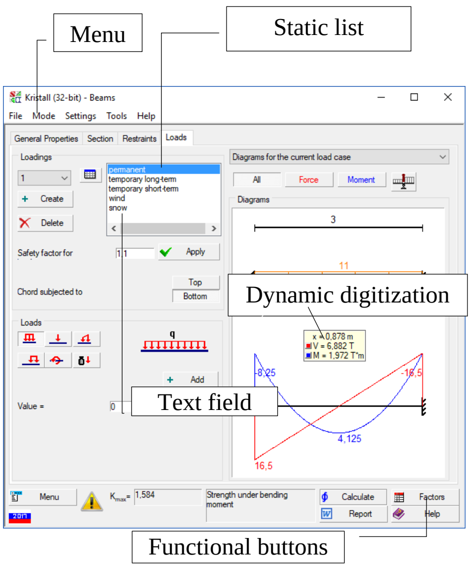


Figure 2.1-1. Controls

Principles and means of control implemented in the software are uniform and provide a consistent interactive environment. Programs use a common multi-tab technique. To activate a window, click on its tab. Besides, there are following controls and means for accessing information (see Fig. 2.1-1):

- menus that open when left-clicked, in the same way as those of any MS Windows application[1];
- functional buttons, “clicking” which is used to invoke particular functions or modes; to do it, place the mouse pointer over a button and left-click;
- radio buttons and checkboxes which enable you to make a choice from a predefined set of options;
- text fields used to enter initial data for the analysis. Initial data consist of numbers. If the entered number is not an integer, its integer and fractional parts are separated by a period or another separator. The separator is assigned by the user during the customization of the operating system (see **Settings** | **Regional Settings** | **Number**). Moreover, the numbers can be entered in scientific notation, for example, 1.56e-7 (when specifying numerical data, you can enter not only numbers, but also simple arithmetic expressions, for example, 0.9*190.8+4.5);
- drop-down or static lists for selecting data;
- tables for entering or displaying information in a tabular form;
- dynamically digitized diagrams that display values of functions for an argument pointed to by the mouse pointer on the screen.

[1] Microsoft, Windows are registered trademarks of Microsoft Corporation. All other trademarks mentioned here are properties of respective companies.

2.2 Settings

The **Application Settings** dialog box is invoked by the **Settings** button of the main window or from the respective menu. It can be invoked at any moment when working with the program. It is used to customize general parameters of the program. The dialog contains the following tabs: **Units of Measurement, Report and Languages**, **Visualization**, **Sections**, and **General**.

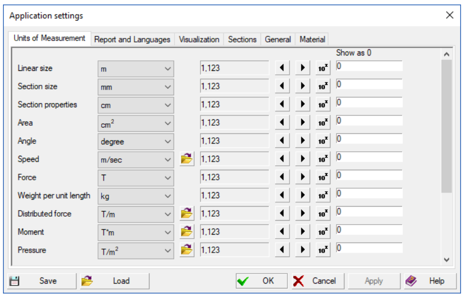


Figure 2.2-1. *The* **Units of Measurement** *tab*

Each tab opens a page where you can adjust certain types of settings.

The **Units of Measurement** tab (Fig. 2.2-1) enables you to define units of measurement used in the analysis. It contains two groups of data. The first group is used to specify measurement units of linear sizes, forces, moments, etc.

For compound units (such as those of moment, pressure, etc.), there is a possibility to define their component units (such as those for force and for moment arm) separately using the button .

The second group helps to choose a representation and precision of numerical data. Special controls are used here to select data representation formats. Make sure to specify the number of significant digits in either the fixed-point decimal representation or the floating-point scientific notation.

The precision of the data representation (the number of significant digits after the decimal point) can be assigned using the (decrease) and (increase) buttons, while the scientific notation is turned on by the button . You can also specify in respective text fields which values should be treated as negligibly small, so that all absolute values less than the given ones will be displayed as 0 in all visualizations.

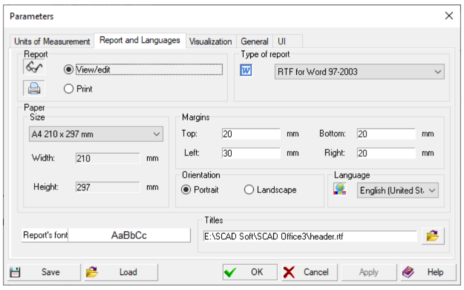


Figure 2.2-2. *The* **Report and Languages** *tab*

The **Report and Languages** tab (Fig. 2.2-2) enables you to choose a language for the user interface and for the report.

There are two modes for working with a report document: **View/Edit** or **Print**.

In the **View/Edit** mode, clicking the **Report** button in any active dialog will open the report and allow you to view/edit it. An application associated with **RTF** (Rich Text Format) files (such as MS Word Pad or MS Word) will be invoked to serve this purpose.

Obviously, it is the user who is fully responsible for any changes made to the text of the report (note that even results of the calculation can be edited).

There are differences in RTF formats used by MS Word v.7, MS Word 97 (2000/XP) and Open Office. Therefore, the program allows you to choose one of the formats in the **Type of Report** mode (besides RTF a report can be created in the following formats DOC, PDF, HTML). Clicking the **Print** button in the **Report** group will print the report in the form it has been generated by the program (see Sec. 2.5).

Use the **Titles** text field to specify an RTF file containing headers and footers for pages of the report document. The file can be selected from a standard list by clicking the button .[2]

The **Paper Size** setting enables you to choose the paper format for printing the report (the size is selected from a drop-down list).

Moreover, the margins and the page orientation can be selected before generating the report.

The **Short report** checkbox enables to reduce the volume of the report document. In particular, a report generated in the **Resistance of Sections** mode will contain only the results for the load cases where a maximum value has been obtained for at least one factor.

Report Font is used to choose a font type for the report. A double left click invokes a standard Windows dialog for selecting the font. Only the selected font type will be used in the generated reports (style and size are assigned by the software).

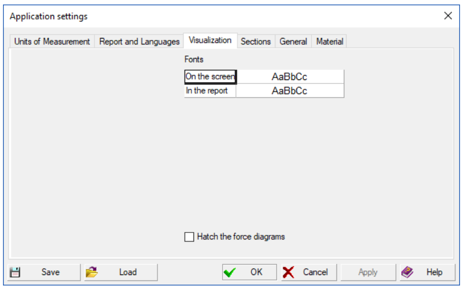


Figure 2.2-3. *The* **Visualization** *tab*

The **Visualization** tab (Fig. 2.2-3) is used to choose a font for the user interface and for the report. A double left click invokes a standard Windows dialog for selecting the font.

This tab also contains a checkbox which enables to hatch force diagrams.

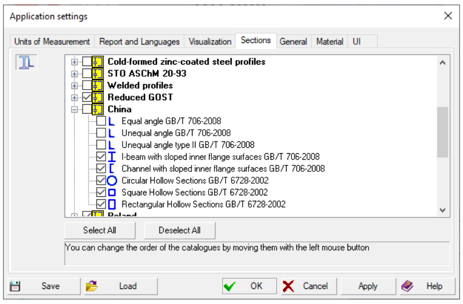


Figure 2.2-4. *The* **Sections** *tab*

The **Sections** tab (Fig. 2.2-4) enables you to browse a list of steel profile catalogues available in the application database and select some (or all) of them for further use. There is a checkbox next to each group of catalogues. If the checkbox is not checked, the respective group of catalogues will not be available in the application. Catalogues can be arranged in any convenient order (the same order will be used in the lists or the dialog boxes for the profile selection). To move an item, drag it while holding down the right mouse button.

The **General** tab allows you to activate the **Hide window when minimized** checkbox. When it is checked the window disappears from the task bar, and an icon appears in the tray area. The window can be opened from the tray area by the left click, and a context menu – by the right click.

Moreover, the **Check for a new version at startup of the program** checkbox can be activated as well. If it is active, the program will check for a new version on the company website at each startup, and it will give a respective message if it finds a new release.

[2] If you want to modify the header and footer RTF-file supplied with the software, then when using MS Word for this purpose you should remember that it is not enough just to enter a new text, you also have to use the **Tools|Language|Set Language** menu and set the language for the new text — *Russian*.

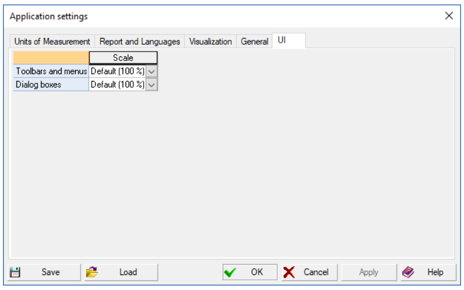


Рис. 2.2-5. *The* **UI** *tab*

The **UI** tab (Fig. 2.2-5) enables to set scales for various user interface elements:

- toolbars and menus;
- dialog boxes.

The settings provided in this tab are intended primarily for users with HiDPI displays.

Settings can be saved to an external file using the **Save** button, which can be subsequently loaded (the **Load** button).

2.3 Menus

File

The **File** menu contains two items: **Menu** and **Exit**, when it is accessed from the main window, and four items: **Menu**, **Open**, **Save As…**, and **Exit**, when accessed from the design modes.

Menu — switches from any mode to the main window;

Open — opens an existing file;

Save As… – saves the entered data to an external file. The directory and filename for saving the data are specified by the user in a standard Windows dialog;

Exit — finishes the current session.

Mode

The **Mode** menu contains a list of all modes available in the application and enables you to switch to any of them without having to go to the main window.

Settings

This menu is used to invoke the **Application Settings** dialog box where you can customize the program.

Service

When working with the application, you may need to perform some auxiliary calculations. The **Service** menu contains items for invoking a standard calculator of the Windows environment (provided it has been installed with the system), a formula calculator, and a converter of units of measurement.

Help

This menu provides the help information on using the application and on its functionality, information about the application (No. of its version and the date of the last modification), and it also contains an item that enables you to check for update.

2.4 Working with Tables

In most cases, the initial data for the analysis is specified in a tabular form (Fig. 2.4-1). The following general rules are used for working with the tabular data:

- the data are entered in the table as decimal numbers; the fractional and the integer parts of a number must be separated by a period or another separator assigned by the user during the customization of the operating system;
- 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 is used to add a new row after the currently selected one, and the latter deletes one or more selected rows;
- to select one or more successive rows, place the mouse pointer over the number of the first one, click and hold the left mouse button, and drag the pointer across the numbers of the rows to be selected;

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

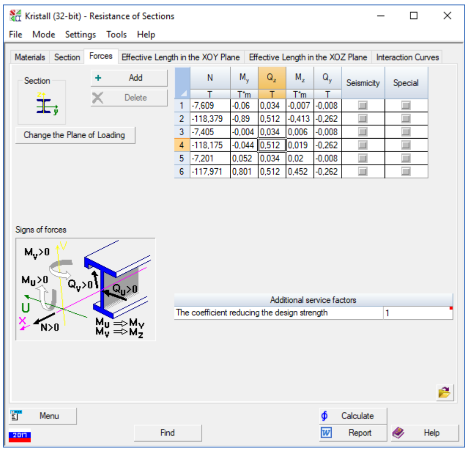


Figure 2.4-1. *Example of a table with forces*

Since the new rows are added after the selected one, 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** (or **Ctrl+C**) 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** (or **Ctrl+V**) 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 actions listed above can be also used to copy one or more selected rows of a table.

Operations with Tables

Operations with the data given in tabular form are performed with the help of a set of commands (Fig. 2.4-2) invoked by right-clicking when the cursor is in the table field.

The set of commands for the control of the table includes the following commands:

- **Copy** — copies the contents of the selected rows or cells (horizontally or vertically) to the **Clipboard**;
- **Paste** — inserts the information from the **Clipboard** into the table starting with the selected cell. If in the process of inserting it turns out that the existing rows are not enough, they are automatically added. Columns are not added, i.e. information on the right from the last column gets lost (disappears);
- **Insert row before selected row(s)** (a selected row is a row with at least one active cell) — a row is added before the selected row; all the cells of the new row will have default values (unless otherwise specified, they are equal to zero);
- **Insert row after selected row(s)** — a row is added after the selected row (the command is performed according to the same rules as the previous one);
- **Clear selected row(s)** — all cells of the selected rows are filled with default values (zero, as a rule);
- **Delete selected row(s)** — deletes all the selected rows;
- **Copy current rows** — the entire row is copied to the clipboard regardless of whether it is selected fully or partially;
- **Paste current rows** — pastes rows that were copied by the **Copy current rows** command.

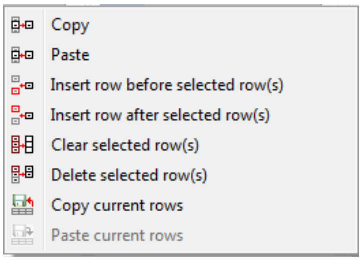


Figure 2.4-2. Set of commands for working with tabular data.

Commands (**Copy** and **Paste**) can be used not only for operations with the current table of the program but also to transfer the tabular data to other tables and applications, including those that are not a part of the **SCAD Office** system (for example, MS Excel).

2.5 Generating a Report

All the design modes have the **Report** button. Provided there are no mistakes in the initial data, clicking this button causes:

- performance of all the analyses;
- creation of an RTF (Rich Text Format) file containing a list of the initial data and results of the analysis. The file format is specified in the settings. Depending on whether the **Short report** checkbox in the **Application Settings** dialog box is checked, the report document will (or will not) contain results of certain intermediate calculations (e.g., geometric properties of the sections used, values of some (not only the maximum) load-bearing capacity utilization factors, etc.).;
- a Windows application associated with the RTF files is started. Depending on the options selected in the **Application Settings | Report and Languages** dialog box (**Print** or **View/Edit**), the application will either print the document immediately or open it for viewing and possible editing. In the latter case, it is the user who is responsible for making a hard copy (the document can be printed by using the print command of the external application).

Note. There are differences between the RTF files used by MS Word v.7, MS Word 97 (2000) and Open Office. Therefore, the file format can be selected from the **Type of report** drop-down list (besides RTF the report can be created as a DOC, PDF, HTML file).

2.6 Saving and Export Data

All the design modes enable you to save the entered data in an external file. To do it, use the **File|Save As** menu item when in the respective mode. This will open a standard Windows dialog box for choosing a directory and a file for saving the data. The filename and its extension are specified by the user.

To open the saved data use the **File|Open** menu item.

In some modes, the **File** menu has options allowing to export data to another mode. In this case, all data (common for the two modes) will be automatically transferred and the selected mode will be activated.

3. WeST

WeST enables to perform the analyses dealing with the determination of loads and actions on building structures in compliance with the recommendations of four design codes: SNiP 2.01.07-85, KMK 2.01.07-96, which is an interstate document of the CIS countries; SNiP 2.01.07-85* with the modification No. 2 dated 2003 as regards the snow loads, which is valid in the Russian Federation; DBN V.1.2-2:2006 with the modification No. 1 as regards the wind loads, which define loads and actions; and DSTU B. V.1.2-3:2006, which defines deflections and displacements and is valid in Ukraine. Besides, the updated version of SNiP 2.01.07-85* (SP 20.13330.2011 and SP 20.13330.2016) and modification No. 1 of SNiP 2.01.07-85 of the Republic of Belarus are implemented as well.

The application handles only the most frequently occurring load cases, as well as those cases for which meeting the requirements of the design codes involves a complicated logic and which, as experience shows, often result in errors.

In addition to this function, **WeST** to a certain extent plays the role of a manual which can be used to check data concerning zoning of territories according to loads and actions, or to obtain other reference information.

Finally, the application can be used to store the solutions frequently used by a designer, thus creating one's own reference information collection.

Further description of the **WeST** functions will combine description of the common performance capabilities, while indicated separately will be only the most significant differences related to realization of the differences contained in the requirements of the design codes. To a greater degree this concerns the differences between two versions of SNiP and DBN. DSTU, which specifies deflections and displacements, substitutes Chapter 10 of both versions of SNiP and, in comparison with this chapter, does not have any essential differences.

Hereinafter, in the text of description, we will not discriminate between SNiP 2.01.07-85, KMK 2.01.07-96 and SNiP 2.01.07-85* due to their identity in terms of the specified data and the form of results presentation.

3.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 3.1-1), with a set of buttons for selecting a working mode. These modes can be subdivided into two groups:

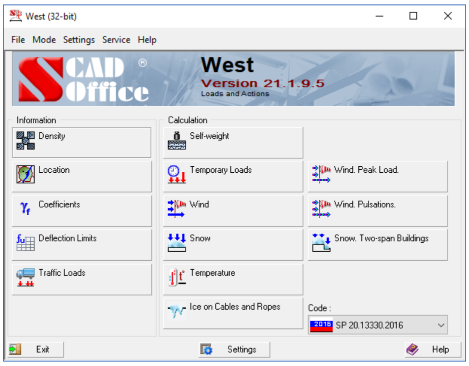


Figure 3.1-1. *The main window*

- reference modes — intended to perform reference and auxiliary operations related to the determination of loads and actions;
- design modes — intended to perform calculations of loads and actions in compliance with the SNiP, SP and DBN requirements (the modes are selected from the **Code** drop-down list).

A detailed description of each mode and their distinctive features associated with meeting the requirements of different documents is given in the subsequent sections. Their brief characteristic is given here.

The **reference modes** include:

- **Density** — reference data on the densities of the basic building materials;
- **Location** — determination of the load parameters that depend on the geographic location of the construction site;
- **Coefficients** — reference data on the values of the safety factor for load, γ_f;
- **Deflection Limits** — reference data on the rated values of limit deflections and displacements;
- **Working Life** (according to DBN) — reference data on the working life of buildings and engineering structures.

- **Traffic Loads** — reference data on the vertical traffic loads according to SP 20.13330.

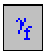
The **design modes** include:

- **Self-weight** — determination of the load from the self-weight of a multilayer pack consisting from different materials per unit area;
- **Temporary Loads** — determination of values of uniformly distributed temporary loads in various rooms in accordance with the instructions of Table 3 of SNiP, Table 8.3 of SP 20.13330, and Table 6.2 of DBN;
- **Wind** — calculation of the static component of wind loads for the structures of various types specified in the Annex 4 of SNiP, KMK, Annex E of SP20.13330.2011[*], Annex C of SP 20.13330.2016, and Annex I of DBN;
- **Wind. Pulsations** — calculation of the static component of wind loads for the rectangular in plan prismatic structures and determination of the dynamic amplification factor according to SNiP, KMK and SP;
- **Total Wind** — calculation of wind loads with the account of the dynamic action of the pulsation component for vertical prismatic and cylindrical structures according to Sec. 9.4 - 9.13 of DBN;
- **Wind Peak Load** — calculation the values of local wind loads according to SP 20.13330.
- **Snow** — calculation of the snow loads for the structures of various types specified in Annex 3 of SNiP, KMK, Annex D of SP 20.13330.2011, Annex B of SP 20.13330.2016, and Annex G of DBN;
- **Snow. Two-span Buildings** — calculation of the snow loads for two-span buildings of various types specified in Annex 3 of SNiP, KMK, Annex D of SP 20.13330.2011, Annex B of SP 20.13330.2016, and Annex G of DBN;
- **Temperature** — determination of the thermal actions in accordance with Section 8 of SNiP, KMK, Section 13 of SP, and Section 11 of DBN;
- **Ice on Cables and Ropes —** calculation of the loads from the ice weight and wind action on cables and ropes covered with ice in accordance with Section 8 of SNiP, KMK, Section 12 of SP, and Section 11 of DBN.

The **Menu** button switches from any mode to the main window. You can also switch directly from one mode to another using the **Mode** menu.

The design modes enable to obtain the characteristic and design load values. It should be noted that, depending on the selected design code, **WeST** uses the corresponding terminology. For example, if DBN is selected the serviceability, ultimate and quasi-permanent design load values are calculated, while in case of the analysis according to SP such terms as design value (ULS) and design value (SLS) are used.

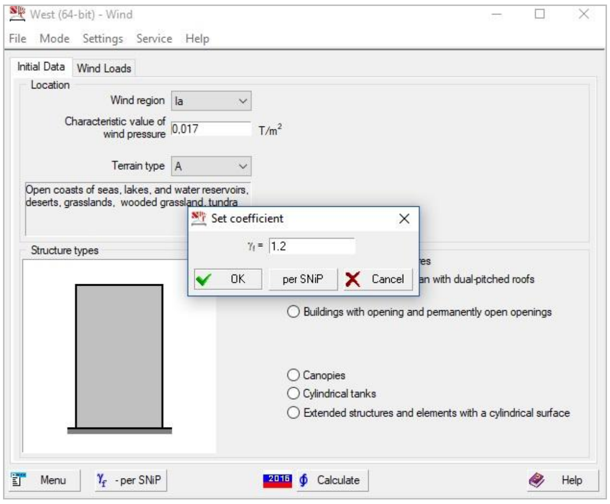
3.2 Settings

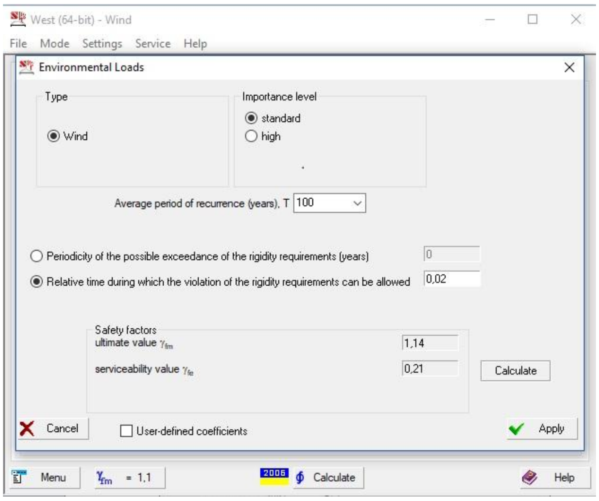
Specific for the settings of this application is the button  (available in most modes) which enables to specify the **Safety factor for load** (γ_f) (Fig. 3.2 -1). This is due to the fact that all working modes use values of the factor γ_f recommended by the respective sections of SNiP, KMK, SP and DBN. However, for certain special structures (e.g., nuclear power plants) the respective codes recommend other values for γ_f.

In such cases you have to specify a different value in the **Safety factor for load** field, which then will be used for the calculation of the design load values (the characteristic load value remains the same).

If the **per code** button is used when specifying this factor, the **Snow** and **Wind** modes will use the value $\gamma_f = 1.4$, which is true for all cases except for the snow load on buildings with light-weight roofs (for analyses according to SNiP see Sec. 5.7, 6.11).

[*] The calculation of wind loads according to SP 20.13330.2011 is not implemented in the current version of WEST due to the possibility of ambiguous interpretation of some provisions of this document (see the **SNiP Problems** section of the SCAD Soft website for details: http://www.scadsoft.com/SNiP/WindQ1.pdf).





a - according to SNiP and according to SP *b - according to DBN*

Figure 3.2-1. *Safety factor for load*

3.3 Information Modes

3.3.1 Density

Tables of the **Density** mode (see Fig. 3.3.1-1) contain the information, taken from the reference literature, on the unit volume weight (or unit area weight) of the following groups of building materials or structural elements:

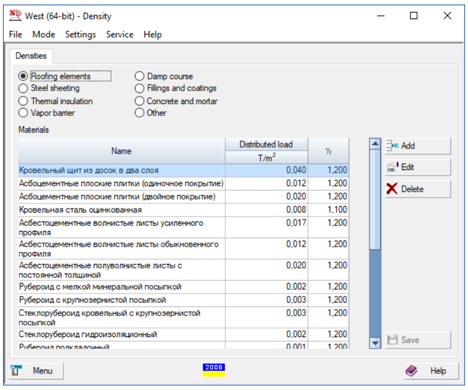


Figure 3.3.1-1. *The* **Density** *dialog box*

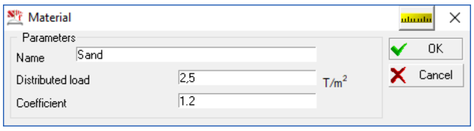


Figure 3.3.1-2. *The* **Material** *dialog box*

- roofing elements;
- steel sheeting;
- thermal insulation;
- vapor barrier;
- damp course;
- fillings and coatings;
- concrete and mortar;
- other.

Selecting a group with the help of respective radio buttons will display a table containing the names, the values of the unit volume weight or the unit area weight, and the safety factors for loads of the corresponding materials, selected in accordance with the recommendations of Table 2 of SNiP, KMK and Table 6.1 of DBN.

The **Edit** button enables you to enter new property values to replace those suggested in the table. Clicking this button invokes the **Material** dialog box (see Fig. 3.3.1-2), which contains the name and properties of the material from the line selected (highlighted) in the table. The most common use of this function deals with updating the previously entered data (e.g., if the specifications have been modified). You can obtain the same result as in the case of clicking the **Edit** button by double-clicking the left mouse button on the table line you want to edit.

The **Add** button enables to expand any of the reference tables with additional lines that describe new building materials or products. The name and properties of the new material are entered in the text fields of the **Material** dialog box which appears once you click the **Add** button.

Materials can be removed from the list with the help of the **Delete** button.

3.3.2 Densities (EN 1991-1-1)

Tables of the **Density** mode contain the information, taken from the Annex A of EN 1991-1-1, on the unit volume weight (or unit area weight) of the building materials.

3.3.3 Location

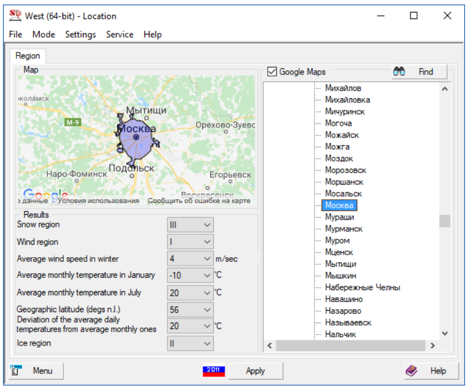


Figure 3.3.3-1. *The* **Location** *dialog box*

Much data on the loads and actions depend on the geographical location of the construction site. The search for such data can be performed in the **Location** dialog box (Fig. 3.3.3-1) using the tree-like structure of the administrative-territorial division in the **Find** field of the dialog. If you select the name of a construction site in the tree, the **Results** group will contain the climatic data and the corresponding map fragment will be displayed in the **Map** field. The **Find** button enables to perform a search for an administrative unit or a city by the specified name in the respective dialog box. If your computer has Internet access, then the **Online Maps** checkbox allows you to display the map based on the cartography online service.

Since the territorial climatic regions have rather vague boundaries on maps, it is assumed that these boundaries do not always coincide with those of the administrative-territorial entities, i.e. an administrative region may not belong entirely to a certain climatic zone but may be divided between two zones. Large cities, where mass-scale construction is concentrated, are shown separately in the tree structure. According to DBN the climatic loads for them may be specified within the region loads in compliance with the instructions of Annex F. The respective tabs of the modes have the button, clicking which will open a dialog box with a list of cities. If you select a city and click the **Apply** button, the data on the characteristic load value will be used in the analysis.

Once you specify the location of the construction site, the following data will appear in the fields of the **Results** group:

- number of snow region (Map 1* of Annex 5 of SNiP, Map 1 of Annex G of SP 20.13330.2011, and Map 1 of Annex F of SP 20.13330.2016, Fig. 8.1 of DBN);
- number of wind region (Map 3 of Annex 5 to SNiP, Map 3 of Annex G of SP 20.13330.2011, and Map 2 of Annex F of SP 20.13330.2016, Fig. 9.1 of DBN);
- number of ice region (Map 4 of Annex 5 of SNiP, Map 4 of Annex G of SP 20.13330.2011, Fig. 10.1 of DBN);
- average wind speed in winter (Map 2 of Annex 5 of SNiP, Map 2 of Annex G of SP);
- average monthly temperature in July (Map 6 of Annex 5 of SNiP, Map 6 of Annex G of SP);
- average monthly temperature in January (Map 5 of Annex 5 of SNiP, Map 5 of Annex G of SP);
- geographic latitude used to determine the maximum solar radiation value (SNiP 2.01.01-82);
- deviation of the average daily temperatures from the average monthly ones (Map 7 of Annex 5 of SNiP, Map 7 of Annex G of SP);
- number of ice-wind region (Fig.10.2 of DBN).

All these values can be modified by the user by selecting them from the respective drop-down list.

Click the **Apply** button to save the selected values for using them in the design modes of the application.

3.3.4 Coefficients

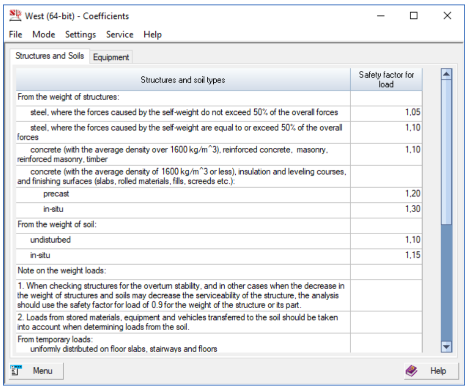


Figure 3.3.4-1. *The* **Structures and Soils** *tab of the* **Coefficients** *dialog box*

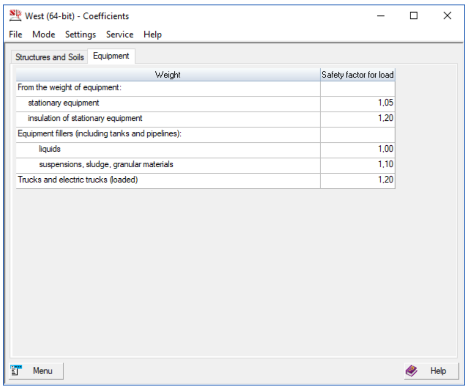


Figure 3.3.4-2. *The* **Equipment** *tab of the* **Coefficients** *dialog box*

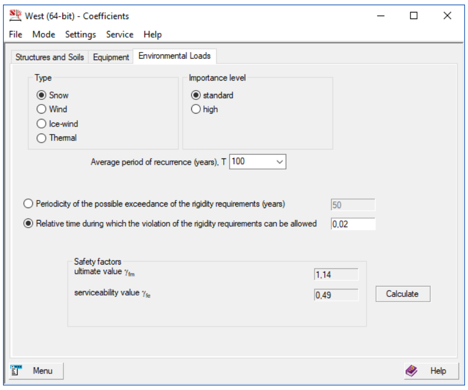


Figure 3.3.4-3. *The* **Environmental Loads** *tab of the* **Coefficients** *dialog box*

This mode enables to obtain reference information on the values of the safety factors for loads γ_f, defined by SNiP, KMK, SP and DBN. The **Structures and Soils** tab (see Fig. 3.3.4-1) contains the values from Table 1 of SNiP, KMK, Table 7.1 of SP 20.13330, and Table 5.1 of DBN, and the **Equipment** tab (see Fig. 3.3.4-2) contains the values from Table 2 of SNiP, KMK, Table 8.1 of SP 20.13330, and Table 6.1 of DBN.

The **Environmental Loads** tab (see Fig. 3.3.4-3) enables to obtain reference information on the values of the safety factors for loads γ_f, for the wind, snow and ice-wind loads according to DBN, which depend on the working life, importance level of a building or structure, and the time during which the limit inequalities of the serviceability limit state can be violated, as well as for the thermal loads.

The calculation of the safety factors for load complies with requirements of the following sections of DBN:

- for the snow load — 8.11, 8.12;
- for the wind load — 9.14, 9.15;
- for the ice-wind load — 10.10, 10.11;
- for the thermal load – 11.8, 11.9.

3.3.5 Deflection Limits

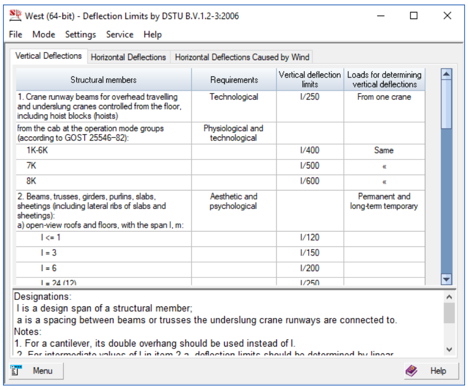


Figure 3.3.5-1. *The* **Vertical Deflections** *tab of the* **Deflection Limits** *dialog box*

This mode enables to obtain information on the values of the vertical deflection limits (see Fig. 3.3.5-1), and the horizontal ones caused by cranes and wind defined in Tables 19, 21 and 22 of SNiP, KMK, Annex F of SP 20.13330.2011, Annex E of SP 20.13330.2016, and Tables 1, 3, 4 of DSTU.

3.3.6 Working Life

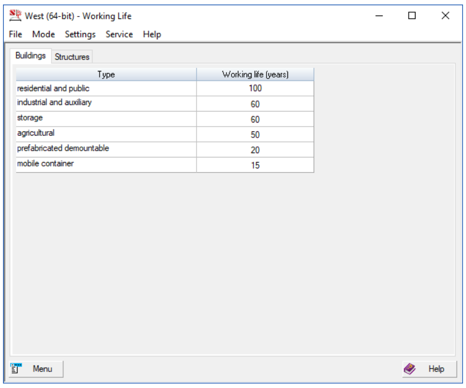


Figure 3.3.6-1. *The* **Working Life** *dialog box*

This mode is used only when DBN is selected. It provides reference information on the working life of buildings and structures in accordance with Annex C of DBN (see Fig. 3.3.6-1). This information is required for specifying the safety factors for loads γ_f, for the wind, snow and ice-wind loads.

3.3.7 Traffic Loads

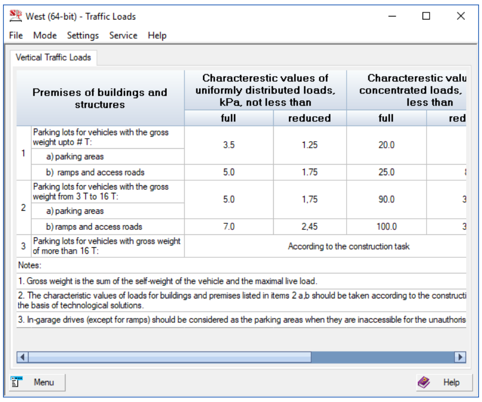


Figure 3.3.7-1. *The* **Traffic Loads** *dialog box*

This mode is used only when SP is selected. It provides the reference information on the vertical loads caused by the transport vehicles (see Fig. 3.3.7-1).

3.4 Design Modes

When SP 20.13330.2011 is used, modes for calculating snow, wind and ice loads contain a special checkbox *Building or structure under construction*, which enables to take into account the requirements of Sec. 4.5 of SP 20.13330.2011. The reduction of the design load for buildings and structures under construction is not obligatory in SP 20.13330.2016, therefore this checkbox is not provided. The reduction of load (allowed by SP 20.13330.2016) can be taken into account by reducing the safety factor for load by 20%.

3.4.1 Self-weight

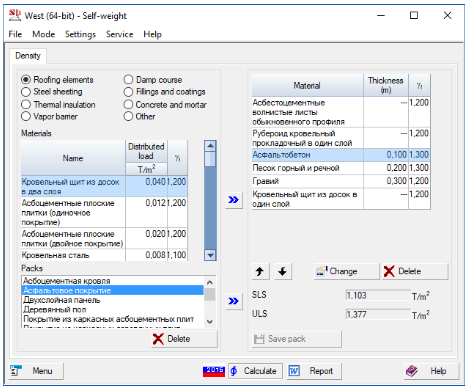


Figure. 3.4.1-1. *The* **Self-weight** *dialog box*

The **Self-weight** mode enables to perform the calculation of loads caused by the self-weight of a pack composed of multiple layers of different materials. This function is implemented in the respective dialog box (see Fig. 3.4.1-1), the left half of which displays a table containing the general list of materials, and its right half contains a table with the list of selected materials included in the pack.

To specify each layer of the pack, it is necessary to select the group, which the material of the layer belongs to. To do this, select a radio button with the name of the respective group (roofing elements, steel sheeting, thermal insulation, vapor barrier, damp course, fillings and coatings, concrete and mortar, and other). Once the group is selected, the left table will display the list of the materials included in this group. Each line of the table contains the name of a material, the value of its volume weight or unit area weight of the finished structural element, as well as the value of the safety factor for load according to Sec. 2.2 of SNiP, Sec. 7.2 of SP 20.13330, and Sec. 6.2 of DBN.

The upper button  enables to transfer the material from the selected (highlighted) line of the left table to the right (working) table, which will thus contain the description of the whole pack.

If the layer thickness is fixed and can not be modified, the respective line of the table is marked with the dash symbol. Otherwise, double clicking on the desired line of the working table enables you to enter the layer thickness data in the **Thickness** column.

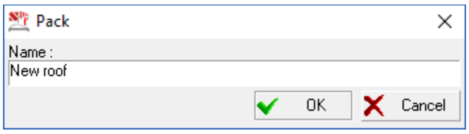


Figure 3.4.1-2. *The* **Pack** *dialog box*

The composition of the pack defined in the working table can be modified using the **Delete** and **Change** buttons. If the pack you have entered is a typical one, it can be saved under any suitable name using the **Save pack** button. Clicking this button invokes a dialog box (see Fig. 3.4.1-2) where you can specify the pack name.

The saved pack will be added to the **Packs** list and can be retrieved from this list to the working table with the help of the lower button . This pack can be supplemented with new layers, or some layers can be removed from it.

Clicking the **Calculate** button will display the respective values of the load per unit of area caused by the pack self-weight in the **Characteristic load** and **Design load** fields (according to SNiP), or **Serviceability load** and **Ultimate load** fields (according to DBN).

3.4.2 Temporary Loads

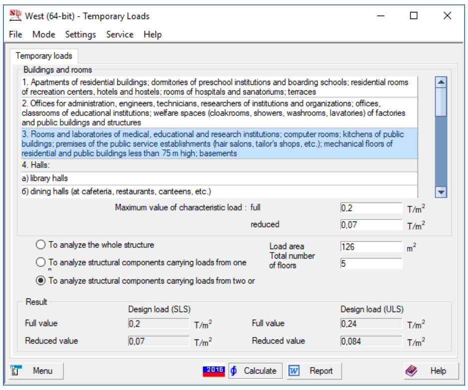


Figure 3.4.2-1. *The* **Temporary Loads** *dialog box*

This mode (see Fig. 3.4.2-1) enables to select the uniformly distributed temporary loads on floor slabs, stairs, floors, etc., in compliance with Sec. 3.5-3.9 of SNiP, KMK, Sec. 8.2 of SP, and Sec. 6.5-6.9 of DBN.

This mode determines full and reduced values of characteristic and design loads (according to SNiP, KMK), full and reduced values of design loads (according to SP), characteristic and quasi-permanent loads (according to DBN) using the reduction factors which allow for the load area subjected to loading (see Sec. 3.8 of SNiP, Sec. 8.2.4 of SP 20.13330, and Sec. 6.8 of DBN), and the combination factors which allow for the number of the loaded floors of a multi-storey building (see Sec. 3.9 of SNiP, KMK, Sec. 8.2.5 of SP 20.13330, and Sec. 6.9 of DBN).

The type of a room is selected from the **Buildings and rooms** list. The type of a structure being analyzed is specified using the respective radio buttons.

To obtain the load values, click the **Calculate** button; this will display the characteristic and design load values (according to SNiP), and the serviceability, ultimate and quasi-permanent load values (according to DBN).

3.4.3 Wind

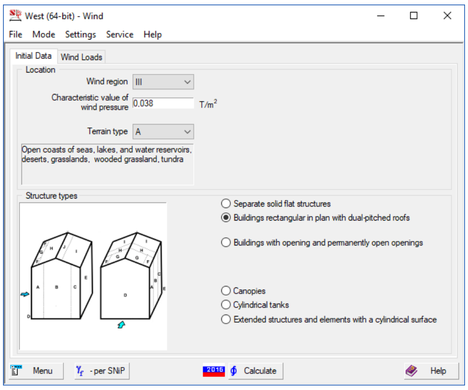


Figure 3.4.3-1. *The* **Initial Data** *tab of the* **Wind** *dialog box*

This mode enables to perform the analysis of wind loads on some of the buildings, structures, and structural members, the models of which are given in Annex 4 of SNiP, KMK, Annex I of DBN, and Annex C of SP 20.13330.2016. The static value of the wind load normal to the structure surface is determined in all cases. The two-tab dialog box, **Wind**, is used to enter the initial data and obtain the results.

The first tab, **Initial Data** (see Fig. 3.4.3-1), is used to enter the information on the terrain type and the wind region where the considered structure is located. The values from the **Location** dialog box (provided the **Apply** button has been clicked in it) are transferred to the **Wind region** list and the **Characteristic value of wind pressure** field.

At the same time, a wind region can be independently selected from the respective list. If the entered characteristic value of the wind pressure is different from that recommended by the codes, the name of a wind region is not indicated in the list.

If DBN is selected, the button provides access to the list of Ukrainian cities, which (based on Annex F) contains the refined data on the characteristic load value. If you select the desired city and click the **OK** button, the characteristic load value will be automatically transferred to the initial data.

Besides, when determining the wind loads according to DBN, it is necessary to indicate whether the first natural vibration period of the structure exceeds 0.25 sec.

The terrain type is selected from the respective list. The structure type is specified with the help of the respective group of radio buttons.

Once you have entered all the required data on the **Initial Data** tab, proceed to the next one – **Wind Loads** (see Fig. 3.4.3-2), where the additional initial data necessary for the analysis have to be specified and the results of the calculation are displayed. The description of this dialog box for different types of structures is given below.

The **sign convention** for the wind pressure values for all types of structures is accepted in accordance with Sec. 6.6 of SNiP and Sec. 9.8 of DBN, namely: wind pressure is considered to be positive if it is directed towards the respective surface, and negative – if it is directed away from the surface.

For some types of buildings, SP 20.13330 allows to calculate the pulsation component of the wind load using formula (11.5) without calculating natural frequencies and vibration modes. In the program, this calculation is implemented when the checkbox of the same name is checked.

Limitations

Variation of the wind pressure with height is taken into account for all structures (except for canopies) and walls of buildings. It is assumed for all roofs that the wind load does not change with height and corresponds to the top of the roof elevation.

Vertical Surfaces and Those Deviating from the Vertical by No More Than 15°

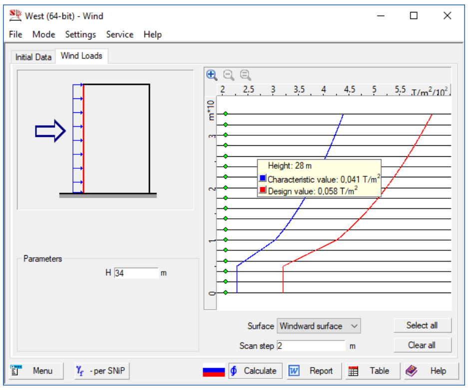
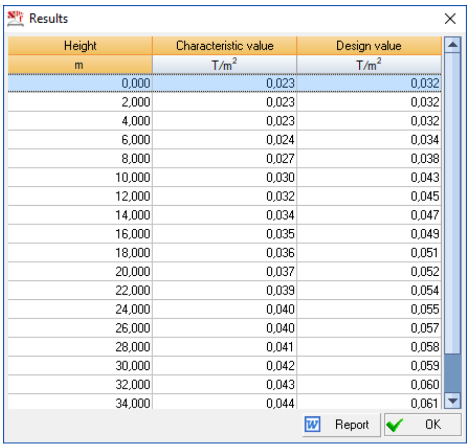


Figure 3.4.3-2. *The* **Wind Loads** *tab of the* **Wind** *dialog box*


Results

Height m	Characteristic value T/m²	Design value T/m²
0,000	0,023	0,032
2,000	0,023	0,032
4,000	0,023	0,032
6,000	0,024	0,034
8,000	0,027	0,038
10,000	0,030	0,043
12,000	0,032	0,045
14,000	0,034	0,047
16,000	0,035	0,049
18,000	0,036	0,051
20,000	0,037	0,052
22,000	0,039	0,054
24,000	0,040	0,055
26,000	0,040	0,057
28,000	0,041	0,058
30,000	0,042	0,059
32,000	0,043	0,060
34,000	0,044	0,061

Report OK

Figure 3.4.3-3. *The* **Results** *dialog box*

When this type of structure is selected (see model 1 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.1 of Annex C of SP 20.13330.2016), the value of the structure height *H* (when the calculation is performed according to SP 20.13330.2016, the length L is also specified) and the scan step for the result have to be specified in the **Wind Loads** tab (see Fig. 3.4.3-2). The type of the surface (windward or leeward), for which the wind load values have to be calculated, is selected from the drop-down list.

When the calculations are performed according to SP 20.13330.2016, it is necessary to specify which area of the roof or walls will be considered.

The result window displays a plot of the load variation with height. This plot can be dynamically digitized, when the application displays the wind load value for the height indicated by the cursor. Heights correspond to the selected scan step.

The green points (marks) on the axis of ordinates indicate the intermediate height values the calculations will be performed for. The spacing between them corresponds to the specified scan step. The marks can be deleted and restored again by placing the mouse pointer over them and left-clicking. The wind pressure values at the elevations with marks which have not been deleted are given in the table generated by clicking the **Table** button and is displayed in the **Results** dialog (see Fig. 3.4.3-3).

The **Select all** and **Clear all** buttons, respectively, enable you to add to the table (plot) or remove from the table (plot) all points corresponding to the scan step.

There are the zooming buttons above the wind variation plot. The plus-sign button serves for zooming in; each time you click this button you enlarge the plot scale twofold. The minus-sign button is used for step-by-step zooming out and functions only after the plus-sign button has been used. You can immediately return to the original size of the plot by clicking the equal button.

Similar buttons are used in all modes where the load variation plots are displayed.

Single-span Buildings with No Skylights

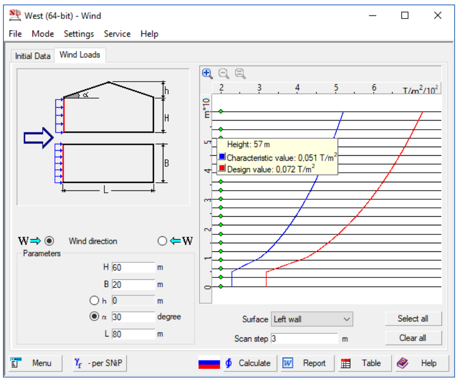


Figure 3.4.3-4. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

For the structures of this type (see model 2 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.2 of SP 20.13330.2016), the dimensions of the building are entered in the **Wind Loads** tab (see Fig. 3.4.3-4). The type of the surface (roof, left wall, right wall, side walls), for which the wind load values have to be calculated, is selected from the drop-down list, and the radio buttons are used to specify the design wind direction.

When the calculations are performed according to SP 20.13330.2016, it is necessary to specify which area of the roof or walls will be considered.

Single-span Buildings with No Skylights, Permanently Open on One Side

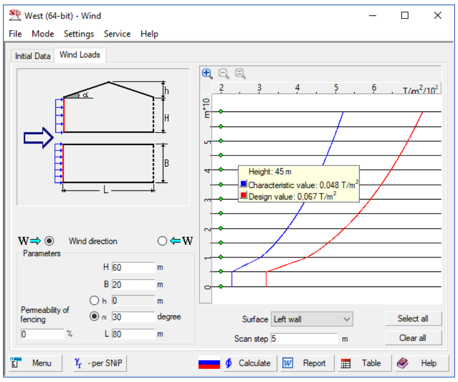


Figure 3.4.3-5. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (see model 9 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.9 of Annex C of SP 20.13330.2016), the dimensions of the building are entered in the **Wind Loads** tab (see Fig. 3.4.3-5). The type of the surface (roof, left wall, interior surfaces, side walls), for which the wind load values have to be calculated, is selected from the drop-down list. The design wind direction is specified using the respective radio buttons.

When the calculations are performed according to SP 20.13330.2016, it is necessary to specify which area of the roof or walls will be considered.

Since in this case we consider a simultaneous action of the wind pressure outside and inside the structure, the total load on the selected surface will be obtained in the result.

The value of the internal pressure aerodynamic coefficient C_i at the given permeability value within the range of 5% to 30% has been calculated by interpolation between the boundary values indicated in the codes.

Buildings with Vaulted Roofs and Those with a Similar Shape

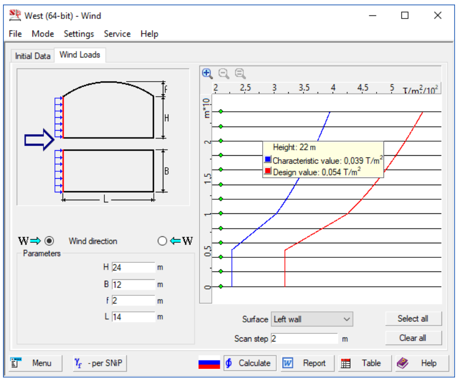


Figure 3.4.3-6. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (see model 3 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.3 of Annex C of SP 20.13330.2016), the dimensions of the building are entered in the **Wind Loads** tab (see Fig. 3.4.3-6). The type of the surface (roof, left wall, right wall, side walls), for which the wind load values have to be calculated, is selected from the drop-down list. The design wind direction is specified using the respective radio buttons.

Canopies

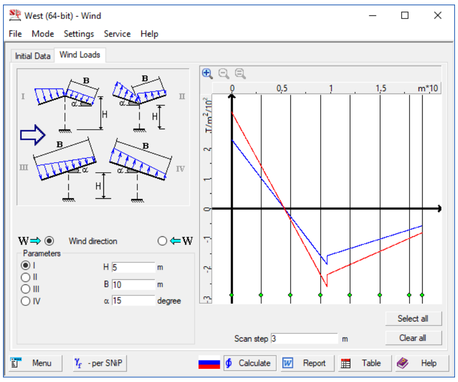


Figure 3.4.3-7. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

For the structures of this type (see model 11 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.10 of Annex C of SP 20.13330.2016), you have to select the design of the shed using the respective radio buttons in the **Wind Loads** tab (see Fig. 3.4.3-7) to enter the dimensions, and specify the design wind direction.

The values of the load on the roof are given in the form of a plot, which can be dynamically digitized. However, the ordinate of the considered point on the horizontal roof projection is selected rather than its elevation.

Cylindrical Tanks

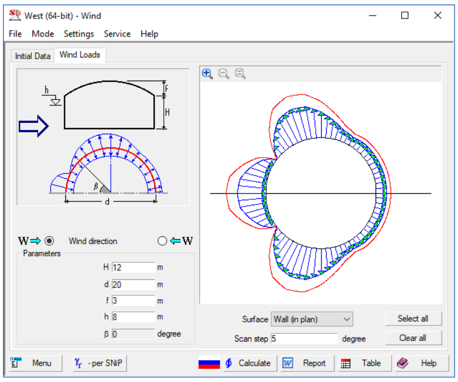


Figure 3.4.3-8. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (see model 12 *b* of Annex 4 of SNiP, KMK or Annex I of DBN), the dimensions of the structure are entered in the **Wind Loads** tab (see Fig. 3.4.3-8). The type of the surface (spherical roof, wall along the height, wall in plan, internal pressure), for which the wind load values have to be calculated, is selected from the drop-down list. The design wind direction is specified using the respective radio buttons.

The conical roof is not considered in the given program version, therefore the roof surface is specified as "spherical".

Since the internal pressure is considered only if there is no roof or if the floating roof is in its lower position, the internal pressure values are given separately here, without adding them to the values of the external pressure on the wall. It is up to the user to select the design case (either the total value or the individual values). Moreover, the user has to consider different options of the tank filling and, therefore, the starting point of the measurement of the internal pressure along the height.

Two plots are used to describe the pressure distribution over the tank wall surface:

- in the horizontal plane at the level defined by the parameter *h* — **Wall (in plan)** in the **Surface** drop-down menu;
- on the vertical the position of which is defined by the angle β between the radius passing through this vertical and the horizontal axis — **Wall (along the height)** in the **Surface** drop-down menu.

Depending on the selected plot, the result will be:

- distribution of the wind pressure in plan at the specified height;
- variation of the wind pressure with height at the specified position of the vertical.

Extended Structures and Elements with a Cylindrical Surface

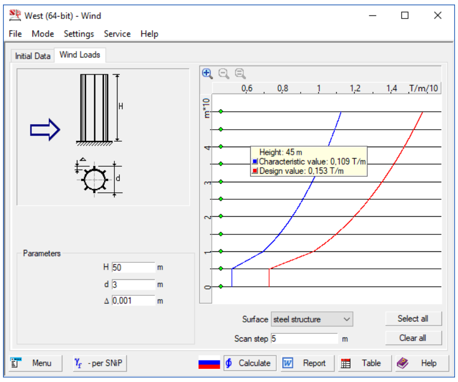


Figure 3.4.3-9. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

For the structures of this type, unlike those described above, the linear load rather than the surface one is determined. Once this type of structure is selected (see model 14 of Annex 4 of SNiP, KMK or Annex I of DBN, model C.1.12 of Annex C of SP 20.13330.2016), you have to specify the height and diameter of the structure in the **Wind Loads** tab (see Fig 3.4.3-9), and select the type of surface from the drop-down list (timber, steel, concrete or reinforced concrete, brickwork), thus defining the roughness parameter Δ. If necessary, this parameter can be specified in the respective text field.

The calculation results are given in the form of a plot of linear load variation with height. This plot can be dynamically digitized, when the application displays the load values for the height indicated by the cursor.

Limitations

The wind pressure value is a function of the Reynolds number Re, for which the value range of up to $Re = 3.2•10^6$ is considered in the codes. When out of this range, it is assumed that the value of the aerodynamic coefficient $C_{x\infty}$ remains constant.

Based on the relative roughness parameter Δ/d, the linear interpolation is performed between the plots given in model 14 of Annex 4 of SNiP, KMK or Annex I of DBN. Meanwhile, in case $\Delta/d < 10^{-4}$ and $\Delta/d > 0{,}05$ the value $C_{x\infty}$ is determined by the limit curves.

Recommendations for Determination of the Wind Load on Vertical Cylindrical Structures

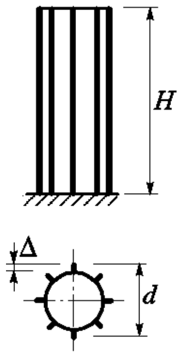


Figure 3.4.3-10. Cylindrical structure

In some cases at certain relationships between the diameter and height of the cylinder (see Fig. 3.4.3-10.), the wind load calculated according to the requirements of the codes for two cylinders of different height (all other initial data of the problem being equal), may differ within the range of the height common to both cylinders. Such a result contradicts the physics of the process.

This is due to the fact that, when determining the aerodynamic coefficients dependent on the Reynolds number, according to the requirements of the design codes, the wind velocity (used to determine the Reynolds number) is taken at the structure top level. Naturally, the Reynolds number grows with an increase of the height, which in turn causes the increase of the aerodynamic coefficients that are assumed to be constant throughout the height of the cylinder; and the wind load increases as well. This leads to the different results obtained at the same height.

In this case, we can recommend the following procedure for solving the problem for cylindrical structures with the height of more than 30 m:

- the structure is divided into several sections along the height (it is recommended to take the height of each section as not less than 10 m);
- then you have to solve a number of problems equal to the number of sections reeived after dividing the structure along the height; and, the height of the structure in each problem is increased incrementally;
- the load on the whole structure is taken as the sum of loads on the individual sections this structure has been divided into along the height for the further use in the analysis.

The use of the suggested procedure is shown in Figure 3.4.3-11.

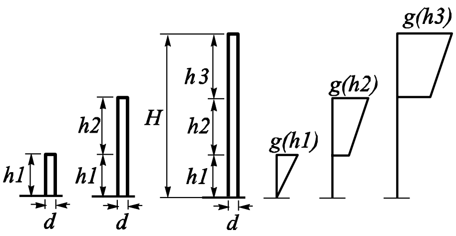


Figure 3.4.3-11. Illustration of the division method

Inclined Tube Elements

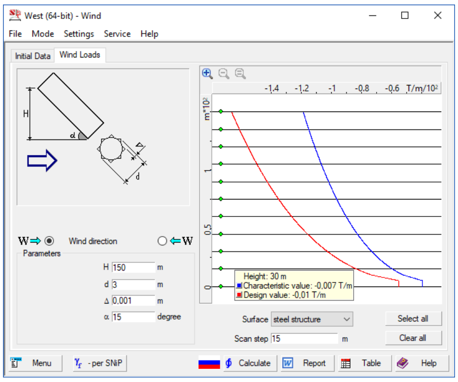


Figure 3.4.3-12. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (see model 18 of Annex 4 of SNiP, KMK or Annex I of DBN), the dimensions of the member and the wind direction are specified in the **Wind Loads** tab (see Fig. 3.4.3-12). Here, like in the previous case, the linear load is determined, and only the load component normal to the member axis is considered.

Since the formulation given in Annex C of SP 20.13330.2016 for this case is clearly incorrect, the calculations according to this code are not performed.

The sign convention for the load is as follows: the load is considered to be positive if its projection on the vertical is downward, i.e. the weight and wind effects are summed.

The calculation results are given in the form of a plot of linear load variation with height. This plot can be dynamically digitized, when the application displays the load values for the height indicated by the cursor.

Guy-wires

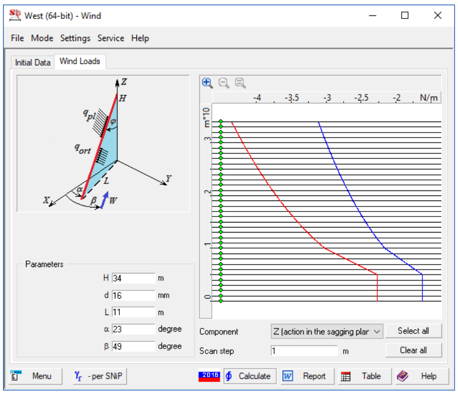


Figure 3.4.3-13. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

The initial data for calculating wind loads on the mast guy-wires (see Annex E of SP 20.13330.2011, Fig. E.18 and Annex C of SP 20.13330.2016, Fig. C.18) is specified in the Wind Loads tab (Fig. 3.4.3-13) and includes the parameters defining the position of a guy-wire in space, its diameter and the direction of the wind speed vector.

The results of the calculation include components of the wind pressure on a guyed-wire given in the global coordinate system:

The results of the calculation include components of the wind pressure on a guyed-wire given in the global coordinate system:

$q_{pl,X}$, $q_{pl,Y}$, $q_{pl,Z}$ — acting in the sag plane;

$q_{ort,X}$, $q_{pl,Y}$ — directed perpendicular to the sag plane.

They have to be considered separately because the qpl,Z component summed with the load from the guy-wire weight is used in the nonlinear analysis.

The result window displays a plot of the load variation with height. This plot can be dynamically digitized, when the application displays the wind load value for the height indicated by the cursor.

3.4.4 Wind. Pulsations

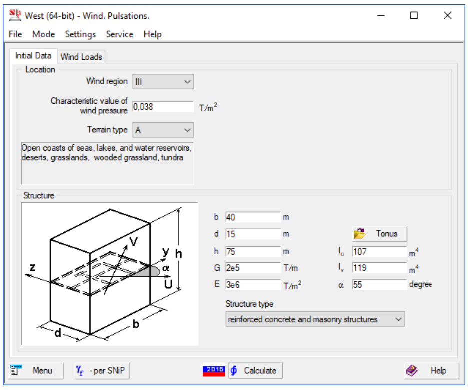


Figure 3.4.4-1. *The* **Initial Data** *tab of the* **Wind. Pulsations.** *dialog box*

This mode enables to perform the analysis of the wind loads on the prismatic, rectangular in plan buildings and structures with a constant stiffness along their height, or those which can be approximately considered as such (see model 13 of Annex 4 of SNiP, KMK).

Unlike the **Wind** mode (see above), this mode allows you to determine not only the static wind load, but also the dynamic action of pulsations, and it also gives recommendations on whether a detailed dynamical analysis is required. Besides, it provides an estimate of the dynamic amplification factor which can serve as the basis for making a decision whether a detailed analysis is really necessary and whether the pulsation component should be taken into account.

Peculiarities of the implementation

The pulsation component of the wind load is calculated according to Sec. 6.7, *c* of SNiP, KMK; and the requirement of Sec. 6.8 regarding the allowance for the inertia forces in the cases when the oscillation frequencies exceed the limit ones is also always taken into account.

The two-tab dialog box, **Wind. Pulsations**, is used to enter the initial data and obtain the results (Fig. 3.4.4-1 and 3.4.4-2).

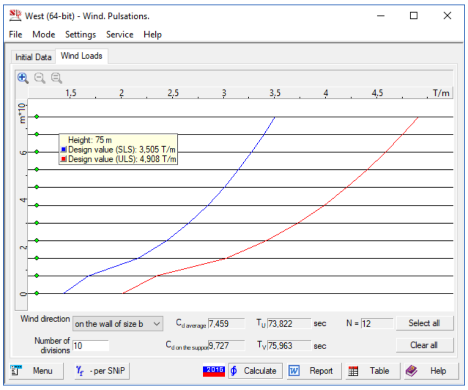
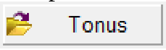


Figure 3.4.4-2. *The* **Wind Loads** *tab of the* **Wind. Pulsations.** *dialog box*

The first tab, **Initial Data** (see Fig. 3.4.4-1), is used to enter the information on the terrain type and the wind region where the considered structure is located. The values from the **Location** dialog box (provided the **Apply** button has been clicked in it) are transferred to the **Wind region** list and the **Characteristic value of wind pressure** field. At the same time, a wind region can be independently selected from the respective list. The types of the terrain and structure are specified using the respective lists.

The following data are also specified in this tab: dimensions of the building plan, its height, weight of the linear meter of the building along its height (allowing for the weight of the floors and the long-term load), and the average value of elasticity modulus of the material of the load-bearing structures. It is necessary to specify the following parameters: the principal moments of inertia about the U and V axes, and the angle of rotation of the U and V axes about the Y and Z axes along which the wind load acts.

The geometric properties of the building plan can be determined with the help of **TONUS**. The properties calculated in **TONUS** are retrieved from a file with the .tns extension (the button 

).

The building must be rectangular in its plan. Obviously, many real buildings and structures can be assumed rectangular only as approximation, hence the results of the analysis will be also approximate.

If the external contour of a building is formed by the enclosing structures rather than by the load-bearing ones (e.g., as in frame buildings), the moments of inertia can be calculated as those for a multipoint section. When using **TONUS** in such a case it is necessary to specify the location in plan and cross-sectional dimensions of all load-bearing members.

It should be noted that an accurate calculation of the pulsation component in accordance with Sec. 6.7-6.9 of SNiP requires to know the modes and frequencies of natural vibrations, i.e. the detailed information about the whole structure. In the **WeST** application they are approximately determined as for a cantilever bar, the stiffness of which is constant throughout the height and corresponds to the stiffness of a bar with the cross-section formed by the load-bearing walls of the building.

Once you have entered all the required data in the **Initial Data** tab, proceed to the next one – **Wind Loads** (see Fig. 0-2), where the following data has to be specified:

- wind direction ("on the wall of size b", " on the wall of size d", or “diagonally”);
- number of sections the building is divided into along the height, and for which the results will be obtained (see Sec. 6.7 *c* of SNiP).

The results of the calculation are given in the form of a plot of variation of the wind load static component (characteristic and design values) with height. This plot can be dynamically digitized, when the application displays the function values for the height indicated by the cursor.

Besides, the results of the calculation also include two dynamic amplification factors:

- for displacements — the average ratio between the pulsation component of the wind load and its static component;
- for the total bending moment at the base – the ratio between the pulsation component of the bending moment at the building base and the corresponding static value.

Guided by these values, the user can make a decision whether to perform a detailed dynamic analysis of the building structure. Generally, the dynamic analysis is not reasonable if the dynamic amplification factors are less than 0.2, and absolutely necessary if they exceed 0.4.

To provide an additional check, this tab also contains the values of the first natural vibration periods for the vibrations in the U and V planes.

3.4.5 Total Wind

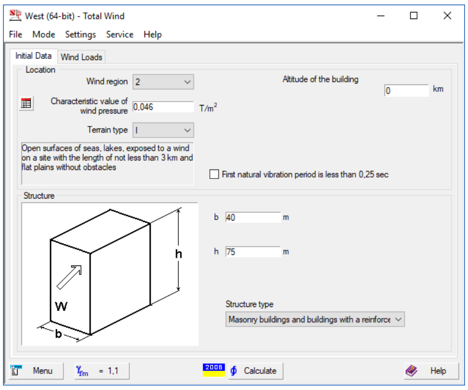


Figure 3.4.5-1. *The* **Initial Data** *tab of the* **Total Wind** *dialog box*

This mode enables to calculate wind loads with the account of the dynamic effect of the pulsation component for vertical prismatic and cylindrical structures no more than 200 m high with a nearly constant stiffness along the height in accordance with Sec. 9.4-9.13 of DBN. The projection of the structure onto the vertical plane in the direction perpendicular to the wind action plane must comply with the models 13, 14 of Annex I of DBN.

When entering the initial data it is necessary to specify whether the first natural vibration period of the structure exceeds 0,25 sec.

Unlike the **Wind** mode (see above), this mode enables to determine not only the static wind load, but also the dynamic action of pulsations and their mutual height correlation.

Peculiarities of the implementation

When calculating the total wind load according to Sec. 9.4-9.13 of DBN, the direction factor C_{dir} and the orography coefficient C_{rel} are taken as 1,0.

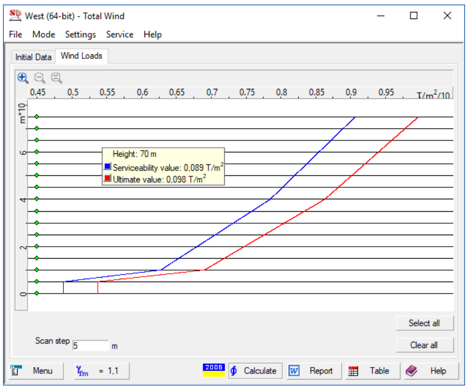


Figure 3.4.5-2. *The* **Wind Loads** *tab of the* **Total Wind** *dialog box*

The two-tab dialog box, **Total Wind**, is used to enter the initial data and obtain the results.

The first tab, **Initial Data** (see Fig. 3.4.5-1), is used to enter the information on the terrain type and the wind region where the considered structure is located. The values from the **Location** dialog box (provided the **Apply** button has been clicked in it) are transferred to the **Wind region** list and the **Characteristic value of wind pressure** field. At the same time, a wind region can be independently selected from the respective list. The button  provides access to the list of Ukrainian cities, which (based on Annex F) contains the refined data on the characteristic load value. If you select the desired city and click the **OK** button, the characteristic load value will be automatically transferred to the initial data. The types of the terrain and structure are specified using the respective lists.

The following data are also specified in this tab: dimensions of the building plan in the direction perpendicular to the wind action plane, and its height.

The building (structure) must be rectangular or round in its plan. Obviously, many real buildings and structures can be assumed as such only as approximation, hence the results of the analysis will be also approximate.

The **Wind Loads** tab (see Fig. 3.4.5-2) provides the results of the calculation in the form of a plot of variation of the total wind load (serviceability and ultimate values) with height. This plot can be dynamically digitized, when the application displays the function values for the height indicated by the cursor.

3.4.6 Peak Wind Load

This mode enables to calculate the values of local wind loads according to SP 20.13330, which can turn out to be critical for elements of wall and roof enclosures, and for their fasteners. These loads can be positive (wind pressure) or negative (suction), and their value depends on the area of the enclosure the load is applied to and its location. This mode considers rectangular in plan buildings with flat roofs or those which can be approximately considered as such.

The two-tab dialog box, **Wind. Peak Load**, (see Fig. 3.4.6-1 and 3.4.6-2) is used to enter the initial data and obtain the results.

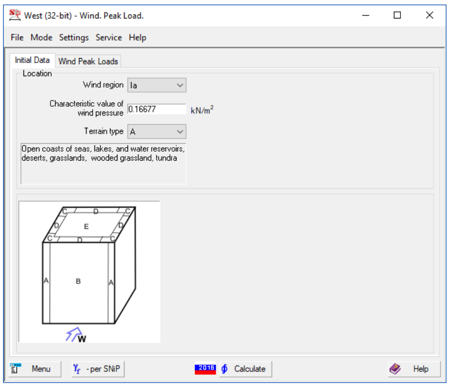


Figure 3.4.6-1. *The* **Initial Data** *tab*

The first tab, **Initial Data** (see Fig. 3.4.6-1), is used to enter the information on the terrain type and the wind region where the considered structure is located. The values from the **Location** dialog box (provided the **Apply** button has been clicked in it) are transferred to the **Wind region** list and the **Characteristic value of wind pressure** field. At the same time, a wind region can be independently selected from the respective list. The terrain type is selected from the **Terrain type** list.

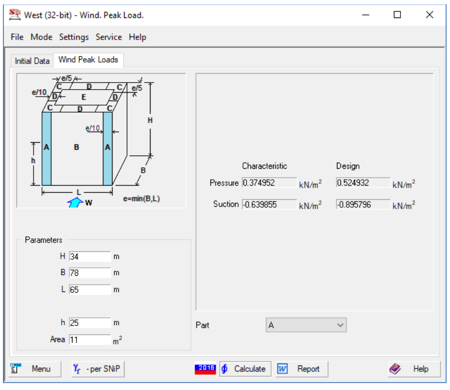


Figure 3.4.6-2. *The* **Wind Loads** *tab*

The **Wind Loads** tab (see Fig. 3.4.6-2) is used to specify which area of the roof or walls will be considered, the dimensions of the building, and the area the load is applied to. It is also necessary to specify the height for wall areas in order to calculate the characteristic and design values of pressure and suction.

3.4.7 Wind (EN 1991-1-4)

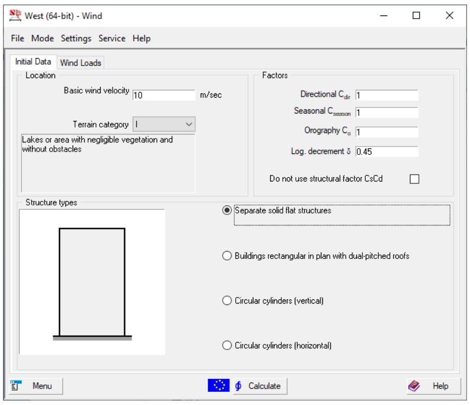


Figure 3.4.7-1. *The* **Initial Data** *tab of the* **Wind** *dialog box*

This mode enables to perform the analysis of wind loads on some of the buildings, structures, and structural members, the models of which are given in EN 1991-1-4. The static value of the wind load normal to the structure surface is determined in all cases. It is assumed that surfaces don't have openings. The two-tab dialog box, **Wind**, is used to enter the initial data and obtain the results.

The first tab, **Initial Data** (Fig. 3.4.7-1) is used to enter the information on the terrain type and the basic wind velocity.

It is also necessary to specify the values of the following factors:

- C_{dir} — directional factor (see Sec. 4.2(2) of EN 1991-1-4);
- C_{season} – season factor (see Sec. 4.2(2) of EN 1991-1-4);
- orography factor C_o (see Sec. 4.3.3 of EN 1991-1-4);
- logarithmic decrements of damping (see Annex F.5 of EN 1991-1-4).

By default, the wind load is determined taking into account the structural factor C_sC_d. Since Sec. 6.2 of EN 1991-1-4 allows to ignore this factor in many cases, the respective checkbox is provided allowing you to do it.

The **sign convention** for the wind pressure values for all types of structures is as follows: wind pressure is considered to be positive if it is directed towards the respective surface, and negative – if it is directed away from the surface.

The application enables to perform the analysis for the following structures:

- free-standing flat solid structures;
- rectangular plan buildings with duopitch roofs;
- extended structures and elements with a cylindrical surface (vertical);
- extended structures and elements with a cylindrical surface (horizontal).

Free-Standing Flat Solid Structures

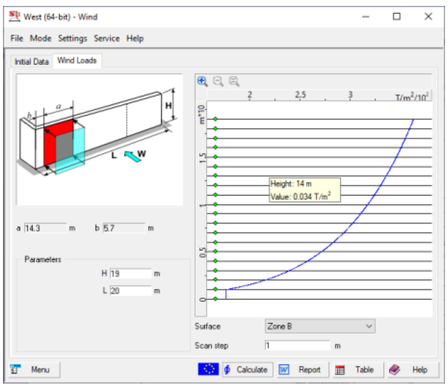
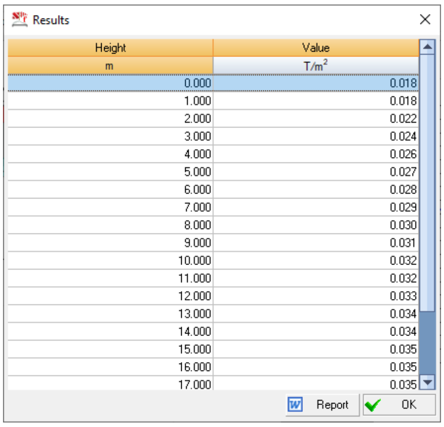


Figure 3.4.7-2. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (Sec. 7.4.1 of EN 1991-1-4), the value of the structure height *H*, its length *L* and the scan step for the result have to be specified in the **Wind Loads** tab (Fig. 3.4.7-2). A zone (A,B,C or D), for which the wind load values have to be calculated, is selected from the drop-down list.

The result window displays a plot of the wind pressure variation with height (and zone sizes are specified as well). This plot can be dynamically digitized, when the application displays the wind load value for the height indicated by the cursor. Heights correspond to the selected scan step.

The green points (marks) on the axis of ordinates indicate the intermediate height values the calculations will be performed for. The spacing between them corresponds to the specified scan step.



Height m	Value T/m²
0.000	0.018
1.000	0.018
2.000	0.022
3.000	0.024
4.000	0.026
5.000	0.027
6.000	0.028
7.000	0.029
8.000	0.030
9.000	0.031
10.000	0.032
11.000	0.032
12.000	0.033
13.000	0.034
14.000	0.034
15.000	0.035
16.000	0.035
17.000	0.035

Figure 3.4.7-3. *The* **Results** *dialog box*

The marks can be deleted and restored again by placing the mouse pointer over them and left-clicking. The wind pressure values at the elevations with marks which have not been deleted are given in the table generated by clicking the **Table** button and is displayed in the **Results** dialog (see Fig. 3.4.7-3).

There are the zooming buttons above the wind variation plot. The plus-sign button serves for zooming in; each time you click this button you enlarge the plot scale twofold. The minus-sign button is used for step-by-step zooming out and functions only after the plus-sign button has been used. You can immediately return to the original size of the plot by clicking the equal button.

Similar buttons are used in all modes where the load variation plots are displayed.

Rectangular Plan Buildings with Duopitch Roofs

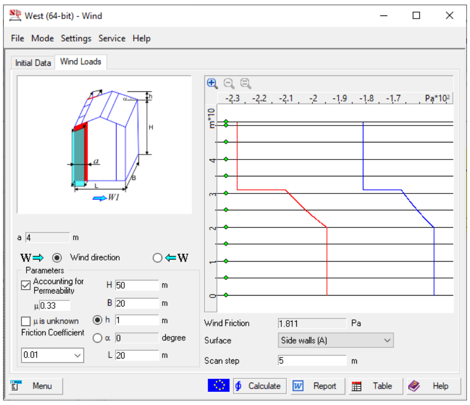


Figure 3.4.7-4. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (Sec. 7.2.5 of EN 1991-1-4) the value of the structure height *H*, its length *L* and the scan step for the result have to be specified in the **Wind Loads** tab (Fig. 3.4.7-4). A zone (A-J), for which the wind load values have to be calculated, is selected from the drop-down list.

If you need to take into account the permeability, check the respective checkbox and specify the value of permeability µ. If the value of **µ is unknown**, you can use the µ is unknown checkbox and the software will perform the analysis according to Note 2 of Sec. 7.2.9 EN 1991-1-4.

If the wind is parallel to the external surface, WEST also calculates the pressure value corresponding to the wind friction. It is necessary to specify the friction coefficients in such cases.

The result window displays a plot of the wind pressure variation with height for walls or the value of wind pressure for different zones of the roof. The values of wind pressure are given for each zone to perform the analysis of the structure as a whole and of its small structural members (with an area of no more than 1 m^2).

EN 1991-1-4 does not provide any guidelines for calculating wind pressure for tall buildings (with a height-to-length ratio of the windward face greater than 5). In such cases you can obtain the wind load values depending on the height.

Extended Structures and Elements with a Cylindrical Surface (Vertical)

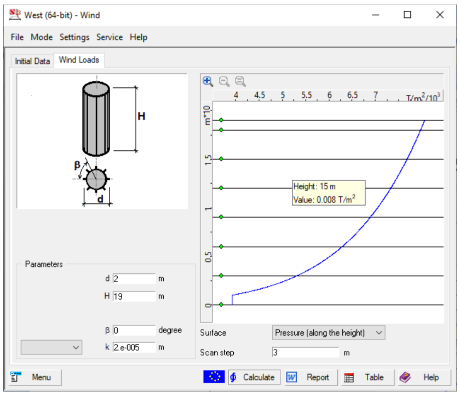


Figure 3.4.7-5. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

When this type of structure is selected (Sec. 7.4.1 of EN 1991-1-4) the dimensions of the structure are entered in the **Wind Loads** tab (Fig. 3.4.7-5). The type of the surface (cylinder in plan, pressure along the height, or load along the height), for which the wind load values have to be calculated, is selected from the drop-down list.

Two plots are used to describe the pressure distribution:

- in a horizontal plane at the reference height, *h* — **Cylinder in plan** in the **Surface** drop-down menu;
- along the vertical line, the position of which is defined by the angle β between the radius passing through this vertical line and the horizontal axis — **Cylinder (along the height)** in the **Surface** drop-down menu.

Depending on the selected plot the result will be:

- wind pressure distribution over the horizontal section at the given height;
- wind pressure variation with height for the given position of the vertical line.

The value of the wind pressure depends on the value *k* (equivalent roughness). You can set this value explicitly or select the surface material from the drop-down list and the application will find the corresponding *k* value according to Table 7.13 of EN 1991-1-4.

If the **Load (along the height)** option is selected, the result is a plot of the load variation with height.

Extended Structures and Elements with a Cylindrical Surface (Horizontal)

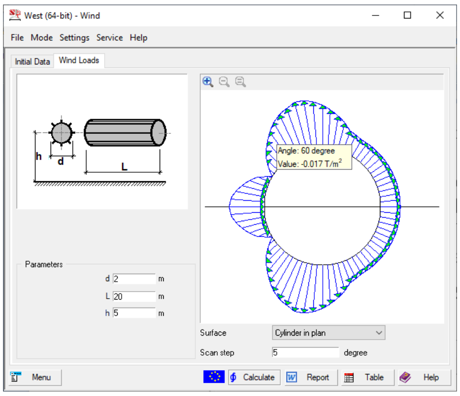


Figure 3.4.7-6. *The* **Wind Loads** *tab of the* **Wind** *dialog box*

The analysis of structures of this type is similar to the previous one. The difference is that, due to the horizontal arrangement, the pressure and load values do not change depending on the considered section of the cylinder; so in both of these cases the result is not a plot but a single number.

3.4.8 Snow. Single-span Buildings

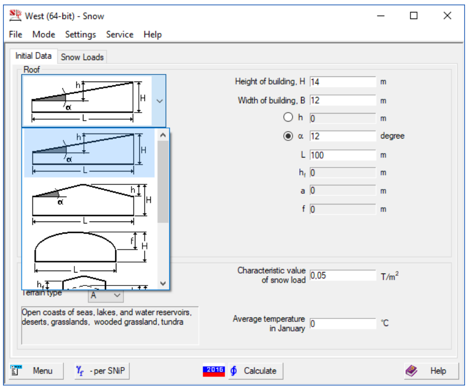


Figure 3.4.8-1. *The* **Initial Data** *tab of the* **Snow** *dialog box*

This mode enables to calculate snow loads on single-span buildings, the roof profiles of which correspond to the models 1, 2, 3 and 10 from Annex 3 of SNiP, KMK or Annex G of DBN, and to the models D.1, D.2.1, D.3.1 and D.10 of Annex D of SP 20.13330.2011, models B.1, B.2 (1), B.3(1), B.13 of Annex B of SP 20.13330.2016.

The **Initial Data** tab (see Fig. 3.4.8-1) is used to enter the information on the roof profile and snow region where the considered structure is located. A snow region is selected from the drop-down list in the **Location** group, and then you have to specify the corresponding characteristic snow load value. To perform the analysis it is also necessary to specify the average temperature in January and the average wind speed in winter (only for the analyses according to SNiP and SP).

Peculiarities of the implementation

According to Sec. 5.3 of SNiP, KMK and Sec. 10.4 of SP 20.13330 "In the cases when the more unfavorable conditions for the behavior of structural members occur under partial loading, the models with the snow load acting on a half or a quarter of the span should be considered". The program always calculates the load over the full span. It is assumed that the cases when the load is applied to a part of the span will be handled by the user himself/herself.

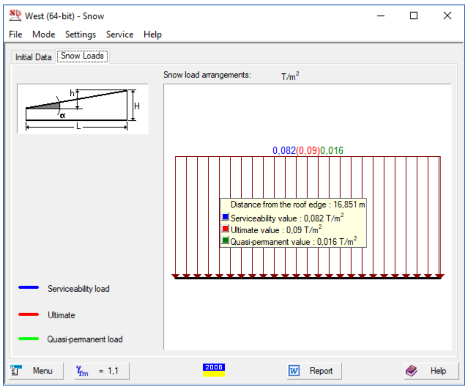


Figure 3.4.8-2. *The* **Snow Loads** *tab (mono-pitched roof)*

All the climatic data will be entered automatically if a snow region has been selected in the **Location** dialog box and the **Apply** button has been clicked. For some codes the button provides access to the list of cities, which contains the refined data on the load value. If you select the desired city and click the **OK** button, the load value will be automatically transferred to the initial data.

The roof profile of the building is selected using the respective functional buttons.

When calculating according to SP 20.13330.2016, the **Extremal Action** marker allows you to obtain results in accordance with the requirements of SP 296.1325800. In this case, it is necessary to introduce safety factor for extreme loads γ_a. Since this involves special effects, results for the SLS are not available.

The checkbox **Ignore snow removal** allows you to refuse to take into account possible snow drift by the wind, for example, in the cases specified in Sec. 10.6 of SP 20.13330.

The analysis starts after clicking the **Calculate** button or selecting the **Snow Loads** tab.

For a mono-pitched roof (see Fig. 3.4.8-2), there is only one variant of the snow load arrangement with the calculated values of the characteristic and design loads (according to SNiP), design loads (according to SP), or serviceability and ultimate loads (according to DBN).

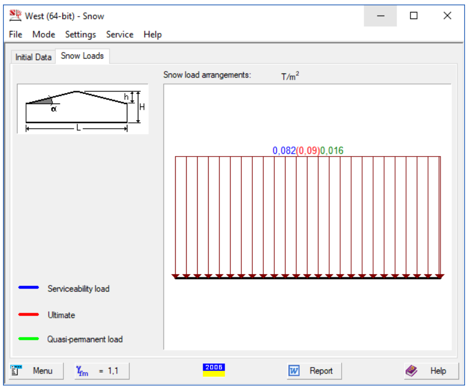


Figure 3.4.8-3. *The* **Snow Loads** *tab (dual-pitched roof)*

For a dual-pitched roof (see Fig. 3.4.8-3), there can be one or two variants of the snow load arrangement (depending on the roof slope) with the calculated values of the characteristic and design loads (according to SNiP, KMK), design loads (according to SP), or serviceability and ultimate loads (according to DBN).

For a vaulted or similar roof (see Fig. 3.4.8-4), there are three variants of the snow load arrangement, and for each one a plot of distribution is displayed.

It should be noted that the plots according to SP 20.13330 (see Fig. 3.4.8-4, *a*) differ from the plots according to SNiP and DBN (see Fig. 3.4.8-4, *b*).

The plots can be scanned (dynamically digitized) in the similar way as described above in the section dedicated to the wind loads.

a *b*

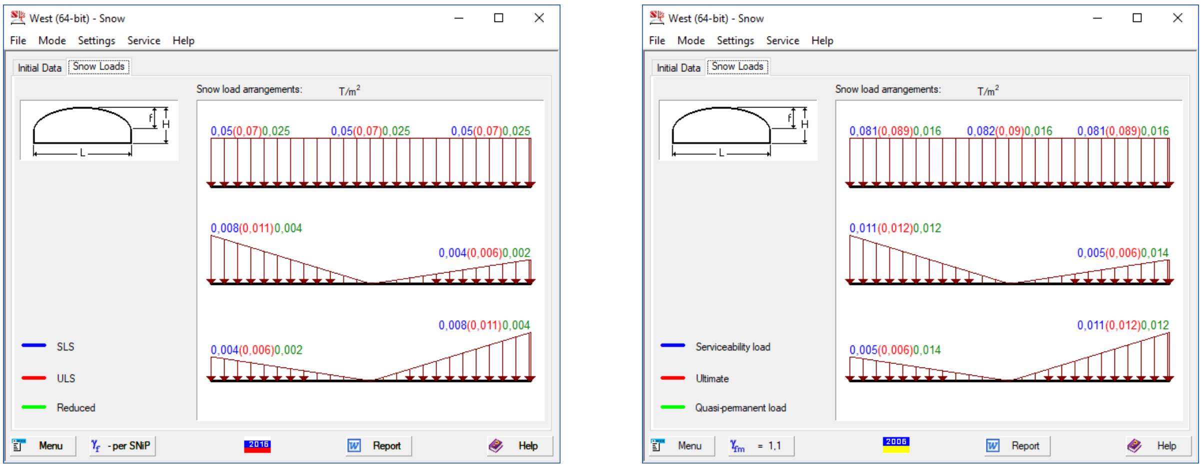


Figure 3.4.8-4. *The* **Snow Loads** *tab (vaulted roof)*: *a* - according to SP; *b* - according to SNiP, KMK and DBN

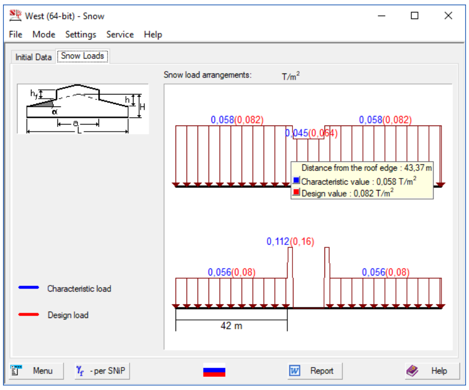


Figure 3.4.8-5. *The* **Snow Loads** *tab (roof with a longitudinal skylight)*

For a roof with a longitudinal skylight, for the roof area that includes the skylight (see Fig 3.4.8-5), there are two variants of the snow load arrangement, and for each one a plot of distribution is displayed.

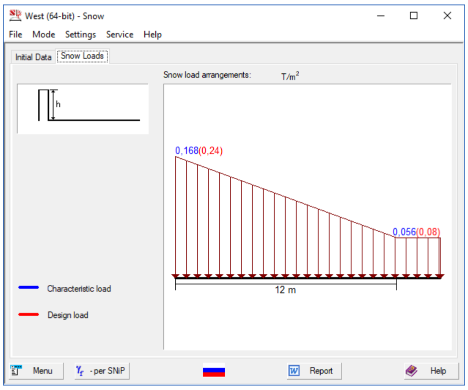


Figure 3.4.8-6. *The* **Snow Loads** *tab (roof with parapets)*

For a roof with parapets (Fig. 3.4.8-6), there is one variant of the snow load arrangement for which the plot of distribution is displayed.

In the case when the snow load near the parapet should not be taken into account, the load plot has zero values.

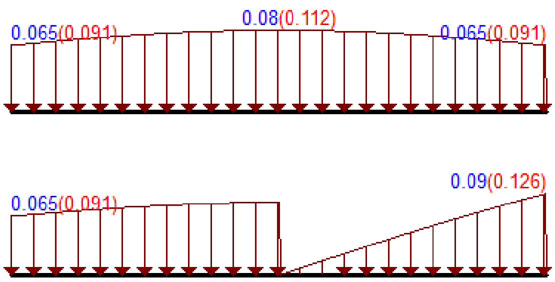
Limitations

For the model 3 (buildings with a longitudinal skylight), the snow load is determined only within the skylight area (area C as designated in SNiP, KMK, SP and DBN).

Note 3 to model 3 according to SNiP and D.3.1 according to SP 20.13330.2011, B.3(1) according to SP 20.13330.2016 is not implemented.

On the Modification No. 1 to SNiP 2.01.07-85 of the Republic of Belarus

The figure for model 2 on page 2 of SNiP specifies μ_1 and μ_2 incorrectly. If what is written there is formally implemented, we will obtain the following physically unrealizable load arrangement:



This is evident from the formula for μ_1=cos 1.8α. In the middle (with α=0) we will not obtain a zero load value.

When calculations are performed by **WeST**, the formulae of SNiP 2.01.07-85 are applied for this case without taking into consideration the above-mentioned modification.

3.4.9 Snow. Two-span Buildings

This mode enables to calculate snow loads on two-span buildings, the roof profiles of which correspond to the models 1b, 4, 5, 6, 7 and 8a from Annex 3 of SNiP, KMK or Annex G of DBN, and to the models D1*b*, D4, D5, D6, D7 and D8*a* of Annex D of SP 20.13330.2011, model B1b, B4, B5, B6, B7, B8a of SP 20.13330.2016.

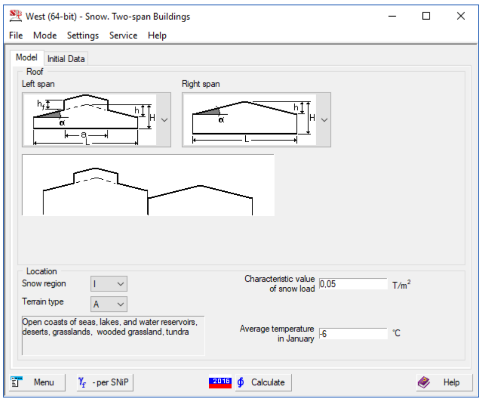


Figure 3.4.9-1. *The* **Model** *tab of the* **Snow. Two-span Buildings** *dialog box*

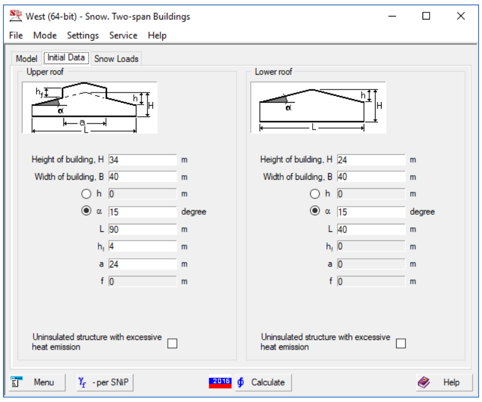


Figure 3.4.9-2. *The* **Initial Data** *tab of the* **Snow. Two-span Buildings** *dialog box*

Two-span buildings are constructed by selecting designs for their left and right spans. Each span can be presented schematically by two mono-pitched roofs (with a slope to the right or to the left), a dual-pitched roof, a dual-pitched roof with a longitudinal skylight, or a vaulted roof. This gives 25 different possible designs of a two-span building.

Every two-span building may have a height difference at the junction of the spans. The height difference is measured from the extreme right point of the left span roof to the extreme left point of the right span roof. In all cases it is assumed that ***the left roof is not lower than the right one***.

The height difference is determined automatically when analyzing the initial data of the selected model.

In the cases when there is no height difference or when it is less than the minimum value specified in the Note 3 to the model 8, an additional load on the right (lower) roof caused by the snow bag is not taken into account.

In the case when a two-span building is formed by two mono-pitched roofs with one-way slopes (shed roofs according to model 4 of Annex 3 of SNiP, KMK, Annex G of DBN, Annex D4 of SP 20.13330.2011, Annex B4 of SP 20.13330.2016), the snow load does not depend on the height difference between the spans.

For two-span buildings (dual-pitched and vaulted), in the case when the height difference is equal to zero or is not taken into account according to Note 3 to the models 8 of SNiP, KMK, D8 of SP 20.13330.2011, B8 of SP 20.13330.2016 of Annexes 2 of SNiP, KMK, D of SP 20.13330.2016, B of SP 20.13330.2016, the snow load is determined from the models 5, 6 of Annexes 2 of SNiP, KMK, D5, D6 of Annex D of SP 20.13330.2011, B5, B6 of Annex B of SP 20.13330.2016.

In all other cases the snow load is determined from the models 8*a* of Annexes 2 of SNiP, KMK, D8*a* of Annex D of SP 20.13330.2011, and B8*a* of Annex B of SP 20.13330.2016.

The analysis determines the snow load on the projections of the right and left spans of the two-span building and (when necessary) the additional load (caused by a snow bag) on the right building.

This mode contains three tabs:

- **Model**;
- **Initial Data**;
- **Snow Loads**.

The **Model** tab is used to select the design of the building and to specify the initial climatic data (common for the whole building). Two-span buildings are constructed by selecting designs for their left and right spans. The selected model of the building (see Fig. 3.4.9-1) is displayed in the central window of the tab.

A snow region is selected from the drop-down list in the **Location** group, and then you have to specify the corresponding characteristic value (according to SNiP), the design value (according to SP) or the characteristic value (according to DBN) of the snow load. To perform the analysis it is also necessary to specify the average temperature in January and the average wind speed in winter (only for the analyses according to SNiP, KMK and SP 20.13330.2011). The average wind speed is not taken into account in SP 20.13330.2016.

All the climatic data will be entered automatically if the location of the construction site has been selected in the **Location** dialog box. In this respect, this mode is identical to the **Snow** mode when determining the snow load on single-span buildings.

The **Initial Data** tab (see Fig. 3.4.9-2) is used to specify the data on the model and design of the roofs of the left and right spans.

The initial data are specified in the same way as in the case of single-span buildings.

The **Snow Loads** tab is similar to the respective tab for single-span buildings.

This tab contains the plots of the design and characteristic values of snow loads on the right and left roofs, and, when necessary, the extra load on the right roof caused by a snow bag (see Fig. 3.4.9-3).

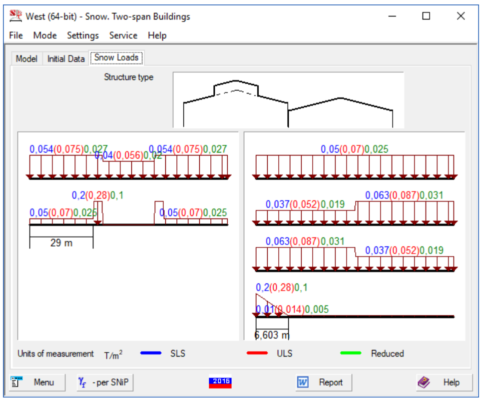


Figure 3.4.9-3. *The* **Snow Loads** *tab (there is an extra load from the snow bag on the right roof)*

Limitations

Model 8a of Annex 3 of SNiP, KMK and Annex G of DBN, and model D8a of Annex D of SP 20.13330.2011, Annex B of SP 20.13330.2016 deal only with buildings (variant *a*) and sheds (variant *b*), whereas the lower roofs perpendicular to the main building (variant *c*) are not considered.

The program does not handle the cases of different-height buildings, the span widths of which (dimension out of the drawing plane) are different for the adjacent spans. This is evident from the fact that the initial data do not include the dimension *a*.

3.4.10 Snow (EN 1991-1-3)

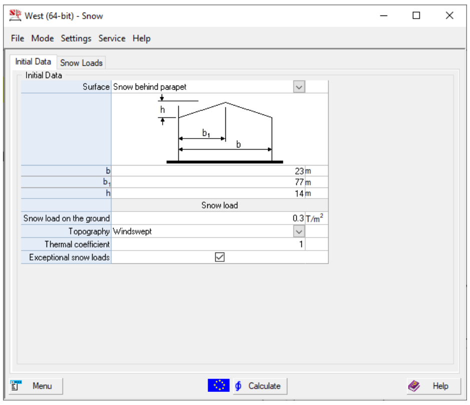


Figure 3.4.10-1. *The* **Initial Data** *tab of the* **Snow** *dialog box*

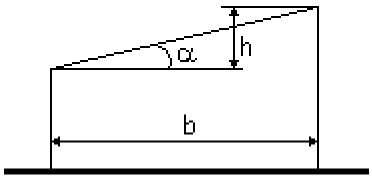
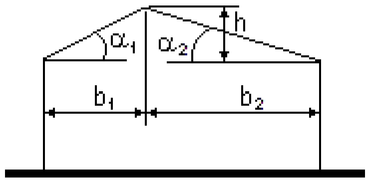
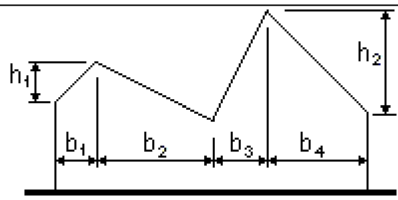
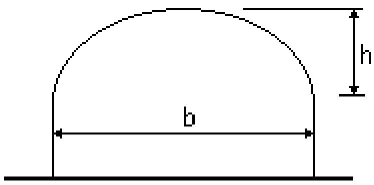
This mode enables to calculate snow loads on the following roof types:

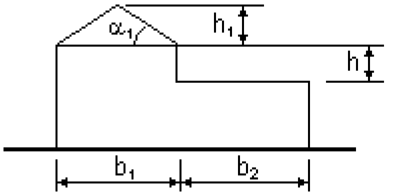
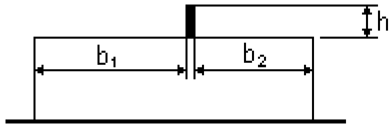
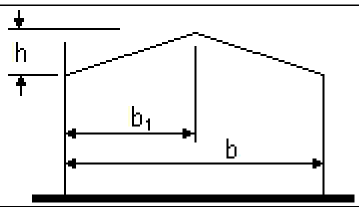
- monopitch roofs;
- pitched roofs;
- two- and multi-span roofs;
- cylindrical roofs;
- roofs abutting to taller construction works;
- drifting at projections and obstructions;
- snow loads on snowguards and other obstacles.

The **Initial Data** tab (Fig. 3.4.10-1) is used to enter the information on the roof profile and snow region where the considered structure is located.

Snow load on the ground and thermal coefficient C_t (see Sec. 5.2(8) of EN 1991-1-3) are specified in the **Snow load** group. **Topography** (windswept, normal or sheltered) is selected from the respective drop-down list. Depending on whether the **Exceptional snow loads** checkbox is checked the analysis will be performed for normal or exceptional snow load.

The roof type is selected from the drop-down list. After selecting it, you have to specify the geometric parameters of the structure:

Roof type	Geometric parameters
Monopitch roofs	
Pitched roofs	
Two- and multi-span roofs	
Cylindrical roofs	

<table>
<tr><td>Roofs abutting to taller construction works</td><td>
</td></tr>
<tr><td>Drifting at projections and obstructions</td><td>
</td></tr>
<tr><td>Snow loads on snowguards and other obstacles</td><td>
</td></tr>
</table>

The analysis starts after clicking the **Calculate** button or selecting the **Snow Loads** tab. As a result, one or more variants of the snow load distribution are calculated according to Sec. 5.3 of EN 1991-1-3.

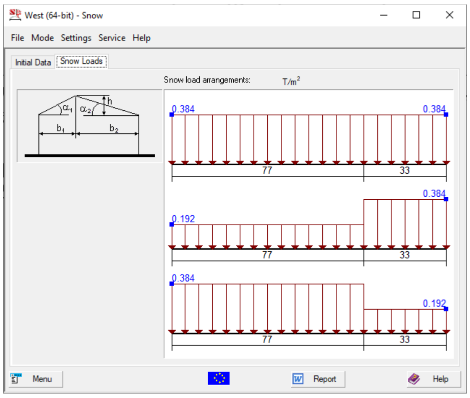


Figure 3.4.10-2. *The* **Snow Loads** *tab*

The respective plots are presented in the **Snow Loads** tab (see Fig. 3.4.10-2). The plots can be scanned (dynamically digitized).

3.4.11 Temperature

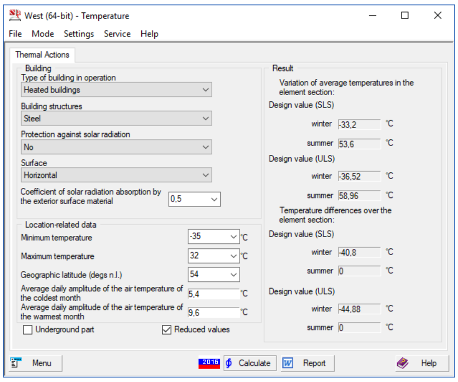


Figure 3.4.11-1. *The* **Thermal Actions** *tab*

The **Temperature** mode (see Fig. 3.4.11-1) enables to calculate the values of the thermal climatic actions in compliance with the instructions of Section 8 of SNiP, KMK, Section 11 of DBN and Section 13 of SP 20.13330.

This mode determines the variation with time of the average temperature of an element Δt as compared with the mounting temperature of the structure, and also a temperature difference across the element section.

All the information on the type and design of a building is selected from the drop-down lists and corresponds to the formulations given in Table 15 of SNiP, KMK, Table 11.1 of DBN or Table 13.1 of SP 20.13330. The data on the location of the construction site can be obtained from the **Location** mode or specified directly in the window of this mode.

The values of the indoor air temperature are assumed to be 22°C during the warm season and 16°C during the cold season.

The default value of the safety factor for load is taken as 1.1.

Checkbox **Reduced values** allows you to calculate the reduced values of the temperature impacts.

When calculating according to SP 20.13330.2016, the **Extremal Action** marker allows you to obtain results in accordance with the requirements of SP 296.1325800. In this case, it is necessary to introduce safety factors for extreme loads $\gamma_{a,min}$, $\gamma_{a,max}$ for minimum and maximum temperatures. Since this involves special effects, results for the SLS are not available.

3.4.12 Temperature (EN 1991-1-5)

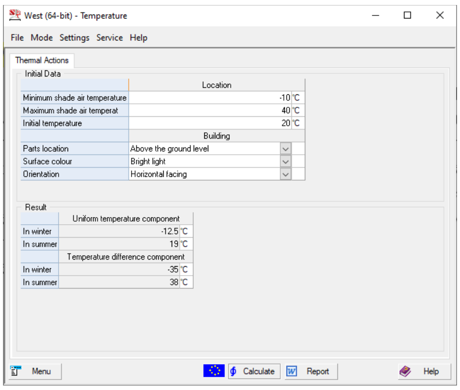


Figure 3.4.12-1. *The* **Thermal Actions** *tab*

The **Temperature** mode (Fig. 3.4.12-1) enables to calculate the values of the thermal climatic actions in compliance with the instructions of EN 1991-1-5. Uniform temperature and temperature difference components (in winter and summer) are determined.

Surface location, color and orientation are selected from the respective drop-down lists. It is also necessary to specify the minimum and maximum shade air temperature values and the initial temperature.

3.4.13 Ice on Cables and Ropes

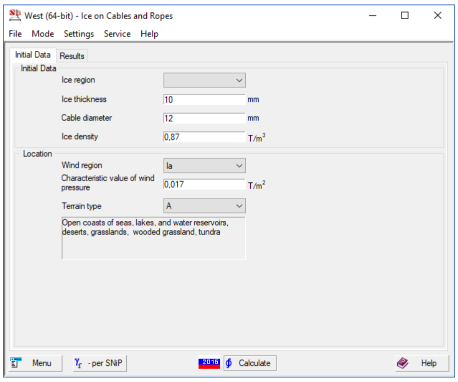


Figure 3.4.13-1. *The* **Initial Data** *tab of the* **Ice on Cables and Ropes** *dialog box* **(according to SNiP and SP)**

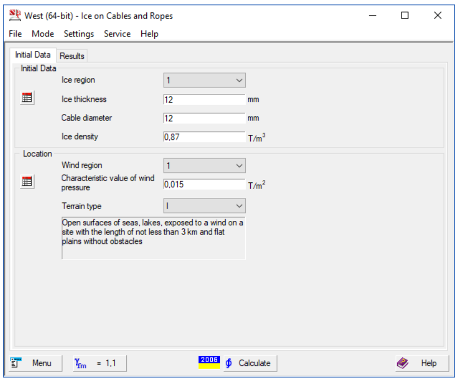


Figure 3.4.13-2. *The* **Initial Data** *tab of the* **Ice on Cables and Ropes** *dialog box* **(according to DBN)**

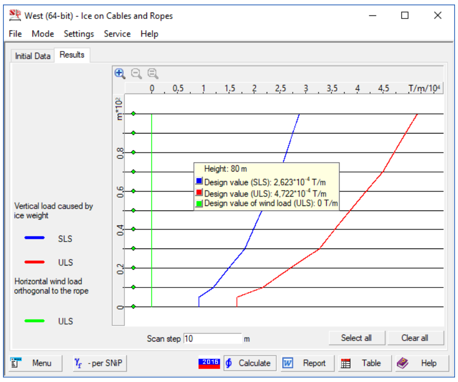


Figure 3.4.13-3. *The* **Results** *tab of the* **Ice on Cables and Ropes** *dialog box* **(according to SNiP and SP)**

This mode enables to determine the ice load on cables and ropes, and implements the requirements specified in Section 7 of SNiP, KMK, Section 10 of DBN and Section 12 of SP 20.13330 as regards the calculation of the linear ice load. The characteristic value of the ice load is calculated per unit length of a cable (rope).

The design (according to SNiP, KMK and SP) and design ultimate (according to DBN) value of the ice load is the product of the characteristic value and the safety factor for load.

For the analyses according to SNiP, KMK and SP, the default value of the safety factor for load is taken as 1.3 in compliance with Sec. 7.3 of SNiP, KMK and Sec. 11.3.4 of SP 20.13330.2011, 1,8 in compliance with SP 20.13330.2016.

For the analyses according to DBN, the safety factor for the ultimate ice load value is determined in accordance with the instructions of Sec. 10.10.

The user can change the value of the safety factor in the edit mode (button).

When calculating according to SP 20.13330.2016, the **Extremal Action** marker allows you to obtain results in accordance with the requirements of SP 296.1325800. In this case, it is necessary to introduce safety factor for extreme loads γ_a. Since this involves special effects, results for the SLS are not available.

In addition to the ice load, the ice-wind load is calculated as well.

For the analyses according to SNiP, the ice-wind load is calculated in compliance with the requirements of Sec. 7.4 of SNiP.

For the analyses according to SNiP, KMK, the initial data on the wind region according to Map 3, and on the ice region according to Map 4 of the mandatory Annex 5 of SNiP, KMK can be specified either directly in the **Initial Data** tab (see Fig. 3.4.13-1) or transferred from the **Location** mode by means common for **WeST**.

For the analyses according to SP, the initial data on the wind region according to Map 3 of SP 20.13330.2011, Map 2 of SP 20.13330.2016 and on the ice region according to Map 4 of SP 20.13330.2011 and Map 3 of SP 20.13330.2016 of the recommended Annex G of SP 20.1333.2011 and Annex F of SP 20.13330.2016 can be specified in the same way as in the case of analyses according to SNiP, KMK.

For the analyses according to DBN, the wind load is calculated in compliance with the requirements of Sec. 10.9. The initial data on the ice-wind region taken according to the map (see Fig. 10.2 of DBN), and the initial data on the ice region taken according to the map (see Fig. 10.4 of DBN) can be specified either directly in the **Initial Data** tab (see Fig. 3.4.13-2) or transferred from the **Location** mode by means common for **WeST**. If DBN is selected, the button provides access to the list of Ukrainian cities, which (based on Annex F of DBN) contains the refined data on the characteristic load value. If you select the desired city and click the **OK** button,

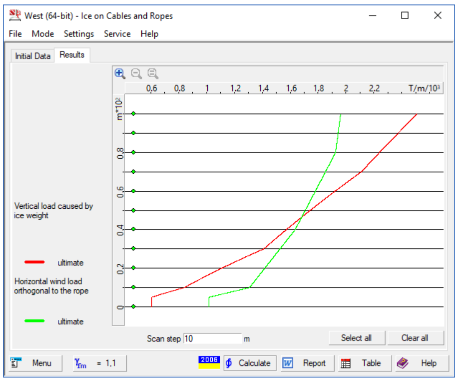


Figure 3.4.13-4. *The* **Results** *tab of the* **Ice on Cables and Ropes** *dialog box* **(according to DBN)**

the characteristic load value will be automatically transferred to the initial data. Moreover, these loads can be further refined within the limits of the regional loads in accordance with Annex F of DBN.

On the **Initial Data** tab the user can specify the values of the ice thickness and the wind load different from those set for the current zone by the design standards.

The characteristic and design values of the ice load, and the wind load on the ice-covered cables (ropes) are determined in the *result of the analysis* according to SNiP, KMK and according to SP (see Fig. 3.4.13-3).

The design ultimate values of the ice load and the wind load on the ice-covered cables (ropes) are determined in the *result of the analysis* according to DBN (see Fig. 3.4.13-4).

The ice-wind load is determined under the assumption that a cable (rope) is located in the plane perpendicular to the wind flow. When determining the wind load, the application allows for an increase of the cable (rope) diameter due to the ice cover.

The results of the analysis are given in the graphical form on the respective tab. The numeric results can be obtained by scanning the plots.

Limitations

The ice load is calculated for cables (ropes) running at the height not more than one hundred meters above the ground.

3.5 Design Codes Implemented by WeST

Mode	References to sections of codes and standards			
	for the analysis according to SNiP 2.01.07-85, KMK 2.01.07-96 and SNiP 2.01.07-85*	**for the analysis according to DBN B.1.2-2:2006**	**for the analysis according to SP 20.13330.2011**	**for the analysis according to SP 20.13330.2016**
Density	Sec. 2.1, 2.2	Sec. 5.2, 5.3	Sec. 7.1, 7.2	Sec. 7.1, 7.2
Location	Annex 5, SNiP 2.01.01-82	Sec. 8.5, 9.6, 10.7, 10.9, Annex G	Annex G	Annex F
Safety factors for loads	Sec. 2.2, 3.4, 5.7, 6.11, 7.3, 8.7	Sec. 5.2, 5.3, 6.8, 6.9, 8.11, 8.12, 9.14, 9.15, 10.10, 10.11, 11.8	Sec. 7.2, 8.3.4, 8.4.5, 10.,12, 11.1.12, 12.5, 13.8	Sec. 7.2, 8.1.4, 8.4.5, 10.12, 12.5, 13.8
Safety factors for importance			Sec.5.2 GOST 27751	
Self-weight	Sec. 2.1, Annex 5, SNiP 2.03.13-88	Sec. 5.2, 5.3, Annex 5, SNiP 2.03.13-88, DBN B.2.6-14-97	Sec. 7.1, Annex 5, SNiP 2.03.13-88	
Deflection limits	Sec. 10.7, 10.11, 10.12	Sec. 5.1, 6.1, 7.1 DSTU B V.1.2-3:2006	Section 15, Annex G	Section 15, Annex E
Traffic loads			Sec. 8.4.1	Sec. 8.4.1
Temporary loads	Sec. 3.5–3.9	Sec. 6.5–6.9	Section 8.2	Section 8.2
Wind	Sec. 6.1, 6.3–6.6, Annex 4 (models 1, 2, 3,	Sec. 9.2–9.10, Annex I (models 1, 2, 3,	Sec. 11.1.1, 11.1.3 – 11.1.7,	Annex C (Sections C11-C13, C.1.9 –

Mode	References to sections of codes and standards			
	for the analysis according to SNiP 2.01.07-85, KMK 2.01.07-96 and SNiP 2.01.07-85*	**for the analysis according to DBN B.1.2-2:2006**	**for the analysis according to SP 20.13330.2011**	**for the analysis according to SP 20.13330.2016**
	9, 11, 12*б*, 14, 18)	9, 11, 12*б*, 14, 18)	Annex E (Sections E.1.1 – E.1.3, E.1.9 – E.1.12)	C.1.12)
Wind. Pulsations	Sec. 6.1–6.5, 6.7–6.10		Sec. 11.1.8 – 11.1.10	Sec. 11.1.8 – 11.1.10
Total wind		Sec. 9.4–9.13		
Snow. Single-span buildings	Sec. 5.1–5.7, Annex 3 (models 1, 2, 3, 10)	Sec. 8.10–8.12, Annex F (models 1, 2, 3, 10)	Section 10, Annex D (models D1, D2, D3, D10)	Section 10, Annex B (models B1, B2, B3, B13), Sec. 6.5 SP 296.1325800
Snow. Two-span buildings	Sec. 5.1–5.7, Annex 3 (models 1, 4, 5, 6, 7, 8*a*)	Sec. 8.10–8.12, Annex F (models 4, 5, 6, 7, 8*a*)	Section 10, Annex D (models D1, D4, D5, D6, D7, D8*a*)	Section 10, Annex B (B1, B4, B5, B6, B7, B8a), Sec. 6.5 SP 296.1325800
Temperature	Sec. 8.1–8.7, GOST 12.1.005-88, Sec. 8.1–8.7, SNiP 2.01.01-82	Sec. 11.1-11.8	Section 13, GOST 12.1.005-88	Section 13, GOST 12.1.005-88, Sec. 6.8 SP 296.1325800
Ice on Cables and Ropes	Sec. 7.2–7.4	Sec. 10.2-10.5, 10.7-10.13	Sec. 12.2 – 12.3	Sec. 12.2 – 12.3, Sec. 6.7 SP 296.1325800

4. Kristall

Kristall enables you to perform structural analysis and various checks of members and joints of steel structures for compliance with requirements of one of the following design codes

- SNiP II-23-81*;
- ShNK 2.03.05-13;
- SP 53-102-2004;
- SP 16.13330.2011 (revised and updated edition of SNiP II-23-81*);
- SP 16.13330.2017 (revised and updated edition of SNiP II-23-81*);
- DBN B.2.6-163:2010;
- DBN B.2.6-198:2014;
- SNiP RK 5.04-23-2002;
- STR 2.05.08:2005.

The design forces are supposed to correspond to loads defined by SNiP 2.01.07-85* “Loads and Actions” (or DSTU B B.1.2-3:2006 when DBN B.2.6-163:2010 or DBN B.2.6-198:2014 is selected). Rules for choosing design combinations of forces implemented in the software comply with the same document as well. If the selected design code is SP 16.13330, then loads will be combined according to the requirements of SP 20.13330.

The following documents were used when creating the software:

- state standards related to SNiP, SP and DBN;
- manual to SNiP II-23-81* on Design of Steel Structures [7];
- SP 294.1325800.2017 Steel Structures. Design Rules.

A particular **Kristall** submodule enables to perform checks of members and joints for compliance with the requirements of Eurocode 3. It can be useful for experts who develop design documentation ordered by European companies. *EN 1993-1-1. Eurocode 3: Design of Steel Structures. — Part 1.1: General rules and rules for buildings* and *EN 1993-1-5. Eurocode 3: Design of Steel Structures. — Part 1.5: Plated structural elements* define checks of members of steel structures, and *EN 1993-1-8. Eurocode 3: Design of Steel Structures. — Part 1.8: Design of joints* is used for the checks of joints of steel structures. A separate chapter is devoted to this branch of the program (see Section 5).

Kristall enables to perform various checks of structural members and joints the sizes of which are predefined by the designer, i.e. the application works in the examination mode. However, for some cases the application has a mode for selecting a cross-section according to the requirements of the ultimate limit state. The check of the selected section for the serviceability limit state should be performed by the user in the examination mode of this section.

The application provides reference information on assortments of rolled profiles and bolts, recommendations of codes. The application has special reference modes to implement these features (see below). In some cases the dialog box has the button  clicking which displays additional reference information.

Since the codes provide recommendations not for all possible designs, the application will not be able to perform a required check for some cases. In most of such cases the application control system will not allow such a request. A warning will be displayed, and the report will contain a message concerning this situation.

According to their structure, prerequisites, checks, structural limitations and recommendations SNiP, SP and DBN are quite close documents. Since the set of problems which can be solved in **Kristall** is the same for these documents, all the general references to SNiP II-23-81* found in the text should be treated as similar references to ShNK 2.03.05-13, SP 53-102-2004, SP 16.13330, DBN B.2.6-163:2010, or DBN B.2.6-198:2014.

4.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 4.1-1), with a set of buttons for selecting a working mode. These modes can be subdivided into the following groups

- reference modes;
- auxiliary modes for designing steel structures;
- modes for checking the sections of load-bearing structures and joints for compliance with the codes;
- design modes for designing simple structural members.

A detailed description of each mode is given below. Their brief characteristic is given here.

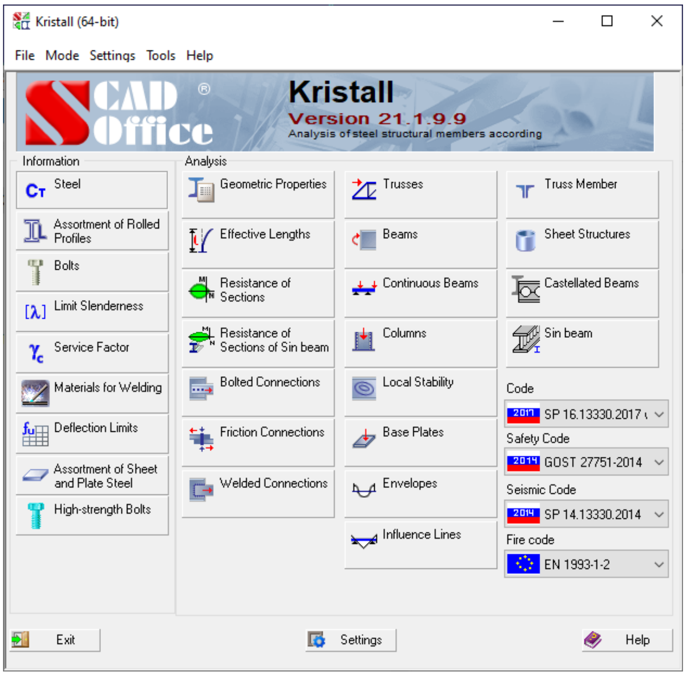


Figure 4.1-1. *The main window*

The reference modes include:

- **Steel —** implements recommendations from Table 50* of SNiP II-23-81* on selection of steel grades (Annex C of SP 53-102-2004 and SP 16.13330, and Annex F of DBN B.2.6-163:2010, and Annex D of DBN B.2.6-198:2014) and uses an approach described in [4] and in DBN B.2.6-163:2010. The mode also gives information on correspondence of steel grades defined by GOST 27771 to specific steel grades defined by GOST or standard specifications, and provides reference data on the mechanical properties;
- **Assortment of Rolled Profiles** — enables to browse through assortments;
- **Bolts** — enables to browse through an assortment of bolts and a list of standards and codes used for the design of bolted joints;
- **Limit Slenderness** — provides recommendations from design codes for limit slenderness of members in tension and in compression;
- **Service Factors** — is used to browse and select values of service factors (γ_c) for structures, elements, bolted joints and high-strength bolted joints according to SNiP, SP, ShNK or DBN;
- **Materials for Welding** — implements requirements of Section 2 of SNiP II-23-81*, Section 6 of SP 53-102-2004, Section 5 of SP 16.13330, Section 1.2 and Annex G of DBN B.2.6-163:2010, Section 6.2.2 and Annex E of DBN B.2.6-198:2014, ShNK 2.03.05-13 for selection of materials for welded joints and for selection of the design strength values for welded joints;
- **Deflection Limits —** provides tables from SNiP 2.01.07-85* “Loads and Actions” (or DSTU B.V.1.2-3:2006 when working with DBN B.2.6-163:2010 or DBN B.2.6-198:2014) with limitations on deflections of structural members;
- **Assortment of Sheet and Plate Steel —** provides information on assortments of sheet and plate steel;

- **High-Strength Bolts —** provides sizes and properties of high-strength bolts, nuts, and washers.

The auxiliary modes include:

- **Envelopes** – enables to determine unfavorable combinations of multiple loads acting on bending members, and to plot the envelope diagrams of moments and shear forces;
- **Influence Lines** – enables to plot influence lines for multi-span continuous beams of constant cross-section;
- **Geometric Properties** — enables to calculate the geometric properties of a given cross-section;
- **Effective Lengths** – implements recommendations from Tables 11, 12, 13*, and 17*a* of SNiP II-23-81* (Tables 25, 28, 29 of SP 53-102-2004, Tables 25, 30, 31 of SP 16.13330, Tables 1.9.1, 1.9.2, 1.9.4, 1.9.8 of DBN B.2.6-163:2010, Tables 13.1, 13.2, 13.4, 13.8 of DBN B.2.6-198:2014) and from EN 1993-1-1 Eurocode 3.

The modes described below are functional, used to perform checks of structural designs of steel structures and connections for compliance with strength, stability, and flexibility requirements of design codes. The modes are also capable of performing partial checks, if those are of particular interest (such as the investigation of the load-bearing capacity area by plotting the interaction curves). They include:

- **Resistance of Sections** – enables to determine the utilization factors of restrictions for any cross-section type available in the application database, under the action of arbitrary forces; and to plot the interaction curves for any admissible combination of the pairs of forces;
- **Bolted Connections** – enables to determine the utilization factors of restrictions for a certain set of structural designs of joints with bolts with non-controlled tightening; and to plot the interaction curves for any admissible combination of the pairs of forces;
- **Friction Connections** — the mode is similar to the previous one but works with a different set of structural designs of joints with high-strength bolts with non-controlled tightening;
- **Welded Connections** – enables to determine the utilization factors of restrictions for a certain set of structural designs of joints with welded connections; and to plot the interaction curves for any admissible combination of the pairs of forces;
- **Local Stability** – enables to check the local stability of webs and flange plates of column, beam and beam-column structural members; the mode does not consider crane beams or beams with a web reinforced by longitudinal stiffeners.

Finally, a complex check and selection of the cross-section can be performed for some types of the most widespread structural members. These include:

- **Trusses** — this mode enables to perform all the necessary strength and stability checks of the truss members for the most common structural designs. The work begins with determining the design values of internal forces caused by the given vertical external loads. It is possible to find the cross-section;
- **Truss Member** — this mode enables to perform all the necessary strength and stability checks for a separate bar element which is a member of a truss structure. It is possible to find the cross-section;
- **Beams** — this mode is similar to the previous one, but it deals with I-section (both welded and rolled) or channel single-span beams with various boundary conditions;
- **Continuous Beams** — this mode implements the same functions as the **Beam** mode, but in application to a multi-span structure (up to five spans are allowed) which can have cantilevers at its ends. The available cross-sections include rolled or welded I-beams and channels;
- **Columns** — this mode is similar to the previous one, but it deals with columns or posts of various cross-sections;
- **Base Plates** — this mode deals with parts of a column base plate with various types of bearing;
- **Sheet Structures** — this mode enables to determine the utilization factors of restrictions for strength and stability of sheet structures of one of three types: round cylindrical or conical shells, or cylindrical panels.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

The main window also contains a number of buttons which are common controls for all working modes. These include the **Exit**, **Settings**, and **Help** buttons. The **Help** and **Exit** buttons perform functions common for a Windows application: providing reference information and finishing the current session respectively.

The **Settings** button invokes the **Application Settings** dialog box where you can customize the program.

Design codes can be selected from the respective list. Information on the selected code is displayed in the bottom left corner of the active mode window.

The **Menu** button switches from any mode to the main window.

4.2 Settings

The **Application Settings** dialog box is invoked by the **Settings** button of the main window or from the respective menu. In addition to the general settings described in Section 3, in **Kristall** this dialog box also contains the **Material** tab.

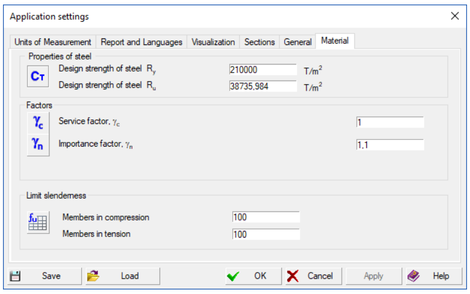


Figure 4.2-1. *The* **Material** *tab*

The **Material** tab (Fig. 4.2-1) enables you to specify properties of steel (values of R_y and R_u), limit slenderness, and values of the following factors:

- service factor for the structure (γ_c);
- service factor for members connected by bolts (γ_{ce});
- importance factor (γ_n).

It is advisable to follow the recommendations of the design codes when specifying this information. However, the deviations are not detected or prohibited, so the user is actually free to do as he likes. If the deviation is undesirable, the data should better be specified in the modes **Steel**, **Limit Slenderness**, and **Service Factor** (see below). From there, the data will be automatically transferred to the tab described here. In such a case the tab provides only reference information.

The values of the limit slenderness can be modified by the user if he thinks that the recommendations of the respective mode are inaccurate or insufficient. The values entered here must be positive integers.

4.3 Creating Cross-Sections

The action of selection of a cross-section for a structural member is common for most working modes of the application. To avoid unnecessary repetitions, these operations are described here separately.

The application works with various types of cross-section shown in Fig. 4.3-1. They include rolled and roll-formed profiles (rectangular pipes are considered in two possible orientations, horizontal and vertical), three types of sections welded from sheets/plates, and built-up sections from rolled profiles. Moreover, the **Resistance of Sections** mode enables to check (for strength) any section created in **Section Builder**, **Consul**, or **Tonus**. It should be noted that when a section created by **Section Builder** or **Tonus** is recognized as a standard section for **Kristall** (welded I-section, welded hollow section or double angles), an application generates a respective message and treats this section as a standard one.

If the section is a rolled or roll-formed profile, or if it contains one, it can be selected from a tree-like profile database (Fig. 4.3-2). First level of the tree is used to choose a catalogue of rolled profiles from which to select the desired section. It contains only the catalogues included in the **In Use** list.

Second level (Fig. 4.3-3) enables you to select a group of rolled profiles of one type (such as I-beams, channels, angles, etc.). The list of accessible profile groups is defined by the selected cross-section type. For example, if you choose the first section type, only the **Equal Angles** and **Unequal Angles** will be accessible, and if the last section type is selected, **I-beams with parallel flanges** or **I-beams with sloped inner flange surfaces** groups of profiles will be accessible.

Third level (Fig. 4.3-4) enables to select a particular profile which will be used in the cross-section of the member.

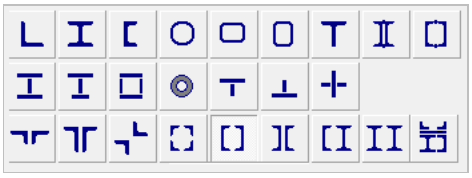


Figure 4.3-1. Types of cross-sections

Figure 4.3-2. First level of the tree (selection of a catalogue)

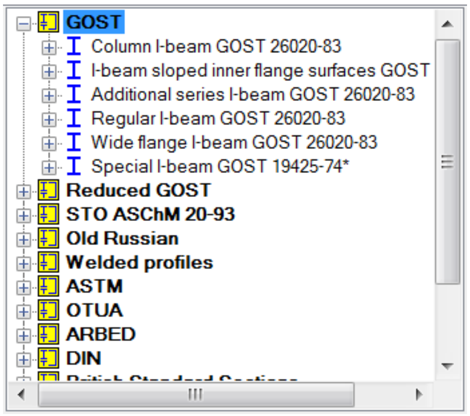


Figure 4.3-3. Second level of the tree (selection of a group)

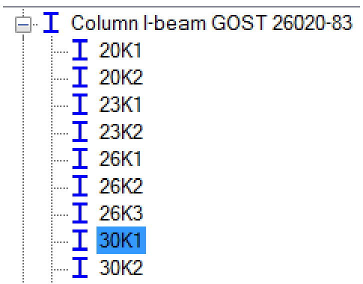


Figure 4.3-4. Third level of the tree (selection of a profile)

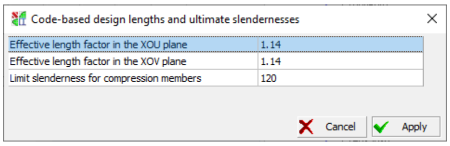
If you have selected a single angle section, you can also specify the type of the structural member in the drop-down list:

You can choose from the following types:

- General type element
- The leg member of a space lattice structure with symmetrical bracing
- The leg member of a space lattice structure with unsymmetrical bracing
- The bracing member of a space lattice structure
- The member of a space structure

Based on the selected type, software will automatically perform the stability analysis in the principal axes or structural axes according to the requirements of the selected design codes.

Clicking the button invokes the dialog box which provides the standard values of the effective lengths and limit slenderness recommended by the codes. Clicking the **Apply** button will transfer this information to the calculation parameters.



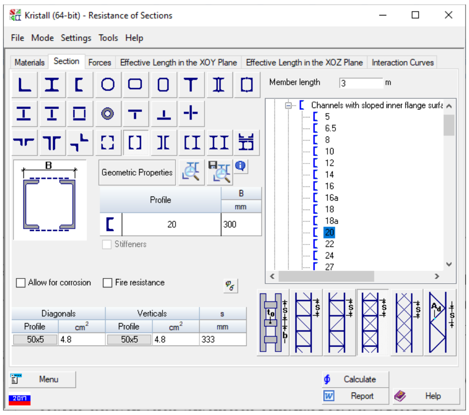


Figure 4.3-5. Specifying a lattice

The type of the connecting lattice for a built-up section with chords connected by lattices can be selected using the respective buttons. Once it is selected, a table appears (Fig. 4.3-5), which includes buttons for selecting sections of the lattice members and text fields for specifying properties of the lattice.

The limitations imposed on the properties of the lattices are listed in Tables 4.3-1 and 4.3-2.

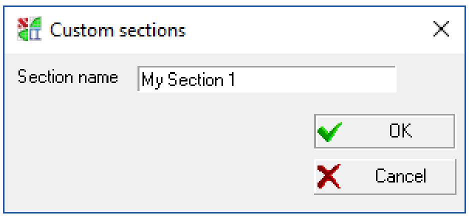


Figure 4.3-6. *The* **Custom sections** *dialog box (specifying a section name)*

Built-up (welded) cross-sections defined in the application can be saved in a custom user database. This can be done by clicking the **Save cross-section in the user database** button — .

Custom sections dialog box will appear where you can specify a name for the section you want to save (Fig. 4.3-6). Since the application does not verify the uniqueness of the names used, it is the user who has to take care of it.

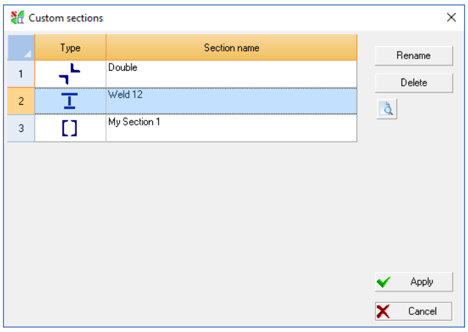


Figure 4.3-7. *The* **Custom sections** *dialog box*

To access the sections from the user database click the **Load cross-section from the user database** button — . The list of sections is displayed in the **Custom sections** dialog box (Fig. 4.3-7).

If duplicate names are detected in the database, use the **Rename** button to change the name of the respective profile**.**

Any of the sections listed in the **Custom sections** dialog box can be selected for further work. To do this, highlight the respective row in the table and exit the dialog by clicking the **Apply** button.

It should be noted that the program does not allow to select a previously saved section, if this section type is *not applicable* in a certain mode. For example, having saved a section from two I-beams in the database, you will not be able to load it in the **Beam** mode.

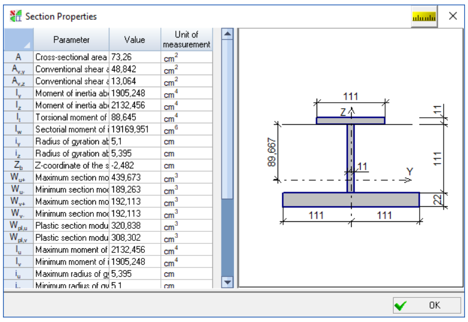


Figure 4.3-8. *The* **Section properties** *dialog box*

Properties of the selected section can be browsed in the **Section properties** dialog box (Fig. 4.3-8) which is invoked by the **Geometric Properties** button. The section is displayed to scale, principal axes of inertia and main sizes are indicated.

The geometric properties are calculated by the application with certain accuracy. Therefore some of them may be slightly different from the reference data given in catalogues.

If the section includes rolled profiles, the catalogues containing data on these profiles must be included in the **In use** list.

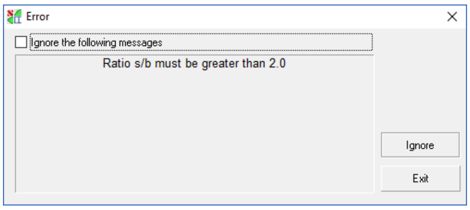


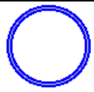
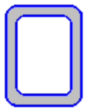
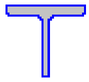
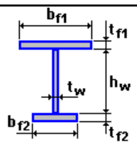
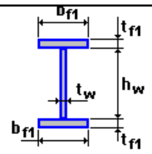
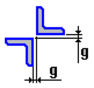
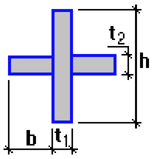
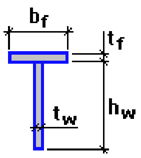
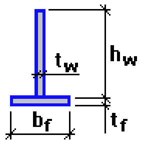
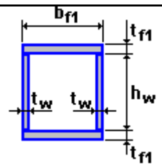
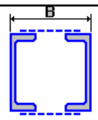
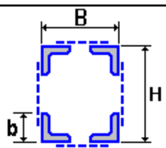
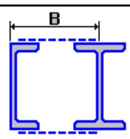
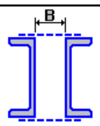
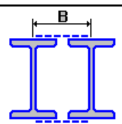
Figure 4.3-9. *An* **Error** *message*

The application performs an automatic check of the section design. The list of limitations is given below in Tables 4.3-1 and 4.3-2. When some of the limitations are violated, an error message with the description of the error will appear (see an example in Fig. 4.3-9). Moreover, clicking the button in the respective dialog boxes will display the sizes limitations.

In some cases the application allows you to ignore the warning that some limitations are violated (*Ignore* button). However, the negative consequences of this decision will not be analyzed.

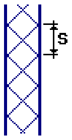
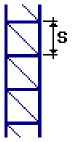
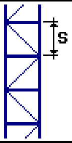
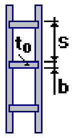
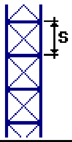
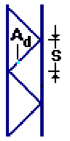
In cases when an invalid numerical value is specified for a parameter, the error message will say **Invalid data**. Such a check is performed in all working modes.

Table 4.3-1. Limitations of the section sizes

Section	Limitations	Section	Limitations
	From available databases only		From available databases only
	From available databases only		From available databases only
	From available databases only		From available databases only
	$b_{f1} / t_{f1} \geq 5$; $b_{f2} / t_{f2} \geq 5$; $h_w / t_w \geq 5$		$b_{f1} / t_{f1} \geq 5$; $h_w / t_w \geq 5$
	$0{,}8t \leq g \leq 2t$		$0{,}8t \leq g \leq 2t$
	$0{,}8t \leq g \leq 2t$		$b / t_2 \geq 5$; $h / t_1 \geq 5$
	$b_f / t_f \geq 5$; $h_w / t_w \geq 5$		$b_f / t_f \geq 5$; $h_w / t_w \geq 5$
	$b_{f1} / t_{f1} \geq 5$; $h_w / t_w \geq 5$; $5 \geq b_{f1}/ h_w \geq 0{,}2$		$B/b_{fc} \geq 3$
	$B/b \geq 4$; $H/b \geq 4$; $2.0 \geq B/H \geq 0{,}5$		$B/b_{fd} \geq 3$ Channel and I-beam from the same rolled profile catalogue and of about the same height
	$B/b_{fc} \geq 2$		$B/b_{fd} \geq 2$

Notes:
1. The flange width of rolled profiles is denoted by b_{fc} for channels and by b_{fd} for I-beams.
2. The angle thickness is denoted by t.

Table 4.3-2. Limitations of the lattice properties

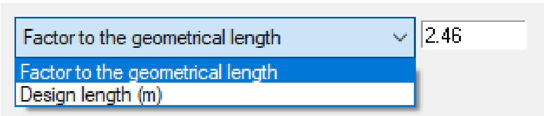
Design	Limitations	Design	Limitations
	$s \geq h$; $A_d < A$		$s \geq h$; $A_v < A$; $A_d < A$
	$s \geq h$; $A_d < A$		$s \geq 2b$
	$s \geq h$; $A_v < A$; $A_d < A$		$s \geq h$; $A_d < A$

Notes:

h is the distance between the chord axes;
A is the chord area;
A_d is the diagonal area;
A_v is the vertical area.

4.4 Specifying the Effective Lengths

It is necessary to specify the data on the effective lengths of structural members in some modes. In many cases you can specify it either in the form of the effective length factor or the effective length, which can be selected from the drop-down list next to the text field:



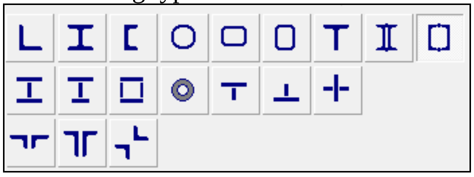
4.5 Fire Resistance

The fire resistance analysis of individual unprotected elements of steel structures is implemented in **Kristall**. The check is performed by the critical temperature method according to DSTU-N B B.2.6-211:2016 (the recommendations of this standard are similar to EN 1993-1-2:2005). The method assumes that when the structure is heated by a fire, the temperature of the steel is uniformly distributed throughout the structure and its critical value is determined by the reserve of strength of the considered element. This reserve of strength is characterized by the load-bearing capacity utilization factor μ_0, the physical meaning of which corresponds to the coefficient K_{max}. Taking into account μ_0 the critical temperature is calculated by the following formula:

$$\theta_{a,cr} = 30{,}19 \ln\left(\frac{1}{0{,}967\mu_0^{3{,}833}} - 1\right) + 482.$$

The program determines the value of the load-bearing capacity utilization factor K_{max} from the combinations of loads which includes only the characteristic values of permanent and long-term loads at t = 0 using characteristic values of the material strength properties (see, for example, СП 296.1325800.2017).

The fire resistance analysis is implemented in the following modes: *Resistance of Sections, Columns, Beams, Continuous Beams, Truss Member* for the following types of sections:



Characteristic values of the permanent and long-term loads are taken into account only.

The factor for safety factor for loadings and factor for sustained load has to be specified for each load case when specifying the initial data in the *Columns, Beams, Continuous Beams* modes. The safety factor for loadings and factor for sustained load or characteristic long-term values of internal forces can be specified in the *Resistance of Sections* and *Truss Member* modes.

The critical temperature, time to reach this temperature, and the air and steel temperature-time curves are obtained in the result of the analysis. Moreover, the software outputs the reduced thickness (the ratio of the cross-sectional area to the heated perimeter) and the proper fire resistance limit without fire protection (the time it takes to reach the temperature of 500°C in a standard fire, which causes significant change in strength and deformability — see Table 6 of the *Manual for determining the limits of fire resistance of building structures, fire hazard parameters of materials. Procedure for the design of fire protection.* M. 2013 or Annex B STO APCC 11251254.001-018-03). The results are output in the **Fire Resistance** tab (Fig. 4.5-1).

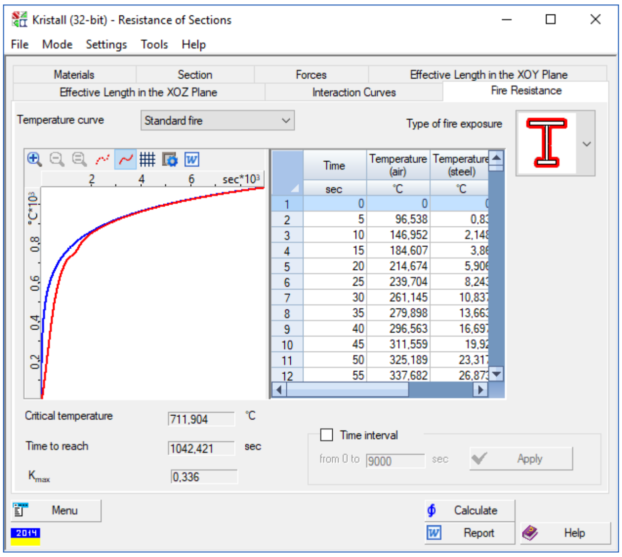


Figure 4.5-1. *The* **Fire Resistance** *tab*

The analysis is performed for one of the following types of the structural heating (Fig. 4.5-2):

- standard fire;
- hydrocarbon fire;
- external fire;
- smoldering fire.

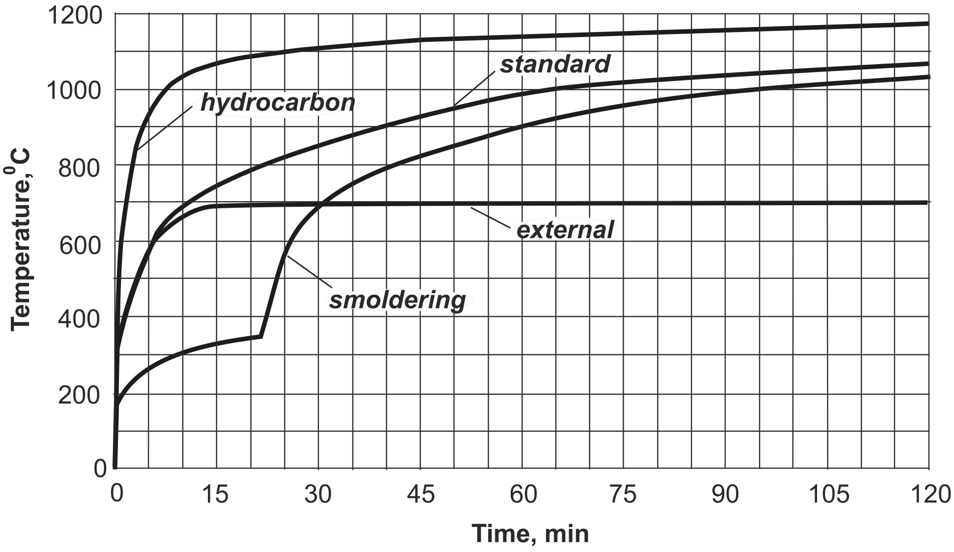


Figure 4.5-2. *Temperature-time curves*

The type of fire is selected from the respective drop-down list in the **Fire Resistance** tab. The analysis can be performed for the following types of fire exposure:

- heating from four sides for all cross-sections;
- heating from three sides for all sections except for angles and hollow sections.

The temperature increase of steel at heating is calculated taking into account the change in the thermal conductivity and the specific heat of steel (Fig. 4.5-3).

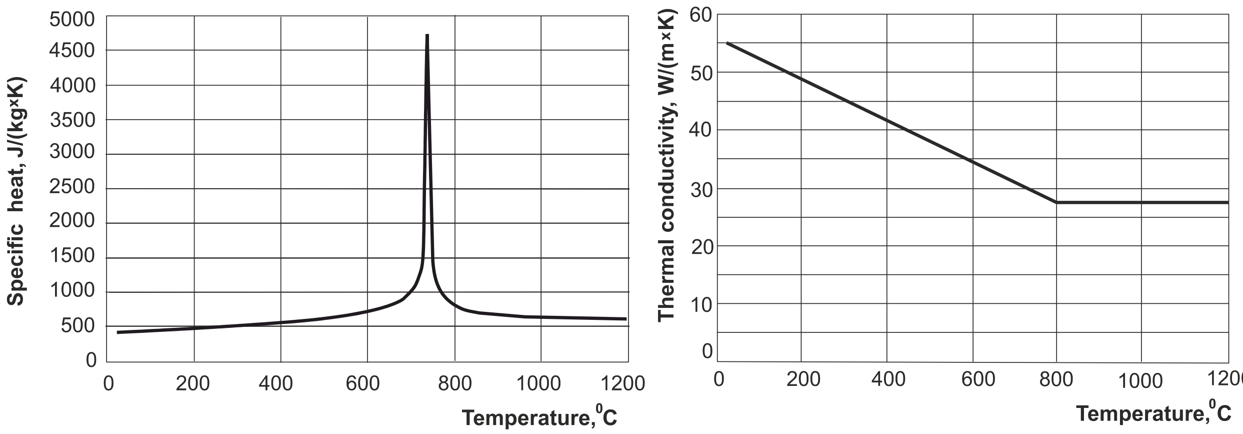


Figure 4.5-3. *Variation of the thermal properties of steel*

4.6 Addvanced Settings

In many modes there is a button next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional restrictions for selecting steel sections (Fig. 4.6-1).

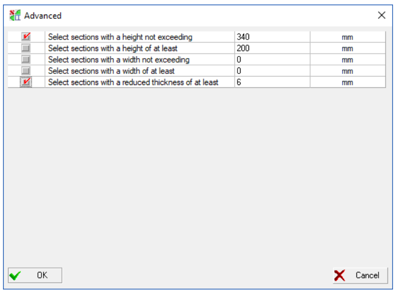


Figure 4.6-1. *The* **Additional Settings** *dialog box*

In particular, you can specify the restrictions on the height, width and reduced thickness of a section (if the fire resistance analysis is enabled).

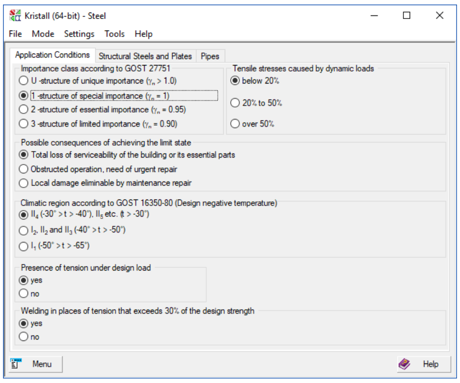
If the analysis is performed according to DBN B.2.6-198:2014, an additional option appears which allows to skip the slenderness check, since it is now not mandatory according to Change No.1.

When calculating for the action of accidental loads, by default characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017). Since this section was excluded by Change No. 2 to SP 296.1325800.2017, the user can (using the appropriate checkbox) choose which material properties (characteristic or design) to use.

4.7 Reference Modes

4.7.1 Steel

a



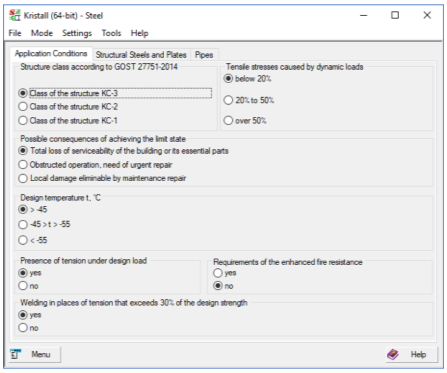
This mode is used to select a steel grade for the designed structure. The choice is made for four groups of structures according to

- Table 50* of SNiP II-23-81*;
- ShNK 2.03.05-13;
- Annex C of SP 53-102-2004;
- Annex C of SP 16.13330;
- Annex F of DBN B.2.6-163:2010;
- Annex D of DBN B.2.6-198:2014.

The procedure for classifying a structure into a certain group is described in [4] and in DBN B.2.6-163:2010.

The **Application Conditions** tab (Fig. 4.7.1-1, *a*, *b*) contains six groups of data.

b



c

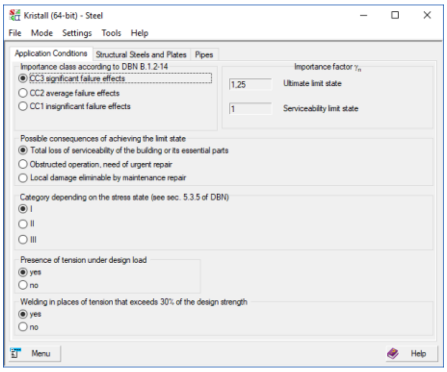


Figure 4.7.1-1. *The* **Application Conditions** *tab of the* **Steel** *dialog box*
a) — for SNiP II-23-81*
b) — for SP 53-102-2004 or SP 16.13330
c) — for DBN B.2.6-163:2010 or DBN B.2.6-198:2014

In the **Importance class according to GOST 27751** group [17] you should select one of the four cases defined in the codes, keeping in mind that the importance factor, γ_n, for unique projects is, as a rule, defined individually for each particular project by an institution which gives an approval of the design. To implement this procedure, use the **Application Settings** dialog box (see Section 4.2). The default value for the objects of the said type is $\gamma_n = 1{,}2$.

The **Possible consequences of achieving the limit state** group enables to select one of the three importance classes of the considered structural member. The paper [4] provides recommendations on how to classify some structures into these classes (Classes A, B, and C correspond to three grades of the considered group). This rating is accurate for DBN but can be used in other cases as well.

Necessary extracts from these recommendations are given below.

Structures of service platforms and decks:

- main and secondary beams, girders of frames — A;
- metal sheeting — B.

Columns of industrial buildings and open crane trestles:

- columns of service platforms and decks — A;
- main members of the cross-section — A;
- main vertical bracings between columns — A;
- bracings with the stress below $0.4R_y$ — C.

Roofing structures:

- trusses, girders — A;
- skylight panels, roof panels, purlins, longitudinal bracings — B;
- other bracings — C.

Framework structures:

- girders under brick walls and above gates — A;
- columns, end and wind trusses — B;
- other members — C.

Auxiliary structures:

- stringers — A;
- landings, imposts, window and skylight casements — C.

Conveyor galleries:

- span structures, column bracings — A;
- other bracings, roof beams, framework members — B.

Supports of power transmission lines and structures of outdoor switch gears (OSG):

- supports of power transmission lines, pylons for OSG switches — A;
- pylons for other OSG equipment — B.

Aerial equipment:

- trunks of masts and towers — A;
- diaphragms of towers, stairways, landings — B.

Vents and chimneys:

- chords and lattices of towers, shell of a separate pipe — A;
- flue-gas stacks, pipe shells with bracings — B;
- decks, mantles, stiffeners — C.

Cooling towers, water towers:

- chords of lattice towers, lattices — A;
- framework, decks, covering of cooling towers — C.

Bunkers, silos — A.

Other groups serve to characterize the stress state (**Presence of tension under design load** and **Tensile stresses caused by dynamic loads**), and to indicate the temperature mode of operation (**Climatic region according to GOST 16350-80**). The latter group requires that you indicate whether there is welding in the tension area (**Welding in places of tension that exceeds 30% of the design strength**).

Once you have filled all the data in the first tab, click on the **Structural Steels and Plates** or **Pipes** tab to open the respective tab of the dialog box.

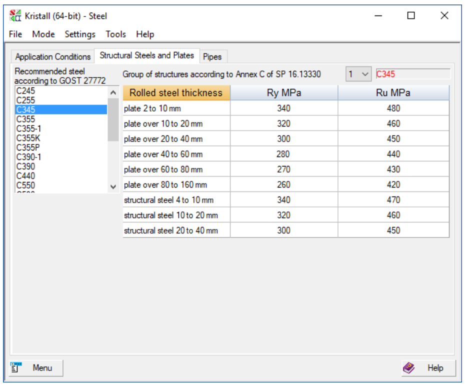


Figure 4.7.1-2. *The* **Structural Steels and Plates** *tab of the* **Steel** *dialog box*

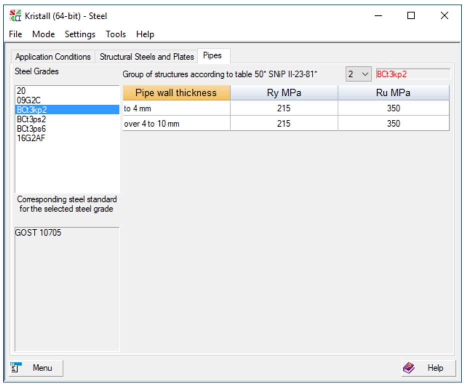


Рис. 4.7.1-3. *The* **Pipes** *tab of the* **Steel** *dialog box*

Tabs (Fig. 4.7.1-2, 4.7.1-3) contain a reference to a group of structures according to

- Table 50* of SNiP II-23-81*;
- Annex C of SP 53-102-2004;
- Annex C of SP 16.13330;
- Annex C of DBN B.2.6-163:2010;
- Annex A of DBN B.2.6-198:2014,

which conforms to operating conditions specified on the previous tab, a list of steel grades recommended for this group according to GOST 27772-88, and a list of steel grades according to other standards or codes which can be used instead of the recommended steel. This tab also provides reference data on the design strength based on the yield strength (R_y) and on the ultimate strength (R_u).

Steels are selected from one of the following groups: structural steels and plates (Fig. 4.7.1-2) and steels for hot rolled pipes (Fig. 4.7.1-3) selected according to the Table 51* of SNiP II-23-81* or according to the Table C6 of SP 53-102-2004, SP 16.13330, Table F.1 of DBN B.2.6-163:2010, Table D.1 of DBN B.2.6-198:2014.

A certain group of structures may require steel of a better grade than that recommended by SNiP or SP. Therefore the user can increase (by no means decrease!) the group of structures by selecting its number from the respective drop-down list. Obviously, the list of recommended steel grades will change as well.

The **Apply** button is used to transfer the properties of the selected steel grade to the active design mode (from which the **Steel** mode has been invoked). This grade will be used to assess the load-bearing capacity. If you have to use a non-standard steel given by R_y and R_u parameters, it can be done in the **Application settings** dialog box (Fig. 4.2-1).

It should be noted that the application does not list all the details which are obligatory to include in the order on steel, such as in notes to Tables 50* and 51, *b* of SNiP II-23-81* and notes to text and tables of the Annex C of SP 53-102-2004 or SP 16.13330 and Annex F of DBN B.2.6-163:2010 and D of DBN B.2.6-198:2014. Primary regulatory codes and specifications should be used to create an order.

4.7.2 Assortment of Rolled Profiles

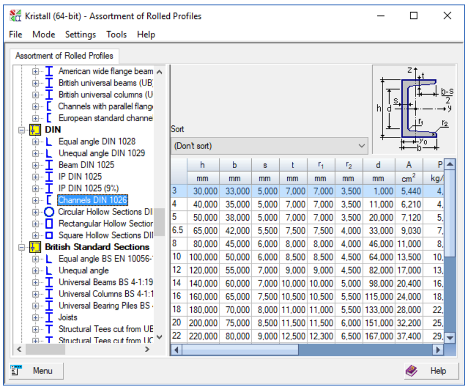


Figure 4.7.2-1. *The* **Assortment of Rolled Profiles** *dialog box*

This mode (Fig. 4.7.2-1) enables you to browse through steel profile assortments available in the database of **Kristall**.

The dialog box contains a list of assortments represented by a tree-like structure, and a table with data on the respective rolled profiles. A list of profile properties which can be used to sort the table is placed above the table.

To view the section with its dimensions, open the profile group list and choose the respective profile.

4.7.3 Bolts

This mode is used to select bolts for the structure being designed. The selection is available for four groups of structures according to Table 57* of SNiP II-23-81*, Annex D of SP 53-102-2004 or SP 16.13330, Annex C of DBN B.2.6-163:2010, Annex A of DBN B.2.6-198:2014.

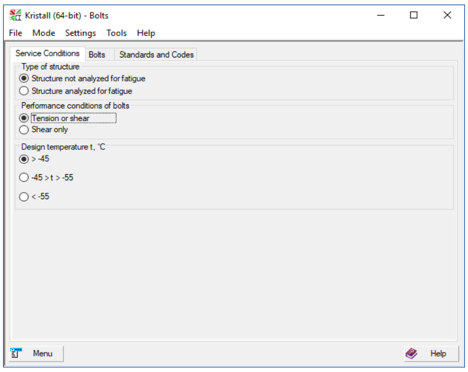


Figure 4.7.3-1. *The* **Service Conditions** *tab of the* **Bolts** *dialog box*

The **Service Conditions** tab (Fig. 4.7.3-1) contains three groups of data:

- **Type of structure**, where you can select one of two cases, defined by the codes, according to the requirements imposed on durability of the bolted joint.
- **Performance conditions of bolts**, where you can select a type of performance conditions for a bolt in a joint. The **Shear only** option is the recommended one for all joint types provided by the **Bolted Connections** mode.
- **Climatic region according to GOST 16350-80**, where you can specify one of the temperature modes of operation defined by the codes (it is not used if DBN is selected).

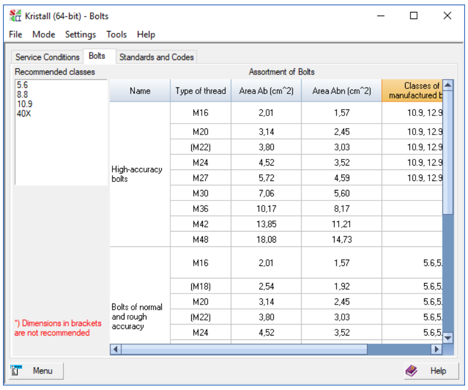


Figure 4.7.3-2. *The* **Bolts** *tab of the* **Bolts** *dialog box*

Once you have specified all the data in the first tab, open the next one by clicking on the **Bolts** tab. The **Bolts** tab (Fig. 4.7.3-2), contains a list of recommended bolt classes and their properties.

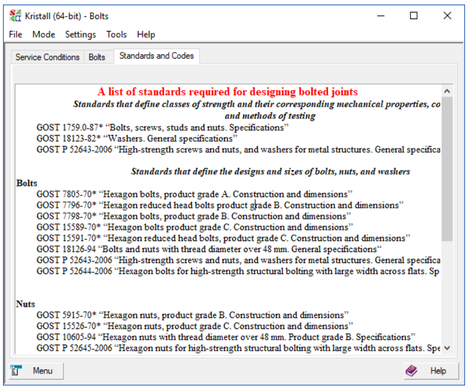


Figure 4.7.3-3. *The* **Standards and Codes** *tab of the* **Bolts** *dialog box*

The **Standards and Codes** tab (Fig. 4.7.3-3) provides the user with the reference information necessary for the design of a bolted joint.

4.7.4 Limit Slenderness

The dialog box of this mode (Fig. 4.7.4-1) contains the data given in Tables 19* and 20* of SNiP II-23-81*, Tables 30 and 31 of SP 53-102-2004 (Tables 32, 33 of SP 16.13330, Tables 1.9.9 and 1.9.10 of DBN B.2.6-163:2010, Table 13.9 and 13.10 of DBN B.2.6-198:2014).

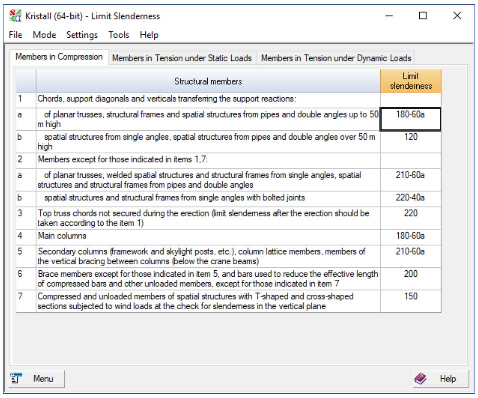


Figure 4.7.4-1. *The* **Limit Slenderness** *dialog box*

Selecting one of the tabs **Members in Compression**, **Members in Tension under Static Loads**, or **Members in Tension under Dynamic Loads** will open respectively Tables 19* and 20* of SNiP II-23-81*, or Tables 30 and 31 of SP 53-102-2004 (Tables 32, 33 of SP 16.13330 or Tables 1.9.9 and 1.9.10 of DBN B.2.6-163:2010, Table 13.9 and 13.10 of DBN B.2.6-198:2014).

If the **Limit Slenderness** mode has been invoked from a design mode, then after clicking the **Apply** button the selected value of the limit slenderness will be used in the analysis of structural members. Since the structure can be checked for various loading patterns, including ones where the longitudinal force sign is altered, the **Apply** button should be used twice — when the member is in tension and when it is in compression. The selected values can be modified in the **Application settings** dialog box (Fig. 4.2-1).

Limitations

Members in tension defined by Table 20* of SNiP II-23-81* and Table 31 of SP 53-102-2004 (Table 33 of SP 16.13330, Table 1.9.10 of DBN B.2.6-163:2010, Table 13.10 of DBN B.2.6-198:2014) which are subjected to the crane loads are not considered in the application.

The allowance on the increase of the ultimate slenderness by 10% given in Sec. 11.4.1 of SP 53-102-2004, Sec. 10.4.2 of SP 16.13330, Sec. 1.9.4.2 of DBN B.2.6-163:2010, Sec. 13.4.2 of DBN B.2.6-198:2014 is not implemented.

4.7.5 Service Factor

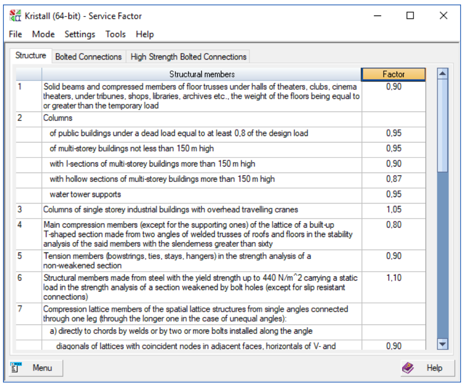


Figure 4.7.5-1. *The* **Structure** *tab of the* **Service Factor** *dialog box*

The dialog box of this mode (Fig. 4.7.5-1) contains data given in Table 6* of SNiP II-23-81* and Table 1 of SP 53-102-2004 (or SP 16.13330), Table 1.1.1 of DBN B.2.6-163:2010, Table 5.1 of DBN B.2.6-198:2014. In addition to the eleven items given in these tables, the dialog box contains additional cases implementing the instructions of the notes to these tables.

There are separate tabs with appropriate factors for bolted connections and high strength bolted connections. They can be opened by clicking on the respective tabs at the top of the dialog.

There is a separate tab for bolted connections (Fig. 4.7.5-2), which implements the requirements of Table 35* of SNiP II-23-81*, Table 38 of SP 53-102-2004, Table 41 of SP 16.13330, Table 1.12.4 of DBN B.2.6-163:2010, Table 16.4 of DBN B.2.6-198:2014.

a *b*

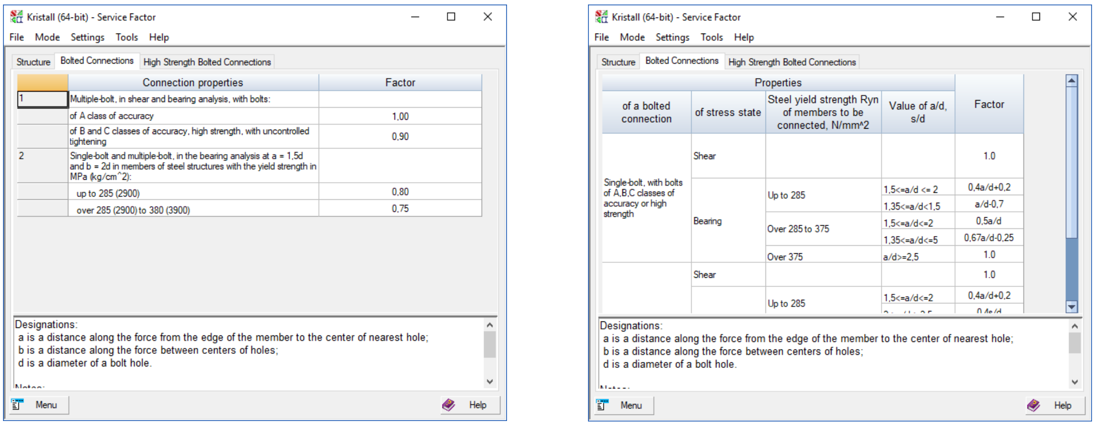


Figure 4.7.5-2. *The* **Bolted Connections** *tab of the* **Service Factor** *dialog box*
a — according to SNiP II-23-81*
b — according to SP 53-102-2004, SP 16.13330, DBN B.2.6-163:2010, or DBN B.2.6-198:2014

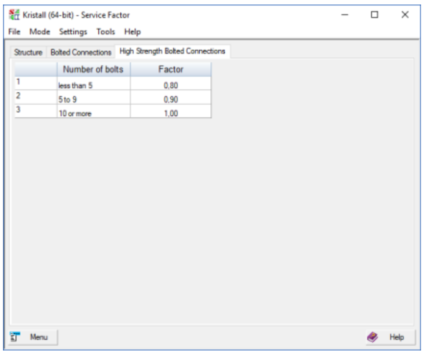


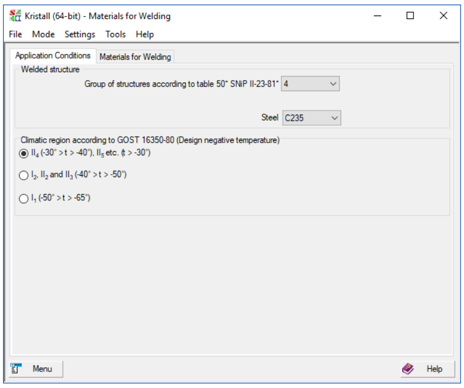
Figure 4.7.5-3. *The* **High Strength Bolted Connections** *tab of the* **Service Factor** *dialog box*

There is a separate tab for high strength bolted connections (Fig. 4.7.5-3), which implements the requirements of Sec. 11.13* of SNiP II-23-81*, Sec. 15.3.4 of SP 53-102-2004, Sec. 14.3.4 of SP 16.13330, Sec. 1.12.3.4 of DBN B.2.6-163:2010, Sec. 16.3.4 of DBN B.2.6-198:2014.

If the **Service Factor** mode has been invoked from a design mode, then after clicking the **Apply** button the selected value of the factor will be used in the analysis of structural members. The value can be modified in the **Application settings** dialog box (Fig. 4.2-1).

4.7.6 Materials for Welding

a



b

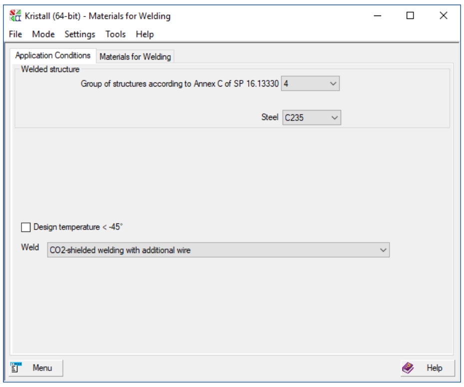


Figure 4.7.6-1. *The* **Application Conditions** *tab of the* **Materials for Welding** *dialog box*
a — according to SNiP II-23-81*
b — according to SP 53-102-2004 or SP 16.13330

This mode is used to select welding materials for the structure being designed. The selection procedure complies with Table 55* of SNiP II-23-81* and Sec. 6.4 and Annex D of SP 53-102-2004 (or Sec. 5.4 and Annex D of SP 16.13330, Annex G of DBN B.2.6-163:2010, Annex E of DBN B.2.6-198:2014).

The **Application Conditions** tab (Fig. 4.7.6-1) contains two groups of data.

The **Welded structure** group is used to specify the number of the group this structure belongs to according to Table 50 of SNiP II-23-81*, Annex C of SP 53-102-2004 and SP 16.13330, or Annex F of DBN B.2.6-163:2010 and Annex D of DBN B.2.6-198:2014 or according to the results of the **Steel** mode, and steel the structure is made of.

The second group is used to specify the climatic region according to GOST 16350-80 when SNiP is selected, and the design temperature and the welding technology when SP is selected.

Once you have specified all the data in the first tab, open the next one by clicking on the **Materials for Welding** tab.

The **Materials for Welding** tab (Fig. 4.7.6-2) contains a list of recommended materials.

If the **Materials for Welding** mode has been invoked from a design mode, then after clicking the **Apply** button the properties of the selected materials will be used in the analysis of structural members.

a *b*

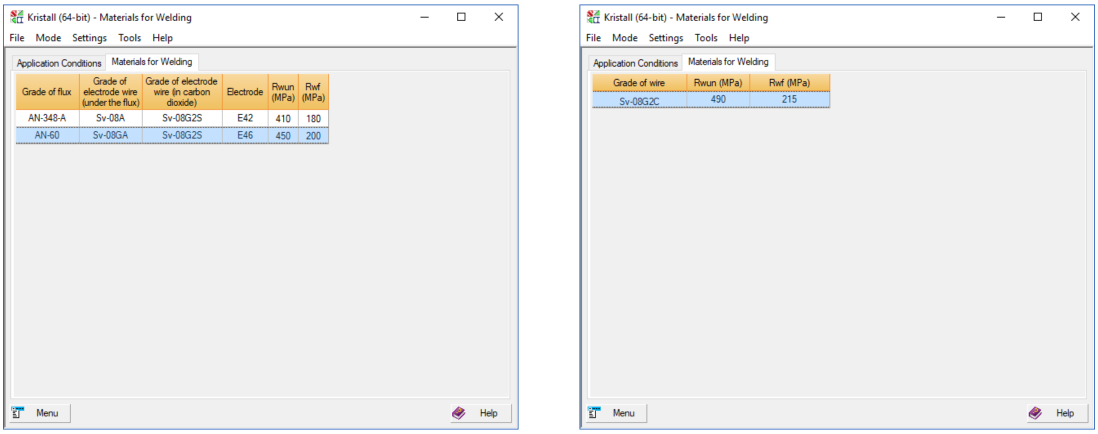
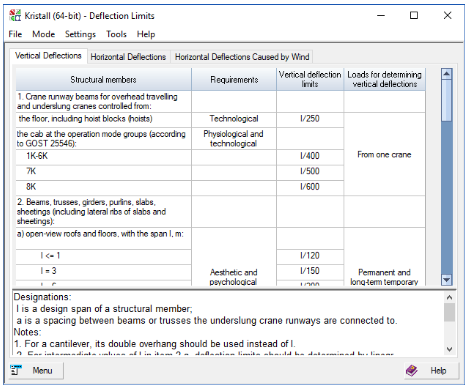


Figure 4.7.6-2. *The* **Materials for Welding** *tab of the* **Materials for Welding** *dialog box*
a — according to SNiP II-23-81*, *b* — according to SP 53-102-2004 or SP 16.13330

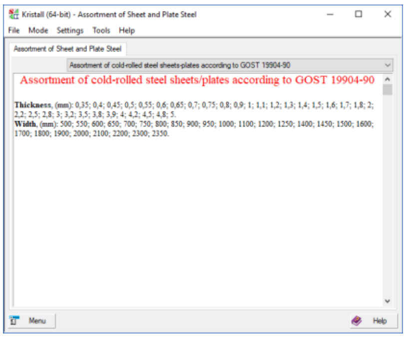
4.7.7 Deflection Limits



Tabs of this dialog box (Fig. 4.7.7-1) provide data from the tables of SNiP 2.01.07-85*, SP 20.13330.2011 or DSTU B B.1.2-3:2006.

Figure 4.7.7-1. *The* **Deflection Limits** *dialog box*

4.7.8 Assortment of Sheet and Plate Steel



This dialog box (Fig. 4.7.8-1) provides information on the assortment of sheets and plates manufactured in compliance with GOST 19903-74*, 19904-90, 82-70 and 103-76.

Figure 4.7.8-1. *The* **Assortment of Sheet and Plate Steel** *dialog box*

4.7.9 High-strength Bolts

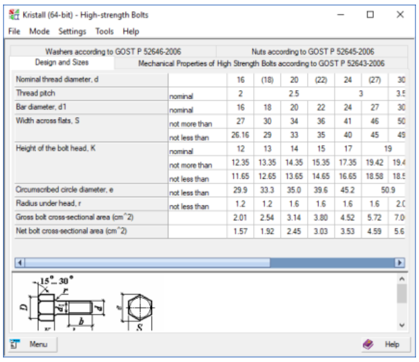


Figure 4.7.9-1. *The* **High-strength Bolts** *dialog box*

Information in this dialog box includes the data on the design and sizes of the high-strength bolts, their mechanical properties, and the description of washers and nuts used together with the high-strength bolts (Fig. 4.7.9-1).

4.8 Auxiliary Modes

4.8.1 Envelopes

This mode is used to solve a partial problem of determination of the basic unfavorable combinations of multiple loads acting on bending members.

It should be noted that the temporary loads implicitly include a zero load (it enables to describe the case of absence of all temporary loads). Therefore, when calculating maximum values (e.g., bending moments) we take the greatest of the positive moments and zero, and when calculating minimum values we take the least of the negative moments and zero.

Specify the beam span in the **Envelopes** dialog box (Fig. 4.8.1-1) and click the **Apply** button. Select one of the following boundary conditions using buttons on the top right: clamped at both ends; simply supported; hinged at one end and clamped at the other; or a cantilever beam.

The application is capable of analyzing several (up to ten) patterns of loading and each load case can in its turn consist of multiple loads.

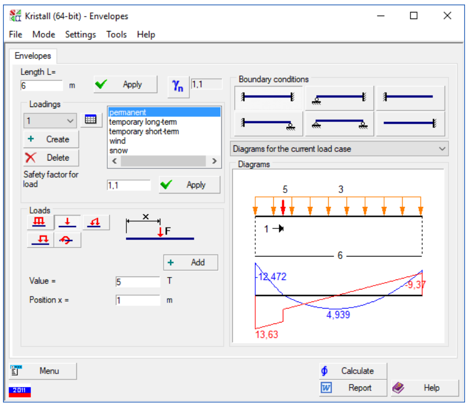


Figure 4.8.1-1. *The* **Envelopes** *dialog box*

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Loadings** group**;**
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors according to SNiP 2.01.07-85*, SP or DBN to be used with the loads of this load case in a combination of loads;
- modify, if necessary, the value of the safety factor for load;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;
- click the **Add** button.

A few load components can be specified for each load case. It is assumed that the design values of loads are entered.

Depending on the load type, its parameters may include:

- for distributed loads — the load intensity;
- for a distributed load on a part of the span — the intensity of the load, its position and width of application;

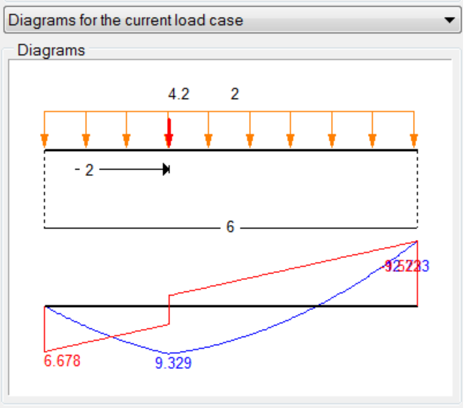


Figure 4.8.1-2. Diagrams of moments and shearing forces

- for a concentrated force — the value of the force and its position in the span;
- for a concentrated moment — the value of the moment and its position in the span.

The **Delete** button is used to delete a load case (not a separate load included in it).

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Loadings** list.

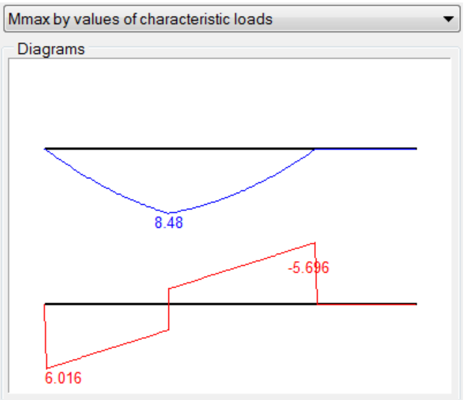


Figure 4.8.1-3. *Envelope diagrams*

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** fields with the superimposed diagrams of the bending moments and shear forces underneath (Fig. 4.8.1-2). Once you have entered all the loadings, you can view the values of the extreme moments and their corresponding shear forces, as well as the extreme shear forces and their corresponding bending moments. This can be done by selecting the respective item from the drop-down list above the **Diagrams** field (Fig. 4.8.1-3). The envelope diagrams of maximal and minimal forces are displayed separately.

If you place the mouse pointer in the diagram field, values of the moment and shear force in a particular cross-section corresponding to the position of the pointer will be displayed (Fig. 4.8.1-4).

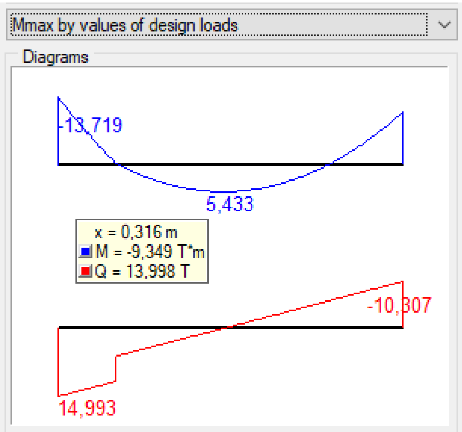


Figure 4.8.1-4. Indication on the diagram of the moment and shear force values in a particular cross-section

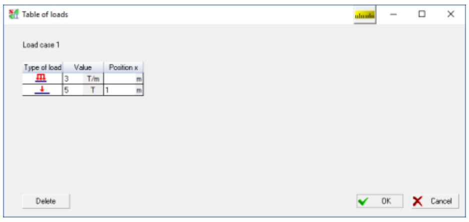


Figure 4.8.1-5. *The* **Table of loads** *dialog box*

To edit (and to delete, if necessary) particular loads from one load case, you can use a table of loads which is displayed in the respective dialog box (Fig. 4.8.1-5) after you click the button. In this table, you can modify the value of a load or its application point, and delete one or more loads. In the latter case select the icon of the load you want to delete and click the **Delete** button.

4.8.2 Influence Lines

The mode is used to plot influence lines for multi-span continuous beams of constant section. Only the influence lines of bending moments and shear forces are considered.

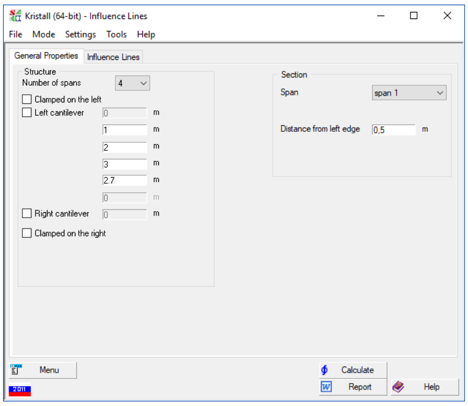


Figure 4.8.2-1. *The* **General Properties** *tab of the* **Influence Lines** *dialog box*

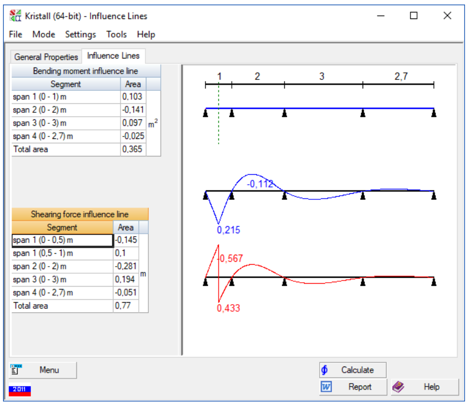


Figure 4.8.2-2. *The* **Influence Lines** *tab of the* **Influence Lines** *dialog box*

The design of a multi-span beam is specified in the **Design solution** group of the **General Properties** tab (Fig. 4.8.2-1). It is defined by the number of spans, their length and the presence or absence of cantilevers. If you specify stiff clamping (on the right and/or on the left), the respective cantilever can no longer be defined. The **Section** group is used to select a span and specify the distance from its left edge to a section for which you want to plot the influence lines. The influence lines can be obtained by clicking the **Calculate** button or opening the **Influence Lines** tab.

The **Influence Lines** tab (see Fig. 4.8.2-2) provides the structural scheme of the beam and two influence lines (of bending moments and shear forces). If you place the mouse pointer over any point along the beam, the values of the moment and the shear force in the pointed section will be displayed.

Tables on the left from the influence lines contain information on the areas of the influence lines in segments between zero points. These values can be used to find the moment and/or the shear force under an evenly distributed temporary load.

4.8.3 Geometric Properties of Sections

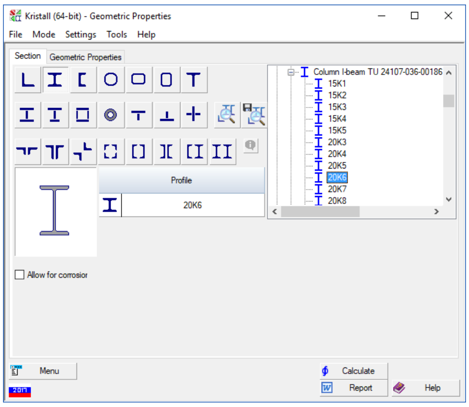


Figure 4.8.3-1. *The* **Section** *tab of the* **Geometric Properties** *dialog box*

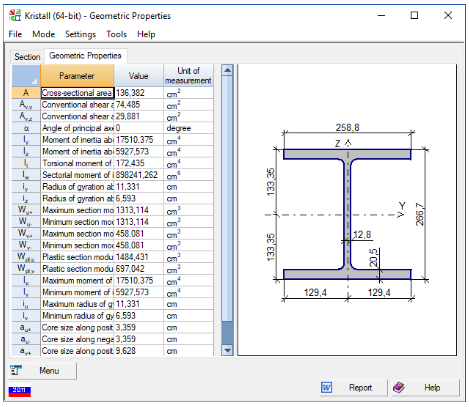


Figure 4.8.3-2. *The* **Geometric Properties** *tab of the* **Geometric Properties** *dialog box*

This mode enables to calculate the geometric properties of cross-sections according to the rules described above (see Section 4.3). The initial data are entered in the **Section** tab (Fig. 4.8.3-1). Results are displayed in the form shown in Fig. 4.8.3-2, in the **Geometric Properties** tab.

4.8.4 Effective Lengths

This mode enables to solve a problem of determining the effective length of members according to the recommendations of Section 6 of SNiP II-23-81* (Section 11 of SP 53-102-2004, Section 10 of SP 16.13330, Section 1.9 of DBN B.2.6-163:2010, Section 13.9 of DBN B.2.6-198:2014), ShNK 2.03.05-13 and EN 1993-1-1. Eurocode 3. This problem is actually a problem of structural mechanics, therefore using the recommendations of Eurocode 3 can not be considered as unacceptable. The result is the value of the effective length factor or the slenderness of the member.

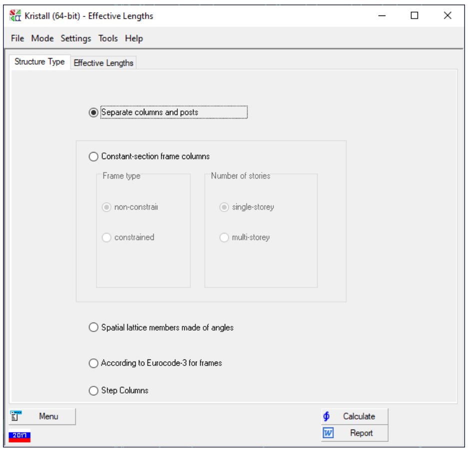


Figure 4.8.4-1. *The* **Structure Type** *tab of the* **Effective Lengths** *dialog box*

The main window of this mode (Fig. 4.8.4-1) contains two tabs. The first of them (**Structure Type**) is used to select a considered structure from the following options:

- separate columns or posts;
- constant-section frame columns (according to SNiP, SP, DBN and EN 1993-1-1. Eurocode 3);
- spatial lattice members made of angles;
- stepped columns.

Solutions for truss members defined by Sections 6.1–6.4 of SNiP II-23-81* (Sec. 11.1 of SP 53-102-2004, Sec. 10.1 of SP 16.13330, Sec. 1.9.1-1.9.4 of DBN B.2.6-163:2010, Sec. 13.1-13.4 of DBN B.2.6-198:2014) are implemented in the **Trusses** mode (see further below).

Once you have selected your structure type, the **Structure Properties** tab is activated. Its configuration depends on the selected structure type.

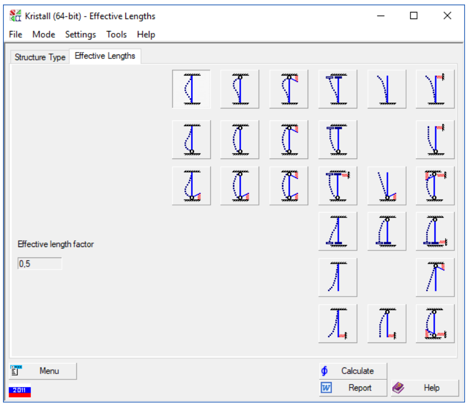


Figure 4.8.4-2. *The* **Effective Lengths** *tab for* **separate columns and posts**

If you select the **Separate columns and posts** structure type, this tab will contain 25 possible conditions of end support (Fig. 4.8.4-2).

For the restraint with idealized boundary conditions (the respective button is pressed), the **Effective length factor** field will display a value that corresponds to the instructions given in design codes. Only the cases when the force is applied to the end are considered.

If the elastic supports with respect to displacements or rotations are selected, you have to specify the numerical data on the stiffness of these supports in the respective text fields. Stiffnesses of elastic linear supports are specified in T/m, and those of elastic rotational supports – in T*m/rad. Once you click the **Calculate** button, the result calculated by the formulas from the Guide to Design of Steel Structures (to SNiP II-23-81*) will be displayed in the respective field.

In some design models where the restraints are not sufficiently stiff, the column may lose its stability as a rigid body. In such cases the result will be a value of the least critical force causing a deformation of the element.

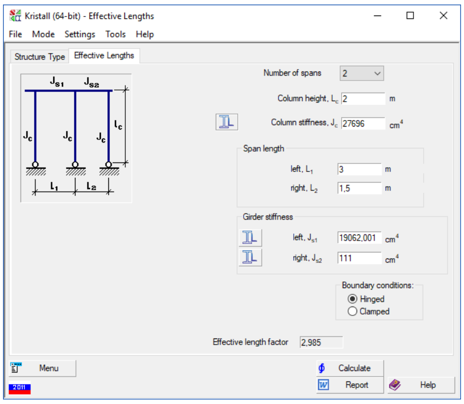


Figure 4.8.4-3. *The* **Effective Lengths** *tab for* **constant-section frame columns** (single-storey)

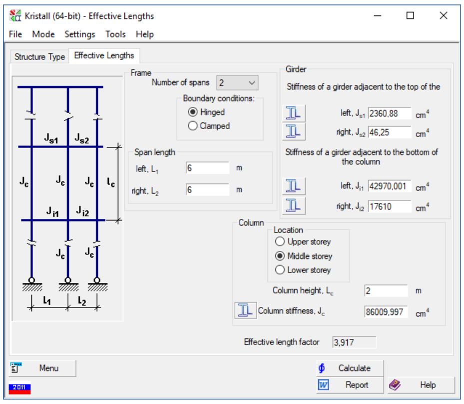


Figure 4.8.4-4. *The* **Effective Lengths** *tab for* **constant-section frame columns** (multi-storey)

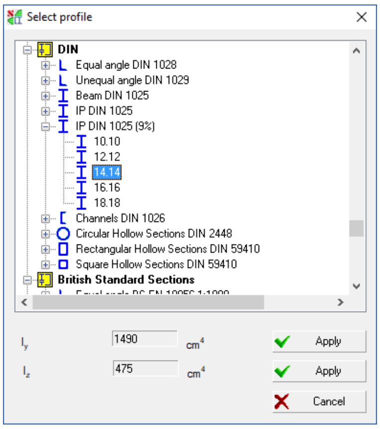


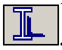
Figure 4.8.4-5. *The* **Select profile** *dialog box*

Once you have selected the **Constant-section frame columns** type and indicated the frame type (**non-constrained/constrained** and **single-storey/multi-storey**), the **Effective Length** tab will contain the respective design model (see examples in Fig. 4.8.4-3 and Fig. 4.8.4-4) together with text fields for entering the appropriate initial data. The problem is solved only in the frame plane, and only a stiff connection between girders and columns is considered.

The set of considered problems corresponds to Sec. 6.10* and Table 17, *a* of SNiP II-23-81* (Sec. 11.3.3 and Table 29 of SP 53-103-2004, Sec. 10.3.4 and Table 31 of SP 16.13330, Sec. 1.9.3.4, Table 1.9.8 of DBN B.2.6-163:2010 and Table 13.8 of DBN B.2.6-198:2014). It is assumed that all columns of a frame are of the same stiffness and are loaded by equal longitudinal forces. Therefore Formula (71)* of SNiP II-23-81* (Formula (127) of SP 53-102-2004, Formula (146) of SP 16.13330, Formula (1.9.8) of DBN B.2.6-163:2010, Formula (13.8) of DBN B.2.6-198:2014) is not used. It should be also noted that the concept of a “middle storey” must not be used when considering two-storey frames, and in constrained frames the result does not depend on the number of spans. Therefore the respective text field will not be accessible.

Since the effective lengths of the edge and middle columns in multi-span non-constrained frames differ from each other (it is not taken into account directly), we recommend the following procedure: the solution for the middle columns is determined as for a multi-span frame, and that for the edge columns is determined as for a single-span frame. The designations of lengths and stiffnesses on the left from the considered column are used in the single-span frames.

Calculations of the effective lengths for particular segments of stepped columns, out-of-plane columns, and stanchions of flat pillars under conveyor galleries are not performed.

If members of the frame are rolled profiles, their stiffness can be specified by selecting the respective profile from the assortment; to do it, click the button . This will open the **Select profile** dialog box (Fig. 4.8.4-5), where you can select a desired profile and click the **Apply** button for stiffness I_y or I_z.

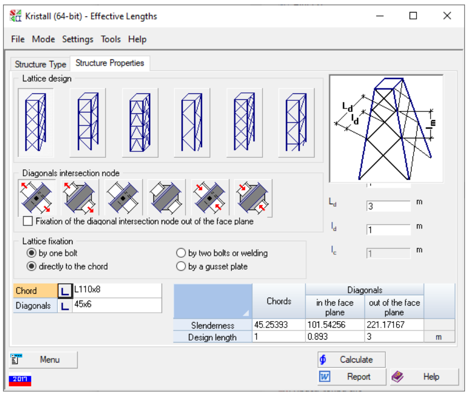


Figure 4.8.4-6. *The* **Structure Properties** *tab for* **spatial lattice members made of angles**

If you have selected **Spatial lattice members made of angles** as the **Structure Type**, the **Structure Properties** tab (Fig. 4.8.4-6) will contain the designs defined by Sec. 6.5 of SNiP II-23-81* (Sec. 11.2 of SP 53-102-2004, Sec. 10.2 of SP 16.13330, Sec. 1.9.2 of DBN B.2.6-163:2010, Sec. 13.2 of DBN B.2.6-198:2014). The lattice design can be selected by clicking the respective button. Sections for the members are selected from the assortments using the following buttons: **Chord**, **Diagonals**, **Horizontal**. Only equal angles can be used for the chords and horizontals, while the crossing diagonals can be made of unequal angles assuming they are connected to the chord through their narrow legs.

For some configurations, you can select a type of joint at the intersection of the diagonals using the respective group of buttons. At the same time you select the type of forces in the reinforcing member according to Table 14* of SNiP II-23-81* (Table 23 of SP 53-102-2004, Table 25 of SP 16.13330, Table 1.9.2 of DBN B.2.6-163:2010, Table 13.2 of DBN B.2.6-198:2014) (a compressed member, highlighted by a darker color in the respective icon, is under consideration).

You have to choose one of the suggested designs by selecting an appropriate radio button in the **Lattice fixation** group. It is assumed that this design is applied to both ends of the lattice member. The application does not consider structures where, for example, one end of the member is fixed directly to the chord and the other one is connected by a gusset plate.

When determining the effective lengths and slenderness of space lattice structures from single angles, the program implements various cases of using the stiffness properties of the section (relative to the principal axes of inertia of the section, as well as relative to the structural axes parallel to the legs of the angle) in full accordance with Table 13* of SNiP II-23- 81* (Table 22 of SP 53-102-2004, Table 24 of SP 16.13330, Table 1.9.1 of DBN V.2.6-163:2010, Table 13.1 of DBN V.2.6-198:2014, Table 7 ShNK 2.03.05-13).

Clicking the **Calculate** button in the **Structure Properties** tab will display the slenderness values of the compressed chord elements and diagonal elements of space lattice structures, as well as the corresponding effective lengths. Moreover, for the elements of the cross lattice, the program calculates the slenderness and effective lengths both in the plane of the face of the space lattice structure and out of its plane according to Table 19 * of SNiP II-23-81 * (Table 30 of SP 53-102-2004, Table 32 of SP 16.13330, Table 1.9.9 of DBN V.2.6-163:2010, Table 13.9 of DBN V.2.6-198:2014, Table. K.1 Annex K ShNK 2.03.05-13).

.

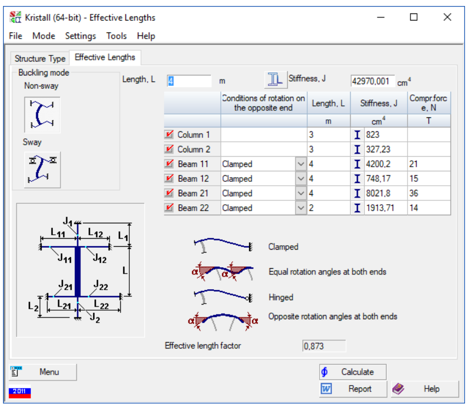


Figure 4.8.4-7. *The* **Effective Lengths** *tab for the* **According to Eurocode-3 for**

If you have selected the **According to Eurocode 3 for frames** mode, the **Effective Lengths** tab will contain a model of the considered post with its adjacent girders and columns (Fig. 4.8.4-7).

Data on the length of the post, L, and its stiffness, J, are specified in the respective text fields. If the post has a rolled section, its stiffness can be entered automatically after selecting the section from the assortment (using the button).

The table of properties of the surrounding members is used to enter the data on the length of the members, conditions of their abutment (rotation) on the end opposite to the post, stiffness and the compressive force value. If any of the members is absent (is not checked in the first column of the table), the respective text fields will be inaccessible.

All the calculations comply with the recommendations from Annex E of EN 1993-1-1.

frames *mode*

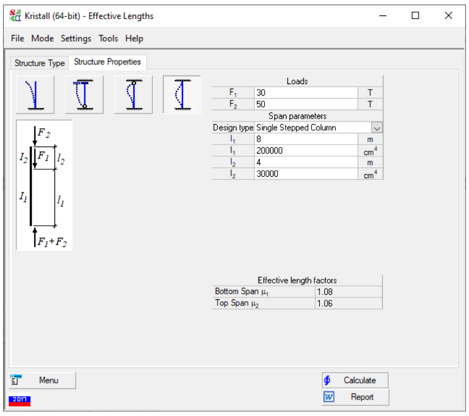


Figure 4.8.4-8. *The* **Effective Lengths** *tab for* **Stepped columns** *mode*

Selecting the **Stepped columns** structure type invokes the **Structure Properties** tab containing 4 possible conditions of end support (Fig. 4.8.4-8):

1) clamped at the bottom and free at the top;
2) clamped at the bottom and fixed against rotation at the top;
3) clamped at the bottom and hinged at the top;
4) clamped at both ends.

This tab also contains the **Loads** and **Segment Properties** tables (see Fig. 4.7.4-8), where you can specify properties of segments of a stepped column (cross-sectional moments of inertia and lengths) and the values of concentrated forces applied at the top of segments of the stepped column.

Clicking the **Calculate** button will display values of the effective length factors of segments of stepped columns determined in accordance with the selected design codes.

4.9 Checks

4.9.1 Resistance of Sections

This mode enables to determine the load-bearing capacity of any cross-section available in the application. In the general case, the analyzes are performed for a longitudinal force, bending moments and shear forces acting in the principal planes of inertia (except for a section made of a single angle for which the design codes do not provide any recommendations on the bending analysis). The whole set of checks for strength, stability and slenderness is implemented in compliance with the selected design code, with the following exceptions:

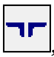
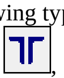
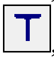
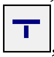
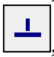
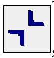
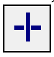
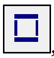
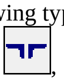
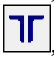
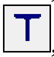
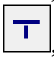
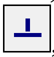
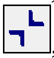
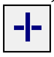
- bars in tension are not checked for strength by the design resistance, as members which can be used even after the yield point is reached;
- the calculations of the effective slenderness values for lattice bars are performed according to more accurate formulas from Table 13 of the Guide to SNiP II-23-81*;
- when calculating the φ_b coefficient, it is assumed that the load is uniformly distributed and applied to the compression chord which is not restrained in its span against buckling. However, a special dialog box can be invoked by clicking the button , where you can select one of the rules for calculating this factor provided by the codes (the user is responsible for the consequences of this selection), or use the **Default** button to return to the standard behavior of the program (described above).

The set of checks depends on the type of the member cross-section and the set of loads it is subjected to.

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength under axial tension/compression	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.1
Excessive deformations of the tension fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3	Sec. 7.28

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength under action of bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength under action of bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength under action of lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength under action of lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1. 1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1. 1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1. 1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.14
Stability under compression in XoY (XoU) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 9.2.1	Sec. 9.4
Stability under compression in XoY (XoU) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 9.4, 9.20

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability under compression in XoZ (XoV) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability under compression in XoZ (XoV) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Stability in compression of angle	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability in the moment M_y plane under eccentric compression	Sec. 5.27*	Sec. 5.37	Sec. 10.2.9, 10.2.10	Sec. 9.2.9, 9.2.10	Sec. 1.6.2.9, 1.6.2.10	Sec. 10.2.9, 10.2.10	Sec. 7.27
Stability in the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.4.6	Sec. 9.2.2, 9.2.10, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.4.5	Sec. 7.27, 9.20
Stability in the moment M_z plane under eccentric compression	Sec. 5.27*	Sec. 5.27	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2	Sec. 9.2.9, 9.2.10, 9.3.1, 9.3.2	Sec. 1.6.2.9, 1.6.2.10, 1.6.3.1, 1.6.3.2	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2	Sec. 7.27
Stability in the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.1, 10.3.2, 10.4.5	Sec. 7.27, 9.20
Stability under compression and bending in two planes	Sec. 5.34	Sec. 5.35	Sec. 10.2.9	Sec. 9.2.9	Sec. 1.6.2.9	Sec. 10.2.9	Sec. 7.34
Stability under compression and bending in two planes (post-buckling behavior)	Sec. 5.34, 7.20*	Sec. 5.35, 7.30	Sec. 10.2.9, 10.4.6	Sec. 9.2.9, 9.2.10, 9.4.6	Sec. 1.6.2.9, 1.6.2.10, 1.6.4.5	Sec. 10.2.9, 10.2.10, 10.4.5	Sec. 7.34, 9.20
Stability out of the moment M_y plane under eccentric compression	Sec. 5.30-5.32	Sec. 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 9.2.4, 9.2.5, 9.2.8	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 7.30-7.32

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability out of the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.30-5.32, 7.20*	Sec. 5.30-5.32, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.5	Sec. 7.30-7.32, 9.20
Stability out of the moment M_z plane under eccentric compression (sections of the following types , , , , , ,  are not checked)	Sec. 5.27*, 5.30-5.32	Sec. 5.27, 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.2	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 7.27, 7.30-7.32
Lateral-torsional buckling taking into account plastic deformation			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6	Sec. 7.27, 7.30-7.32, 9.20
Stability out of the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.32, 7.20*	Sec. 5.27, 5.30-5.32, 7.30	Sec. 10.2.4,10.2.5,10.2.8, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2, 10.4.5	Sec. 7.15
Stability of in-plane bending (sections of the following types , , , , , , ,  are not checked)	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	–

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9,7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.23, 7.26, 7.27, 7.28,	Sec. 8.3.2, Table 8; Sec. 8.3.10 ; Sec. 9.5.1 –9.5.9; Sec. 10.4.2, Table 20 ; Sec. 10.4.3; Sec. 10.4.9	Sec. 7.3.2, Table 9; Sec. 7.3.11 ; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22 ; Sec. 9.4.3; Sec. 9.4.9	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ; Sec. 1.6.4.5	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ; Sec. 10.4.5	Sec. 9.1 – 9.7, 9.10,9.11; Sec. 9.15, 9.16, 9.17, 9.18
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.27*	Sec. 7.32, 7.33, 7.34, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10 ; Sec. 9.5.14; Sec. 10.4.7, Table 21 ; Sec. 10.4.9	Sec. 7.3.8, Table 10; Sec. 7.3.11 ; Sec. 8.5.18; Sec. 9.4.7, Table 23 ; Sec. 9.4.9	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14; Sec. 1.6.4.8, Table 1.6.4 ; Sec. 1.6.4.7	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ; Sec. 10.4.7	Sec. 9.22, 9.23, 9.24, 9.27
Pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2	Sec. 10.6
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9	Sec. 10.5-10.13
Beam web height to thickness ratio	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3	Sec. 9.5
General stability of a build-up member under axial compression in XoY plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5	Sec. 7.3-7.6
General stability of a build-up member under axial compression in XoZ plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5	Sec. 7.3-7.6
Stability out of the moment M_z plane	Sec. 5.30-5.32	Sec. 5.27, 5.30-5.32	Sec. 10.2.4,10.2.5,10.2.8	Sec. 9.2.4,9.2.5,9.2.8	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 7.27*, 7.30-7.32

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of a batten to bending	Sec. 5.8, 5.9, 5.36	Sec. 5.8, 5.9, 5.38	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.2.7, 7.2.8, 9.3.7	Sec. 1.4.2.7, 1.4.2.8, 1.6.3.7	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.8, 7.9, 7.36
Strength of chord under bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength of chord under bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength of chord under lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18*	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3	Sec. 7.12, 7.18
Strength of chord under lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3	Sec. 7.12, 7.18
Strength of chord under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.24, 7.25, 7.33
Strength of chord under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.24, 7.25, 7.33
Strength of chord under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.24, 7.25, 7.33
Stability of chord under compression in XoY plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4	Sec. 7.3, 7.6
Stability of chord under compression in XoY plane (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5	Sec. 7.3, 7.6, 9.20

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of chord under compression in XoZ plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4	Sec. 7.3, 7.6
Stability of chord under compression in XoZ plane (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5	Sec. 7.3, 7.6, 9.20
Stability of chord in the moment M_y plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27*, 5.33, 5.35	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5	Sec. 7.27, 7.33, 7.35
Stability of chord in the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.2, 9.2.10, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.3.3-10.3.5, 10.4.5	Sec. 7.27, 7.33, 7.35, 9.20
Stability of chord in the moment M_z plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27, 5.33, 5.37	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5	Sec. 7.27, 7.33, 7.35
Stability of chord in the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.3, 10.3.4, 10.3.6, 10.3.2, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1-9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.2-1.6.3.5, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.2-10.3.5, 10.4.5	Sec. 7.27, 7.33, 7.35, 9.20
Bending of chord in two principal planes	Sec. 5.24,5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Stability of chord out of the moment M_y plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5	Sec. 7.27, 7.30-7.33, 7.35

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of chord out of the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5	Sec. 7.27, 7.30-7.33, 7.35, 9.20
Stability of chord out of the moment M_z plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5	Sec. 7.27, 7.30-7.33, 7.35
Stability of chord out of the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5	Sec. 7.27, 7.30-7.33, 7.35, 9.20
Strength of chord under tension	Sec. 5.1, 5.6	Sec. 5.1, 5.6	Sec. 8.1.1, 8.2.1	Sec. 7.1.1, 7.2.1	Sec. 1.4.1.3, 1.4.2.1	Sec. 8.1.3, 8.2.1	Sec. 7.1, 7.6
Stability of chord under compression	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3-1.4.2.4	Sec. 8.1.3, 8.2.3-8.2.4	Sec. 7.3, 7.6
Stability of chord under compression (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3-1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3-8.2.4, 8.3.5	Sec. 7.3, 7.6, 9.20
Excessive deformations of the tension chord fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3	Sec. 7.28
Stability of in-plane bending of the chord	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	Sec. 7.15
Chord web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*, 7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9,7.10; 7.23, 7.26, 7.27, 7.28	Sec. 8.3.2, Table 8; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20	Sec. 7.3.2, Table 9; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3	Sec. 9.1 – 9.7, 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Chord flange slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.26*	Sec. 7.32, 7.33, 7.34, 7.37	Sec. 8.3.7, Table 9; Sec. 9.5.14; Sec. 10.4.7, Table 21	Sec. 7.3.8, Table 10; Sec. 8.5.18; Sec. 9.4.7, Table 23	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14; Sec. 1.6.4.8, Table 1.6.4	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4	Sec. 9.22, 9.23, 9.24, 9.27
Strength of lattice posts	Sec. 5.10	Sec. 5.8, 5.10	Sec. 8.2.9	Sec. 7.2.9	Sec. 1.4.2.9	Sec. 8.2.9	Sec. 7.8, 7.10
Strength of lattice struts	Sec. 5.10	Sec. 5.8, 5.10	Sec. 8.2.9	Sec. 7.2.9	Sec. 1.4.2.9	Sec. 8.2.9	Sec. 7.8, 7.10
Stability of lattice posts under compression	Sec. 5.10, 5.3	Sec. 5.8, 5.10, 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3, 7.8, 7.10
Stability of lattice struts under compression	Sec. 5.10, 5.3	Sec. 5.8, 5.10, 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3, 7.8, 7.10
Slenderness in XoY plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1	Sec. 8.18
Slenderness in XoZ plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1	Sec. 8.18

Flexibility checks use the values specified in the **Limit Slenderness** mode.
Only the cross-section of the element is checked.

The following checks are not performed:

- of the weakened sections with holes for bolts;
- of the analysis of the battens and lattices, except for the **Columns** mode;
- of the local stress in the beam web from the concentrated force, except for the **Beams** and **Local Stability** modes;
- of local web buckling for I-sections and box cross-sections taking into account longitudinal stiffeners;
- of the strength of continuous and clamped beams allowing for the redistribution of forces in the plastic stage.

Peculiarities of the implementation

1. SNiP does not consider the problem of stability for a bar under tension and bending, but it would be unwise not to check it for stability at all, because even a relatively small tension can cause the buckling of the bar (in an elastic bar this would occur when some fibers were in compression, while in an elastoplastic bar this boundary would be harder to locate). Since design codes do not define a boundary for such a “relatively small tension”, we assumed it equal to zero, hence a simply bent bar is considered.

2. Since design codes do not provide a general recommendation for checking the stability of in-plane bending for a structure with arbitrary restraints and arbitrary positioning of the loads, a check based on an assumption that there are no intermediate bracings of the member is implemented, i.e. the effective length of the member in the respective formula is taken as its geometric length.

3. When determining the relative eccentricity of eccentrically loaded bars the design codes recommend taking the design moment as the moment in the section, which is located

in a particular area of the bar. This area is determined depending on the boundary conditions of the bar, on which the program has no information. Therefore, the value of the moment maximal along the length of the element is used.

4. Technically the codes do not require to check the axial compression stability of structures under eccentric compression (for example, under the action of N,M_y,M_z). However, without checking that the axial compression stability is ensured, other checks may become meaningless. For example, the value of the limit slenderness of the 180-60α type can become negative (the value of α coincides with the axial compression stability factor). Therefore, the axial compression stability factors are always calculated.

5. The recommendations of the codes for determining the section shape factor η for a welded unequal I-beam are provided only for the case when the ratio of the area of the smaller flange to the area of the larger flange is 0.5. If this ratio exceeds 0.5, the software performs a linear interpolation between the values of η calculated for an equal I-beam and the values of η calculated for an unequal I-beam with the ratio of the area of the smaller flange to the area of the larger flange equal to 0,5. This approach is recommended in Modification No.1 to DBN B.2.6-198:2014.

There is no separate strength analysis of members bending in two principal planes. This check is included in the strength check under the combined action of the longitudinal force and bending moments as a particular case at $N = 0$.

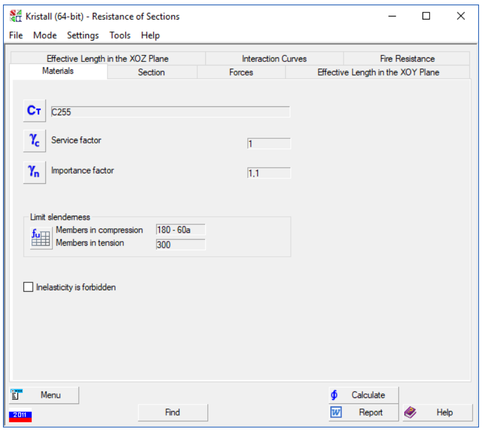
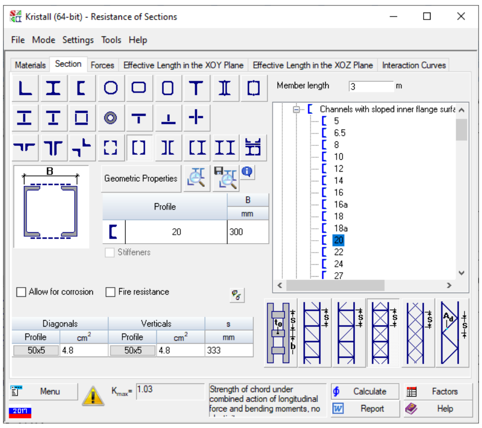


Figure 4.9.1-1. *The* **Materials** *tab of the* **Resistance of Sections** *dialog box*



The dialog box of this mode contains six tabs: **Materials** (Fig. 4.9.1-1)**,** **Section** (Fig. 4.9.1-2), **Forces** (Fig. 4.9.1-4), **Effective Length in the XoY Plane** (Fig. 4.9.1-5), **Effective Length in the XoZ Plane**, **Interaction Curves** (Fig. 4.9.1-7). The first five tabs are used to enter the initial data, and the sixth one to analyze the results of the calculation.

The **Materials** tab (Fig. 4.9.1-1) contains buttons for accessing the reference modes **Steel** — , **Service Factor** — , and **Limit Slenderness** — . Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. **Importance factors** — — are specified in the **Material** tab of the **Application settings** dialog box. This tab is also used in cases when the properties of steel or values of the factors have to be assigned values different from those defined by the design codes.

For members carrying the longitudinal force and the bending moment, design standards for steel structures suggest two possible strength checks:

- taking into account the possible elastoplastic behavior;
- in the absence of the inelastic behavior.

The possibility of the elastoplastic behavior is limited by a number of conditions, such as the absence of the direct action of dynamic loads.

In the case when there is a direct action of dynamic loads, as well as in cases when the user for some other reasons does not want to go beyond the elastic behavior, he can use the **Inelasticity is forbidden** checkbox provided on this tab.

The **Web instability is forbidden** checkbox is used to perform the check of the section taking into account its postbuckling

Figure 4.9.1-2. *The* **Section** *tab of the* **Resistance of Sections** *dialog box*

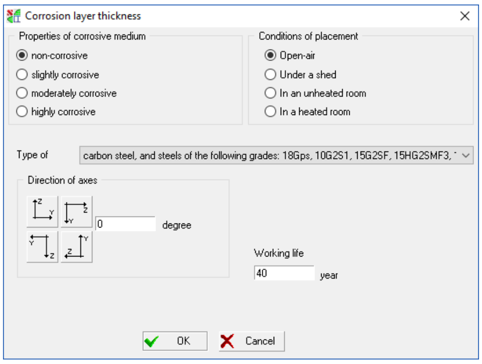


Figure 4.9.1-3. *The* **Corrosion layer thickness** *dialog box*

behavior (after the local buckling of the web). The checked checkbox enables to reject the postbuckling behavior of the section if the check indicates local buckling of the web.

The **Section** tab (Fig. 4.9.1-2) contains eighteen buttons clicking which enables you to set the desired cross-section type. The selected section can be saved in the **Custom Sections** catalogue the access to which is provided by the button .

The *length between restraints out of the bending plane* has to be specified (this value will be used in the analysis of stability of in-plane bending).

If the transverse stiffeners can be installed for the given cross-sections, you can use the **Stiffeners** checkbox, thus indicating that the stiffeners are installed, and specify their spacing. If this spacing is greater than the length of the element, the local stability analysis of the web is performed as for a web without stiffeners. If the web has such a slenderness that the element can be classified as an element with a flexible web, the application outputs the **Ratio between height and width of the web** factor with a value greater than 1,0. The calculation of elements with a flexible web is not implemented in the program due to the extremely limited scope of the method for calculating such structures (only for continuous beams bearing the static load).

This mode enables to perform the checks taking into account the corrosion. To do it, check the respective checkbox, specify the corrosion layer thickness or use the button  to calculate the corrosion layer thickness. In the **Corrosion layer thickness** dialog box (Fig. 4.9.1-3) you should specify the data on the properties of corrosive medium, conditions of placement, its working life, and the direction of the axes of inertia. As a result, a prediction of corrosion will be generated in accordance with recommendations from [2], [3], [9]. The analysis is based on the assumption that the thickness of the corrosion layer is the same along the whole perimeter of the section.

The button  can be used to access an archive of custom sections created by **Section Builder**, **Consul**, and **Tonus** (see [1]).

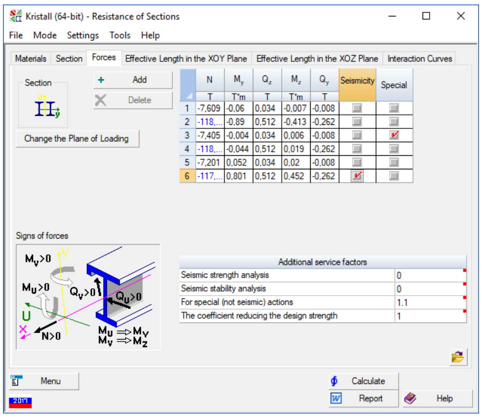


Figure 4.9.1-4. *The* **Forces** *tab of the* **Resistance of Sections** *dialog box*

It should be noted that any section generated by **Section Builder** is treated as a custom section (one different from a standard section). This rule also applies to the cases when the created section has a “standard” shape (for example, it may be just a rolled or welded I-beam, channel etc.). The application provides a lot of other capabilities for creating standard shapes.

Since SNiP, SP and DBN do not provide any recommendations for the determination of some parameters for the analysis of custom sections (section shape factor η, coefficients α and β according to Table 10 of SNiP, etc.), the analysis uses the most unfavorable values of these parameters, and the check for the stability of in-plane bending is not performed at all due to the assumption that the possibility for this mode of buckling to occur is excluded by the appropriate restraints. Moreover, different results will be obtained by the analysis of the section behavior under shear forces. The matter is that calculation of the tangential stresses according to the design codes is based on the assumption that the shear force is resisted only by those parts of the section which are “oriented” along the force

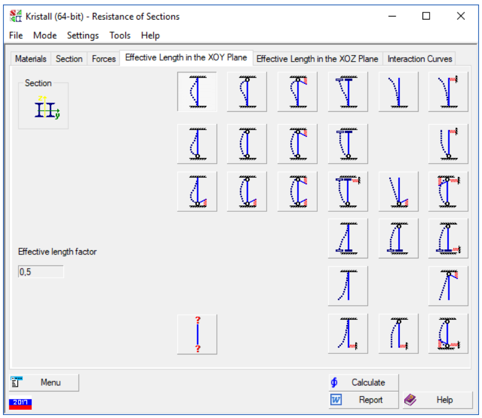


Figure 4.9.1-5. *The* **Effective Length** *tab of the* **Resistance of Sections** *dialog box*

direction. For example, the shear force Q_z is transferred **only** to the web of an I-beam, **only** the flanges of an I-beam resist the shear force Q_y simultaneously. If the section is a custom one, there are no concepts of web and flange, and the application assumes that the shear is resisted by the whole section.

This is the reason for the difference between the results of the analysis of two identical sections: one section created with "standard tools" and the other – by **Section Builder**.

The **Forces** tab (Fig. 4.9.1-4) is used to specify forces acting in the cross-section of the member. It displays a cross-section with the principal axes of inertia and the positive directions of forces. The tab contains a table for specifying the forces acting in the section from one or more load cases. The number of rows in the table corresponds to the number of the load cases and can be increased by clicking the **Add** button. To delete the selected rows, click the **Delete** button.

The **Seismic** checkbox can be checked for some loadings. In this case, requirements of the respective code (selected in the main window) on the use of the additional service factor at the construction in seismic regions will be automatically taken into account. Moreover, a special table will appear in this dialog box where you can specify the coefficients allowing for seismic action at the strength and stability analysis. If zero values are specified, the values are taken in accordance with the respective seismic codes by default. Having specified, for example, these two coefficients equal to 0.9, you can take into account the standard requirements for the calculation of steel structures operating in unheated rooms or in the open air at the design temperature below minus 40 °C.

If the calculation is performed according to SP 16.13330, then the **Special** checkbox can be checked for a certain loading (a special non-seismic loading). In this case you will be able to specify a service factor in accordance with Annex B of SP 296.1325800.2017 in the special table (moreover, the importance factor γ_n = 1.0 in accordance with Sec. 5.5 of SP 296.1325800.2017 will be used for such loadings). Furthermore, an additional service factor which reduces the design strength and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017 can be taken into account.

Note that there is a discrepancy between SP 296.1325800 and SP 385.1325800. SP 296.1325800 introduces a service factor for ductile steels equal to 1.1, but at the same time it speaks of the need for calculations based on the requirements for calculations for progressive collapse (SP 385.1325800). Change No. 1 to SP 385.1325800 provides a service factor of 1.2. The program uses a value of 1.2.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button  above the table. Note that when using sections created by **Section Builder** the forces must be specified in the principal axes, U, V. M_y should be assigned the value of M_u (a moment with respect to the axis of the maximum moment of inertia), M_z should be replaced by the value of M_v, and so on.

To change the load plane, use the respective button. This will transfer the values of M_y and Q_y to the respective columns of the table for M_z and Q_z, and vice versa.

The **Effective Length in the XoY (XoZ) Plane** tabs (Fig. 4.9.1-5) are the exact replicas of the **Effective Lengths** tab for the case of **Separate columns and posts** from the **Effective Lengths** mode, and they suggest 25 possible conditions of end support in the respective planes of loading for a compressed bar member, which differ from one another in combinations of the boundary conditions (free end, hinge, elastic support, elastic clamping, clamped).

This dialog box was described in Section 4.8.4. Unlike the **Effective Lengths** mode, this dialog has the **Effective length factor specified by user** button, . Clicking this button will enable you to enter any desired values for the effective length factor and confirm your choice by clicking the **Apply** button. In all other cases this field is inaccessible.

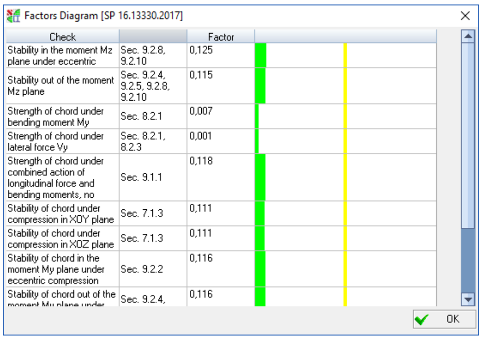


Figure 4.9.1-6. *The* **Factors Diagram** *dialog box*

Once you have entered the initial data, you can click the **Calculate** button, and the $\mathbf{K_{max}}$ field located at the bottom of the dialog will display the maximum (i.e. the most dangerous) value of the checked utilization factors of restrictions and the type of the check (strength, stability, local stability, etc.) in which this maximum took place. You can browse interactively the values of all the other utilization factors of restrictions. To do it, use the **Factors** button which becomes available once the analysis is completed. The **Factors Diagram** dialog box (Fig. 4.9.1-6) displays the respective factors numerically and graphically.

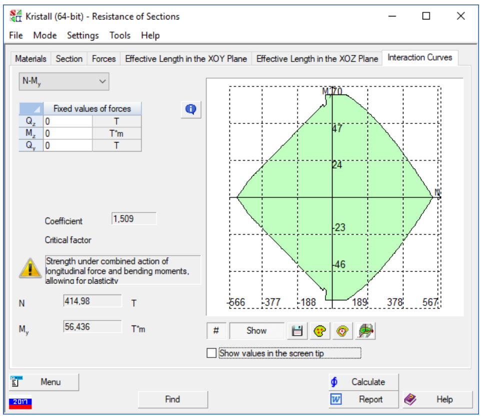


Figure 4.9.1-7. *The* **Interaction Curves** *tab of the* **Resistance of Sections** *dialog box*

Moreover, the curves enclosing an area of the section load-bearing capacity under various pairs of forces which can arise in the considered section are plotted in the **Interaction Curves** tab (Fig. 4.9.1-7). Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of forces, and clicking the button displays a grid in the display field. The curves surround the coordinate origin by a closed line inside which there are points with conditionally acceptable pairs of the considered forces. A pair of forces is deemed acceptable when $K_{max} \leq 1$. All other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the forces variation shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the acting forces; their values are displayed in the respective fields.

Since the slenderness factors do not depend on the forces, they are not calculated when plotting the interaction curves.

The dialog also displays the maximum value of the utilization factor of restrictions that corresponds to these forces and the type of check in which it takes place. When the pointer is placed over a point where $K_{max} > 1$, a warning sign is displayed .

Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the position of the pointer in the plot area of the interaction curve.

The dialog box also contains three buttons: , which enable to perform the following operations:

 — if the forces are specified, clicking this button will draw the points the coordinates of which in the area of the load-bearing capacity correspond to these forces;

 — drawing a convex hull of the points specified above, i.e. an entire set of points which may result from a linear combination of specified forces, including their incomplete values;

 — saving the forces that can lead to K_{max}=1 in a text file (this file can be imported into other programs for further analysis).

4.9.2 Bolted Connections

This mode enables you to check bolted connections of the following three types:

- attachment of single and double angles to a gusset plate;
- connection of sheet elements by gusset plates;
- attachment of the beam web by gusset plates.

The set of checks depends on the type of the connection and the set of loads it is subjected to.

The following checks are performed in accordance with the selected design code:

Attachments of angles are checked for:	**SNiP II-23-81***	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
bearing of the angle	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
shearing of bolts	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec 16.2.9	Sec. 13.7
strength of the angle weakened section	Sec. 5.12, 5.25*	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25

Gusset plate connections are checked for:	**SNiP II-23-81***	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
bearing of the gusset plate	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
bearing of the web	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
shearing of bolts	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
strength of the gusset plate weakened section	Sec. 11.7	Sec. 11.7	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 13.7

Attachments of the beam web are checked for:	**SNiP II-23-81***	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
bearing of the gusset plate	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
bearing of the web	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
shearing of bolts	Sec. 11.7	Sec. 11.7	Sec. 15.2.9	Sec. 14.2.9	Sec. 1.12.2.9	Sec. 16.2.9	Sec. 13.7
strength of the gusset plate weakened section	Sec. 5.12, 5.25*	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25
strength of the beam web weakened section	Sec. 5.12, 5.25*	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25

Limitations

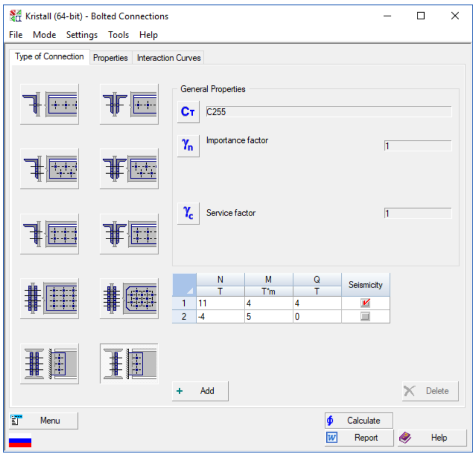
A section weakened by the thread is used in the check of a bolt for shear, because there are many cases when the actual compliance with Sec. 12.18* of SNiP II-23-81* (Sec. 15.2.9 of SP 53-102-2004, Sec. 14.2.9 of SP 16.13330, Sec. 1.12.2.9 of DBN B.2.6-163:2010, Sec. 16.2.9 of DBN B.2.6-198:2014) can not be controlled (for example, when checking the load-bearing capacity of structures in operation for compliance with DBN 362-92).

In the checks the diameter of the bolt hole is assumed to be 3 mm larger than the bolt diameter.

All the connection members (angles, gusset plates, …) are assumed to be made from the same steel.

The **Bolted Connections** dialog box contains three tabs: **Type of Connection**, **Properties**, **Interaction Curves**. The first one (Fig. 4.9.2-1, *a*, *b*) contains buttons for selecting a type of the checked connection, and a table for entering data on the forces acting in it (it is assumed that the point of application of forces is the center of mass of the section). Moreover, this tab contains a typical group of controls, **General Properties**, which is described in Section 3.2.

a



b

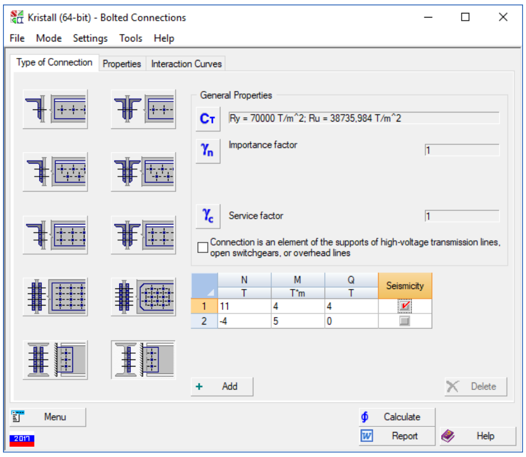


Figure 4.9.2-1. *The* **Type of Connection** *tab of the* **Bolted Connections** *dialog box*
a – according to SNiP II-23-81*, *b* – according to SP 53-102-2004 or SP 16.13330

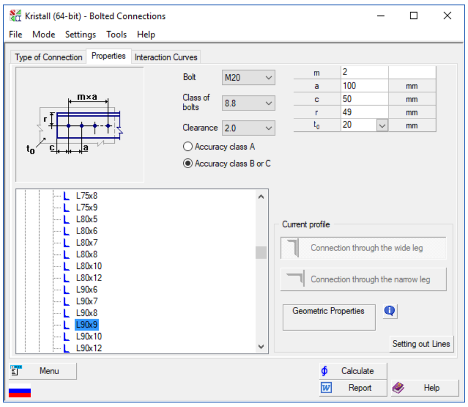


Figure 4.9.2-2. *The* **Properties** *tab of the* **Bolted Connections** *dialog box*

The **Properties** tab (Fig. 4.9.2-2) is used to specify information on the dimensions of the joint, the diameter and class of bolts used, the clearance (the difference between the nominal diameters of the holes and of the bolts).

The application checks the sizes being specified for compliance with requirements of the design codes. The checked values are given in Table 4.9.2-1.

The **Current profile** group enables to specify the leg used for attachment (the wide or narrow one) in joints from unequal angles, and provides the geometric properties for all kinds of profiles.

The button enables you to interactively browse the current size limitations.

You can obtain reference information on the recommended arrangement of the bolt holes within the profile area (the **Setting out Lines** button) for structures with rolled profiles (angles, I-beams).

The **Interaction Curves** tab (Fig. 4.9.2-3) displays interaction

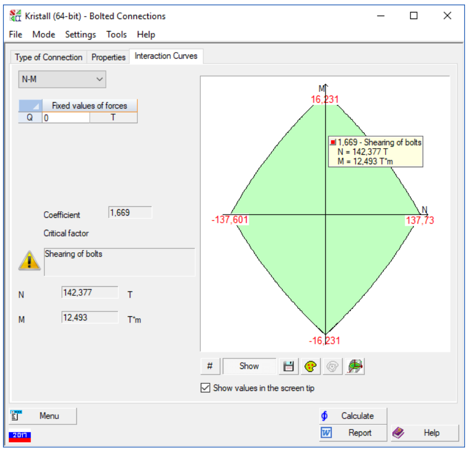
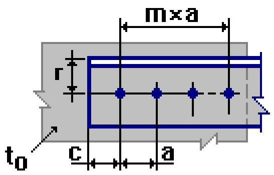
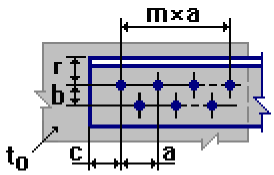
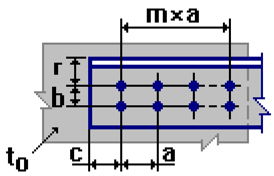
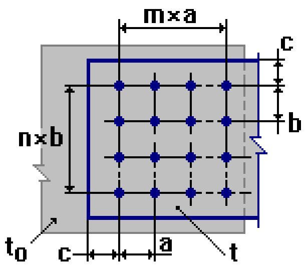
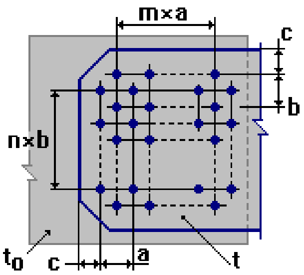
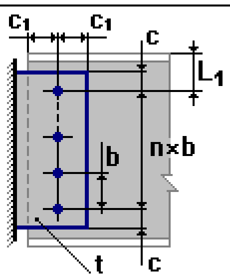


Figure 4.9.2-3. *The* **Interaction Curves** *tab of the* **Bolted Connections** *dialog box*

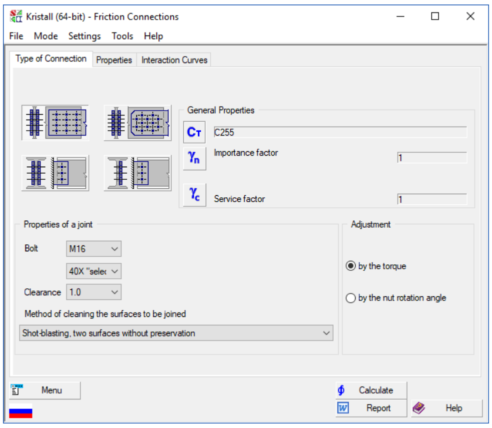
curves of force factors enclosing an area of the load-bearing capacity of the bolted connection. The controls are similar to those described in the **Resistance of Sections** section (see Section 4.9.1).

Table 4.9.2-1

Joint	Checks	Designations
	$m \geq 1$ $c_{maxL} \geq c \geq 1{,}5d_0$ $a_{maxL} \geq a \geq a_{min}$ $r \geq 0{,}95d_0 + R_1 + t_L$ $b_L - r \geq 1{,}2d_0$	d_0 is the diameter of a bolt hole a_{maxL} is the maximum distance between bolts of the end row a_{maxT} is the maximum distance between bolts of the middle row a_{min} is the minimum distance between the bolts R_1 is the corner radius of the rolled profile t_L is the angle thickness b_L is the angle width h_w is the I-beam flange thickness H is the I-beam height c_{maxL} is the maximum distance to the profile edge t_0 is the thickness of the main structure
	$m \geq 1$ $c_{maxL} \geq c \geq 1{,}5d_0$ $a_{maxL} \geq a \geq a_{min}$; $a_{maxL} \geq b \geq 0{,}5a_{min}$ $r \geq 0{,}95d_0 + R_1 + t_L$ $b_L - r - b \geq 1{,}2d_0$	
	$m \geq 1$ $c_{maxL} \geq c \geq 1{,}5d_0$ $a_{maxL} \geq a \geq a_{min}$; $a_{maxL} \geq b \geq a_{min}$ $r \geq 0{,}95d_0 + R_1 + t_L$ $b_L - r - b \geq 1{,}2d_0$	
	$m \geq 1$ $n \geq 1$ $c_{max} \geq c \geq 1{,}5d_0$ $a_{maxT} \geq a \geq a_{min}$; $a_{maxT} \geq b \geq a_{min}$	
	$m \geq 1$ $n \geq 1$ $c_{max} \geq c_1 \geq 1{,}5d_0$ $a_{maxT} \geq a \geq a_{min}$; $a_{maxT} \geq b \geq a_{min}$	
	$n \geq 1$ $c_{max} \geq c_1 \geq 1{,}5d_0$ $c_1 \geq 1{,}5d_0$ $a_{max} \geq b \geq a_{min}$ $H - nb \geq 2L_1$ $L_1 - c \geq (H - h_w)/2$	

4.9.3 Friction Connections

a



b

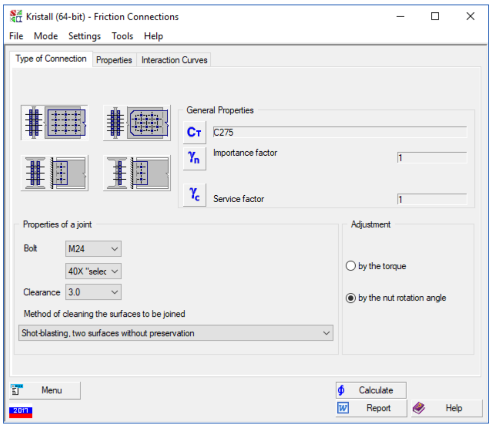


Figure 4.9.3-1. *The* **Type of Connection** *tab of the* **Friction Connections** *dialog box*
a – according to SNiP II-23-81*,
b – according to SP 53-102-2004 or SP 16.13330

The dialog box of this mode is much similar to the **Bolted Connections** mode. The mode enables you to perform checks of the following types of connections with high-strength bolts with controlled tightening:

- connections of sheet elements by gusset plates;
- attachment of the beam web by gusset plates.

The connection type is selected by clicking the respective button of the **Type of Connection** tab (Fig. 4.9.3-1). Friction coefficients μ and safety factors γ_h , defined by Table 36* of SNiP II-23-81* (Table 39 of SP 53-102-2004, Table 39 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, Table 16.2 of DBN B.2.6-198:2014), are calculated by the application on the basis of a cleaning method selected in the **Method of cleaning the surfaces to be joined** drop-down list and an adjustment method selected with a radio button. To calculate γ_h , the difference between the nominal diameters of holes and bolts under static load is assumed to be 1…4 mm.

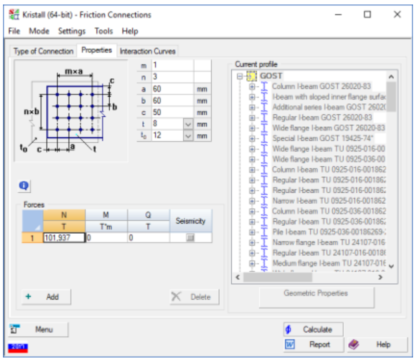


Figure 4.9.3-2. *The* **Properties** *tab of the* **Friction Connections** *dialog box*

The **Properties** tab (Fig. 4.9.3-2) contains tables for entering data on the dimensions of the joint and forces acting in it (it is assumed that the point of application of forces is the center of mass of the section). This tab is also used to specify the beam profile.

The set of checks depends on the type of the connection and the set of loads it is subjected to.

The connections are checked for:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Friction force limit	Sec. 11.13	Sec. 11.13	Sec. 15.3.3	Sec. 14.3.3	Sec. 1.12.3.3	Sec. 16.3.3	Sec. 13.13
Strength of the weakened section	Sec. 5.12, 5.25	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25
Strength of the gusset plate weakened section	Sec. 5.12, 5.25*	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25
Strength of the web weakened section	Sec. 5.12, 5.25*	Sec. 5.12, 5.25	Sec. 9.2.1, 10.1.1	Sec. 8.2.1, 9.1.1	Sec. 1.5.2.1, 1.6.1.1	Sec. 9.2.1, 10.1.1	Sec. 7.12, 7.25

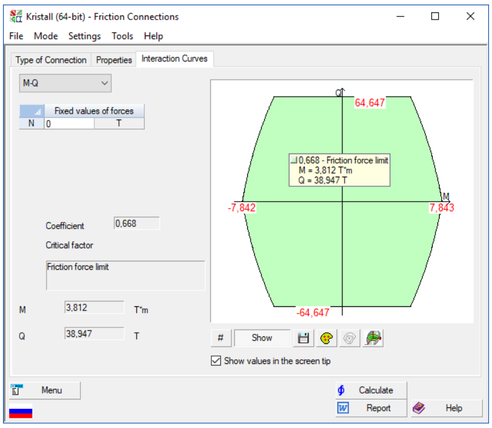
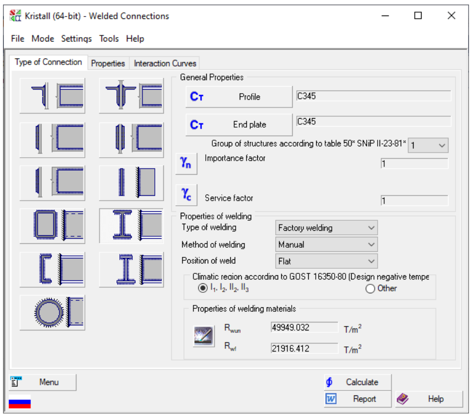


Figure 4.9.3-3. *The* **Interaction Curves** *tab of the* **Friction Connections** *dialog box*

The **Interaction Curves** tab (Fig. 4.9.3-3) displays interaction curves of force factors enclosing an area of the load-bearing capacity of the friction connection. The controls are similar to those described in the **Resistance of Sections** section (see Section 4.9.1).

4.9.4 Welded Connections

a



b

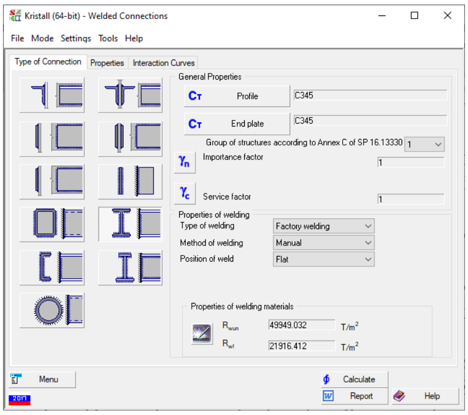


Figure 4.9.4-1. *The* **Type of Connection** *tab of the* **Welded Connections** *dialog box*
a – according to SNiP II-23-81*,
b – according to SP 53-102-2004 or SP 16.13330

The mode enables you to perform checks of welded connections of the following three types:

- attachment of single and double angles to a gusset plate;
- attachment of overlapping steel sheets;
- butt attachment of a steel sheet, I-beam, channel, or pipe.

The set of checks depends on the type of the connection and the set of loads it is subjected to. All types of connections are checked in accordance with Section 11.2, and in necessary cases – with Sections 11.3, 11.5 of SNiP II-23-81*, or according to Section 15.1 of SP 53-102-2004 (Sec. 14.1 of SP 16.13330, Sec. 1.12.1 of DBN B.2.6-163:2010, Sec. 16.1 of DBN B.2.6-198:2014).

The dialog box contains three tabs: **Type of Connection**, **Properties**, **Interaction Curves**.

The first one (Fig. 4.9.4-1) contains buttons for selecting the type of the checked structure. It also contains drop-down lists for specifying data on the type of welding, method of welding, and the position of the weld in the course of work. One option has to be selected in each group, which enables to determine the β_f and β_z factors in accordance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.1 of DBN B.2.6-163:2010, Table 16.1 of DBN B.2.6-198:2014).

Other properties needed for the analysis have been previously defined in the **Steel, Materials for Welding** and **Service Factor** modes. Their values are displayed in the respective fields and can be modified by the user when necessary.

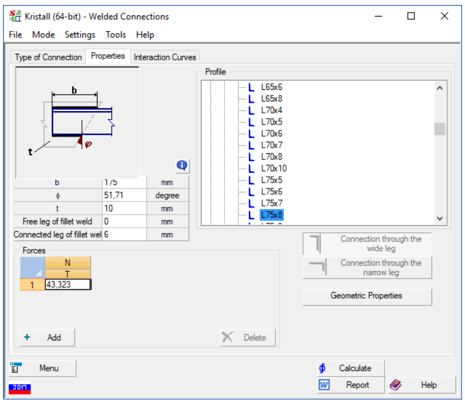
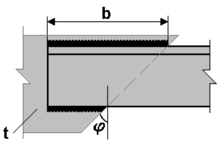
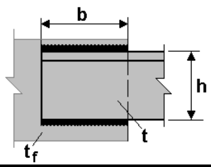


Figure 4.9.4-2. *The* **Properties** *tab of the* **Welded Connections** *dialog box*

The **Properties** tab (Fig. 4.9.4-2) is used to specify information on the dimensions of the joint and to enter data on the forces acting in it (it is assumed that the point of application of forces is the center of mass of the section). This tab is also used to specify the weld legs checked for compliance with the design codes.

Table 4.9.4-1.

Joint	Checks
	$0 \le \varphi < 45^o$ $b \pm h\, tg\varphi \ge 4$ cm $t \ge 0{,}8\ t_{angle}$
	$b \ge 4$ cm $h \ge 4$ cm

Welded connections are checked for:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Of the weld metal	Sec. 11.2 Formula (120), Sec. 11.3 Formula (126)	Sec. 11.2 Formula (147), Sec. 11.3 Formula (163)	Sec. 15.1.16 Formula (155), Sec. 15.1.19 Formula (161)	Sec. 14.1.16 Formula (176), Sec. 14.1.19 Formula (182)	Sec. 1.12.1.16 Formula (1.12.2), Sec. 1.12.1.19 Formula (1.12.8)	Sec. 16.1.16 Formula (16.2), Sec. 16.1.19 Formula (16.8)	Sec. 13.2, 13.3
Of the metal of the fusion border	Sec. 11.2 Formula (121), Sec. 11.3 Formula (126)	Sec. 11.2 Formula (148), Sec. 11.3 Formula (163)	Sec. 15.1.16 Formula (156), Sec. 15.1.19 Formula (161)	Sec. 14.1.16 Formula (177), Sec. 14.1.19 Formula (182)	Sec. 1.12.1.16 Formula (1.12.3), Sec. 1.12.1.19 Formula (1.12.8)	Sec. 16.1.16 Formula (16.3), Sec. 16.1.19 Formula (16.8)	Sec. 13.2, 13.3

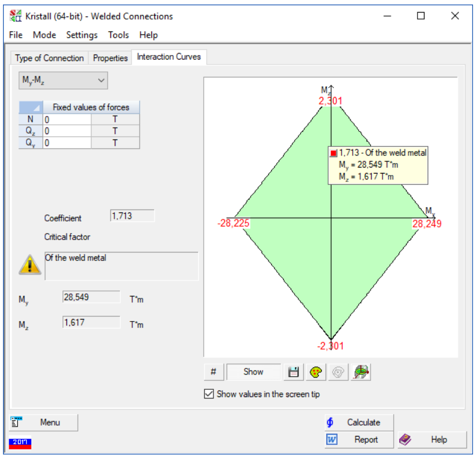


Figure 4.9.4-3. *The* **Interaction Curves** *tab of the* **Welded Connections** *dialog box*

The **Interaction Curves** tab (Fig. 4.9.4-3) displays interaction curves of force factors enclosing an area of the load-bearing capacity of the welded connection. The controls are similar to those described in the **Resistance of Sections** section (see Section 4.9.1).

4.9.5 Local Stability

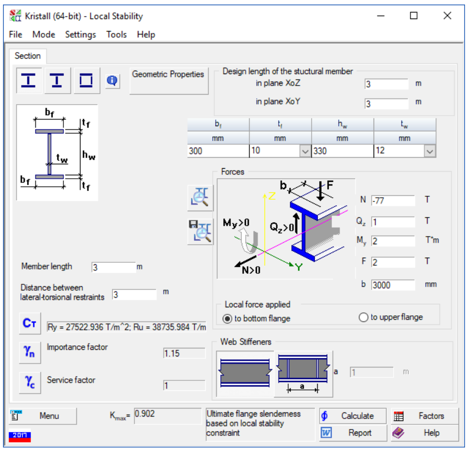


Figure 4.9.5-1. *The* **Local Stability** *mode*

This mode implements local stability checks for webs and flanges of steel structural members with a symmetric or asymmetric I-sections, or a box sections. You can enter the initial data and browse the results in a dialog box shown in Fig. 4.9.5-1.

Local stability is checked for webs of beam structures. The application considers only two types of designs: designs with transverse double-sided stiffeners over the whole height of the web, and designs without stiffeners.

Local stability checks are performed for webs and flanges of beam, column, and beam-column steel structural members. The application considers only two types of structural designs: designs with transverse double-sided stiffeners over the whole height of the web, and designs without stiffeners.

Unlike the **Resistance of Sections** modes this mode enables to perform the analysis of local stability taking into account the local stresses in the beam web from the concentrated force.

The following checks are performed in accordance with the selected design code:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.133 30	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. . 7.1, 7.2, 7.3, 7.4–7.6, 7.9, 7.10, 7.23, 7.26, 7.27, 7.28	Sec. 8.3.2, Table 8; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20 ;	Sec. 7.3.2, Table 9; Sec. 8.5.1 –8.5.9; Sec. 9.4.2, Table 22 ;	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ;	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ;	Sec. 9.1, 9.2, 9.3, 9.4, 9.5 – 9.7, 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30;	Sec. 7.32, 7.33, 7.34	Sec. 8.3.7, Table 9; Sec. 9.5.14; Sec. 10.4.7, Table 21 ;	Sec. 7.3.8, Table 10; Sec. 8.5.1 8; Sec. 9.4.7, Table 23 ;	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14; Sec. 1.6.4.8, Table 1.6.4 ;	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ;	Sec. 9.22, 9.23, Sec. 9.24
Beam web height to thickness ratio	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3	Sec. 9.5

The mode calculates the value of K_{max} and enables to browse the values of all factors using a diagram which can be opened by clicking the **Factors** button.

Limitations

The crane beam structures and structural designs with single-sided web stiffeners or longitudinal web stiffeners are not considered.

4.9.6 Resistance of Sin-Sections

This mode enables to determine the load-bearing capacity of steel welded corrugated I-beams. In the general case, the analyzes are performed for a longitudinal force, bending moments and shear forces acting in the principal planes of inertia. The whole set of checks for strength and stability of load-bearing corrugated I-beams is implemented in compliance with SP 294.1325800.2017 or DBN B.2.6-198:2014.

The set of checks of corrugated I-sections depends on the set of internal forces and moments, and is given below.

Check	SP 294.1325800.2017	DBN B.2.6-198:2014
Strength under action of longitudinal force N	Sec. 20.6.2.1	Sec. 24.2.1
Stability at flexural buckling about the y-y axis under action of longitudinal force N	Sec. 20.6.2.2	Sec. 24.2.2
Stability at flexural buckling about the z-z axis under action of longitudinal force N	Sec. 20.6.2.2	Sec. 24.2.2
Strength under action of bending moment M_y	Sec. 20.6.3.1	Sec. 24.3.1
Strength under action of lateral force Q_z	Sec. 20.6.3.1	Sec. 24.3.1
Strength under action of bending moment M_z	Sec. 20.6.3.1	Sec. 24.3.1
Strength under action of lateral force Q_y		
Strength under combined action of bending moments M_y and M_z	Sec. 20.6.3.1	Sec. 24.3.1
Strength under combined action of bending moment My and lateral force Qz	Sec. 20.6.3.3	Sec. 24.3.3
Strength under combined action of bending moment M_z and lateral force Q_y		
Stability of in-plane bending under bending moment M_y	Sec. 20.6.3.4	Sec. 24.3.4
Stability of in-plane bending under biaxial bending	Sec. 20.6.3.5	Sec. 24.3.5
Local stability of corrugated web under bending	Sec. 20.6.3.6	Sec. 24.3.6
General stability of corrugated web under bending	Sec. 20.6.3.6	Sec. 24.3.6
Local stability of chord overhang of beam with corrugated web under bending	Sec. 20.6.3.12	Sec. 24.3.12
Strength under combined action of longitudinal force N and bending moments M_y, M_z	Sec. 20.6.4.1	Sec. 24.4.3
Stability in the moment plane under eccentric compression	Sec. 20.6.4.2	Sec. 24.4.4
Stability out of the moment plane under eccentric compression	Sec. 20.6.4.3	Sec. 24.4.5
Strength of corrugated web under action of lateral force	Sec. 20.6.4.5	Sec. 24.4.7
Local stability of chord overhang of beam with corrugated web under compression	Sec. 20.6.4.7	Sec. 24.4.9
Local stability of chord overhang of beam with corrugated web under eccentric compression	Sec. 20.6.4.7	Sec. 24.4.9
Stability in web plane under combined action of compression and bending in two planes of inertia	Sec. 20.6.4.4	Sec. 24.4.6
Stability out of web plane under combined action of compression and bending in two planes of inertia	Sec. 20.6.4.4	Sec. 24.4.6

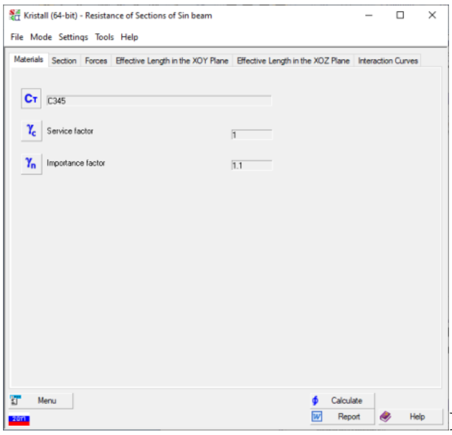


Figure 4.9.6-1. *The* **Materials** *tab of the* **Resistance of Sin-Sections** *dialog box*

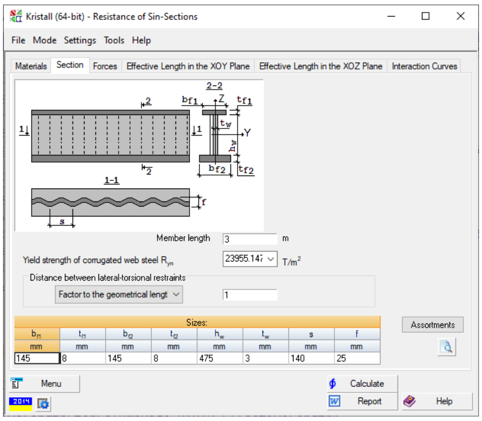


Figure 4.9.6-2. *The* **Section** *tab of the* **Resistance of Sin-Sections** *dialog box*

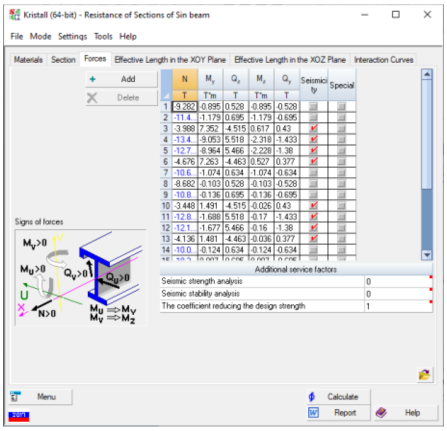


Figure 4.9.6-3. *The* **Forces** *tab of the* **Resistance of Sin-Sections** *dialog*

The dialog box of this mode contains the following tabs: **Materials** (Fig. 4.9.6-1), **Section** (Fig. 4.9.6-2), **Forces** (Fig. 4.9.6-3), **Effective Length in the XoY Plane**, **Effective Length in the XoZ Plane**, **Interaction Curves** (Fig. 4.9.6-4). The first five tabs are used to enter the initial data, and the sixth one to analyze the results of the calculation.

The **Materials** tab (Fig. 4.9.6-1) contains buttons for accessing the reference modes **Steel** — CT, **Service Factor** — γ_c, **Importance Factor** — γ_n. Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again.

The **Section** tab (Fig. 4.9.6-2) contains the **Sizes** table for specifying the cross-sectional dimensions of a load-bearing welded corrugated I-beam: the width and thickness of the lower and upper flanges of the I-beam, the web's height, the corrugated web thickness, corrugation spacing and amplitude. Moreover, the length of the load-bearing member has to be specified in the **Member length** field in this tab.

The **Section** tab (Fig. 4.9.6-2) also contains the **Design strength of corrugated steel Ry** field for specifying different values of the design strength of steel for the flanges of the welded I-beam and for its corrugated web. Here you can also set information about the shape of the web (sinusoidal or triangular).

There is also a field for specifying the distance between out-of-plane restraints, which serves as an effective length in the check of the stability of in-plane bending. The respective drop-down list enables to select a method of specifying this distance (Effective length or the Effective length factor).

The **Assortment** button allows you to select the geometric dimensions of the section from variations of I-beams with corrugated walls.

The **Forces** tab (Fig. 4.9.6-3) is used to specify forces acting in the cross-section of the load-bearing corrugated I-beam. It displays a cross-section with the principal axes of inertia and the positive directions of forces. The tab contains a table for specifying the forces acting in the section from one or more load cases. The number of rows in the table corresponds to the number of the load cases and can be increased by clicking the **Add** button. This tab is described in detail in Sec. 4.9.1

box

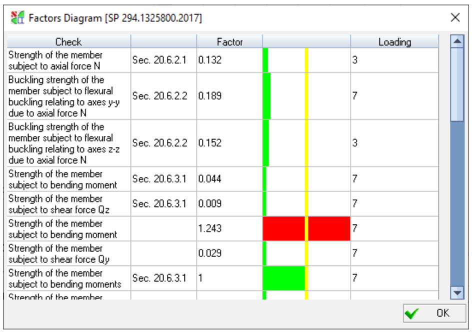


Figure 4.9.6-4. *The* **Factors Diagram** *dialog box*

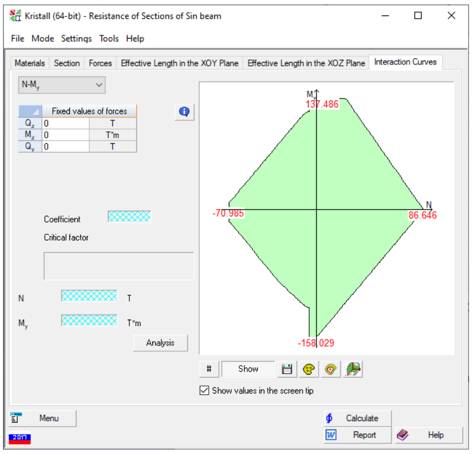


Figure 4.9.6-5. *The* **Interaction Curves** *tab of the* **Resistance of Sin-Sections** *dialog box*

Once you have entered the initial data, you can click the **Calculate** button, and the **K$_{max}$** field located at the bottom of the dialog will display the maximum (i.e. the most dangerous) value of the checked utilization factors of restrictions and the type of the check (strength, stability, local stability, etc.) in which this maximum took place. You can browse interactively the values of all the other utilization factors of restrictions. To do it, use the **Factors** button which becomes available once the analysis is completed. The **Factors Diagram** dialog box (Fig. 4.9.6-4) displays the respective factors numerically and graphically.

The curves enclosing an area of the load-bearing capacity of a welded corrugated I-section under various pairs of forces which can arise in the considered section are plotted in the **Interaction Curves** tab (Fig. 4.9.6-5). This tab is described in detail in Sec. 4.9.1.

4.10 Design of Structural Members

4.10.1 Trusses

This mode enables to perform all the necessary strength and stability checks of truss members, and it also checks their slenderness. The work begins with calculating the design values of the forces caused by the given external loads in structural designs most frequently used in practice. There is an option of finding a section from a pre-composed assortment of cross-sections.

The mode performs the following checks in compliance with the selected design code for each member of the truss:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.1	Sec. 8.1.1	Sec. 7.1
Stability in the	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
truss plane							
Stability in the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Stability out of the truss plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability out of the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*; Sec. 7.14, Table 27*	Sec. 7.1, 7.2, 7.23	Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.10	Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.11	Sec. 1.4.3.1, 1.4.3.2, Table 1.4.3	Sec. 8.3.1, 8.3.2, Table 8.3	Sec. 9.1, 9.2, 9.3, Sec. 9.15
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.27*	Sec. 7.32, 7.33, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10	Sec. 7.3.8, Table 10; Sec. 7.3.11	Sec. 1.4.3.7, Table 1.4.4	Sec. 8.3.7, Table 8.4	Sec. 9.22, 9.23, Sec. 9.27
Pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2	Sec. 10.6
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9	Sec. 10.5-10.13
Stability of the curved member							
Rigidity of the truss							
Slenderness	Sec. 6.1-6.4, 6.16	Sec. 6.1-6.4, 6.15	Sec. 11.1.1-11.1.4, 11.4.1	Sec. 10.1.1-10.1.4, 10.4.1	Sec. 1.9.1.1-1.9.1.4, 1.9.4.1	Sec. 13.1.1-13.1.4, 13.4.1	Sec. 8.1-8.4, 8.18

Limitations

The following values of the service factor, γ_c, are used for finding and checking the truss members:

- 0,95 for chords, support diagonals, lattice members in tension, lattice members in compression with cross-shaped sections;
- 0,8 for lattice members in compression of tee section, the slenderness being greater than 60.

Since the standard requirement to the limitation of the slenderness of a tension member is related to the limitation of its self-weight sag, the slenderness check of tension members is performed only in the vertical plane (for example, according to Note 1 to Table 33 of SP 16.13330).

The truss plane is assumed to be the vertical one in the **Trusses** mode. Slenderness of the tension truss members out of the truss plane is not checked. Thus, the out-of-plane bracing of the bottom (tension) chord does not affect the result of its slenderness check.

If the user has to check the slenderness of the tension members out of the truss plane (for horizontal and inclined trusses), the **Resistance of Sections** mode can be used. When the calculation of a tension member is performed in the **Resistance of Sections** mode, both slenderness values are checked, because the very concept of a vertical plane is absent in this mode.

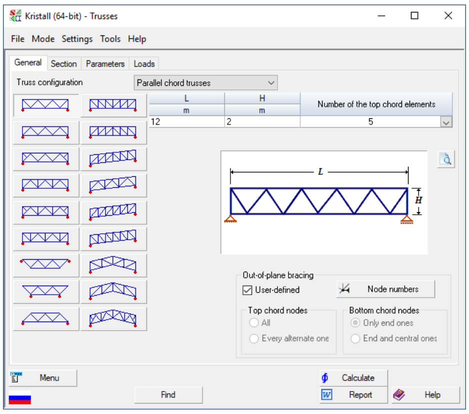


Figure 4.10.1-1. *The* **General** *tab of the* **Trusses** *mode*

The dialog box for this mode contains four tabs: **General**, **Section**, **Properties** and **Loads**.

The **General** tab (Fig. 4.10.1-1) contains a drop-down list for selecting the truss type by its chord shape and a group of buttons for selecting the truss configuration. The following types of trusses can be analyzed: parallel chord, triangular, trapezoid, with a polygonal top chord, one slope roof trusses and double slope roof trusses. All trusses are statically determinate and are assumed to be fixed in the end nodes of their bottom chord in a statically determinate way according to a beam scheme.

Specify the span of the truss and its height on the support for the selected configuration. Additional geometric parameters have to be specified in case of a trapezoid truss.

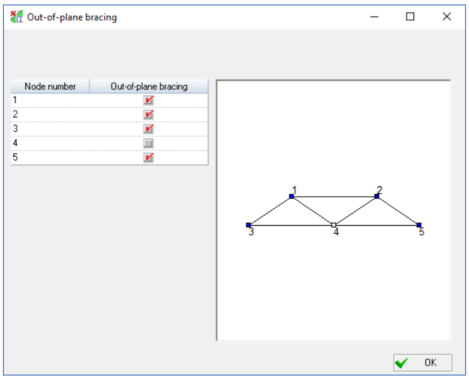


Figure 4.10.1-2. *The* **Out-of-plane bracing** *dialog box*

The respective radio buttons and checkboxes are used to specify the method of out-of-plane bracing of the top and bottom chord nodes (the bracing in the truss plane is assumed to be statically determinate: a hinge support for the left support node and a roller support for the right one). When the **User-defined** checkbox is checked, the **Node numbers** button becomes accessible. Clicking this button invokes the **Out-of-plane bracing** dialog box (Fig. 4.10.1-2). The dialog provides a design model of the truss with numbered nodes and a table where each truss node is assigned a checkbox. A checked checkbox means that there is a bracing in the given node. The braced nodes are highlighted in blue in the model. Nodes in grey are braced by default and their condition cannot be modified.

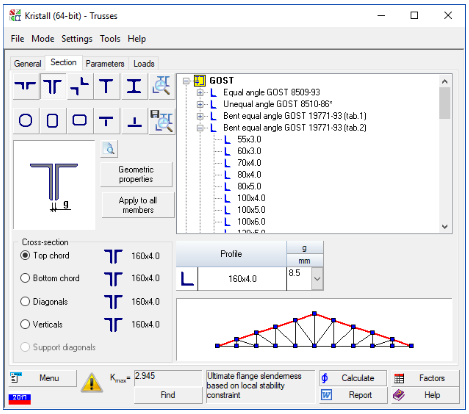


Figure 4.10.1-3. *The* **Section** *tab of the* **Trusses** *mode*

The **Section** tab (Fig. 4.10.1-3) is used to assign cross-sections to truss members. It is assumed that neither the sections of the chords nor those of the lattice members vary along the truss. The sections are made from double equal or unequal angles arranged as a tee (the latter come in two variations) or a cross of equal angles; and from round and rectangular pipes.

The sections are selected from a database of rolled profiles. The gap between the angles is specified in a table above the truss model when selecting a profile for each section type. Members of the same type selected with the respective checkboxes are highlighted in red in the model, and the selection field displays their section.

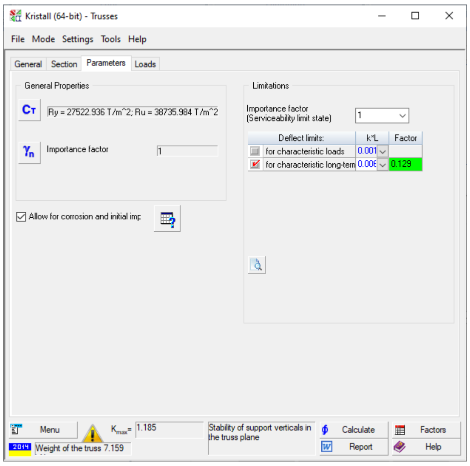


Figure 4.10.1-4. *The* **Properties** *tab of the* **Trusses** *mode*

The **Properties** tab (Fig. 4.10.1-4) enables you to specify a deflection limit (for characteristic and/or characterisric long-term loads) of the truss in fractions of its span length (it will be compared with the relative deflection under the loads with design values which correspond to the serviceability limit state). You can specify the deflection for the truss in fractions of the span expressed as $1/A$ where A is one of the most frequently used values (500, 750, etc.).

The **Web instability is forbidden** checkbox is used to perform the check of the section taking into account its postbuckling behavior (after the local buckling of the web). The checked checkbox enables to reject the postbuckling behavior of the section if the check indicates local buckling of the web.

This mode enables to perform the check analysis taking the corrosion into account. Moreover, you can specify not only the information on corrosion but also the values of initial imperfection for each truss member made from double angles. To do it, check the **Allow for corrosion and initial imperfections** checkbox and click the button , which invokes a dialog box (see Fig. 4.10.1-5). This dialog is used to enter information on the damages of the structure (once you have entered the information, the button changes into:).

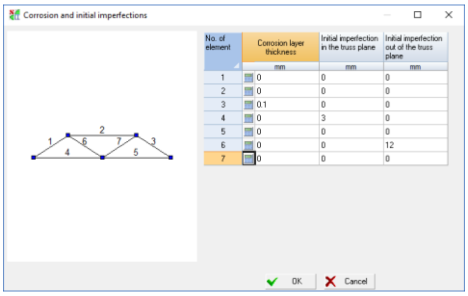


Figure 4.10.1-5. *The* **Corrosion and initial imperfections** *dialog box*

The table is used to specify the data on the thickness of the corrosion layer for each member of the truss (numbers of the members are shown in the model) and the value of the initial imperfection both in the truss plane and out of the truss plane. Moreover, every row of the table has the button , which invokes the calculation of the corrosion layer thickness (Fig. 4.10.1-5). Note that the inclination angle of a member to the horizon does not need to be specified because it is calculated automatically for all the truss members.

If you check the **Apply for all elements of the bottom (top, ...) chord** checkbox in the **Corrosion layer thickness** dialog box, the result of the analysis will be inserted not only into the current row of the table but also into all the rows that correspond to the members of the whole group. The prediction of the corrosion layer thickness complies with [2], [3], [9]. The analysis is based on an assumption that the corrosion layer thickness is uniform over the whole perimeter of the member section.

The behavior analysis of the damaged structure will be performed according to the recommendations of [6]. The analysis takes into account the possibility for the member with an initial imperfection to experience spatial buckling, therefore the set of factors generated in the result of the analysis may not include the results of in-plane or out-of-plane stability checks, or the results of both of these checks.

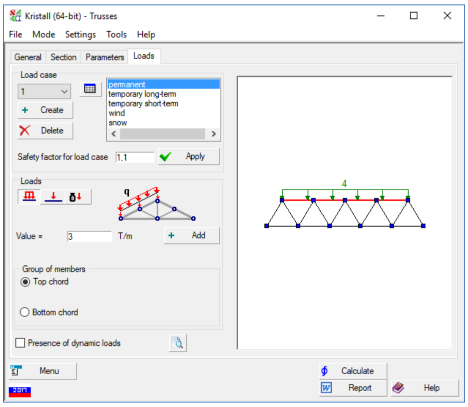
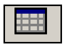


Figure 4.10.1-6. *The* **Loads** *tab of the* **Trusses** *mode*

The **Loads** tab (Fig. 4.10.1-6) is much similar to that described in the **Envelopes** mode (see Section 4.9.1). However, there are some differences. The user can specify either a uniformly distributed load or a concentrated load upon nodes. The distributed load is applied to either the whole chord (upper or lower) or to the half of it. The load application area is defined using the respective radio buttons in the **Group of elements** group. Number of the load application node is selected from the **Node number** drop-down list.

The graphical control of the input data can be performed in the special window located in the **Loads** tab on the right and displaying a truss model. Once you click the button to select the chord the uniformly distributed load is applied to, the respective chord will be highlighted in red in this window, and when you select the number of the node the concentrated load is applied to, the respective node will be highlighted in red.

After clicking the **Add** button, a schematic of the respective load case with all the specified loads will appear in the window.

To edit the values of particular loads, you can use a table invoked by the button  (see Section 4.9.1).

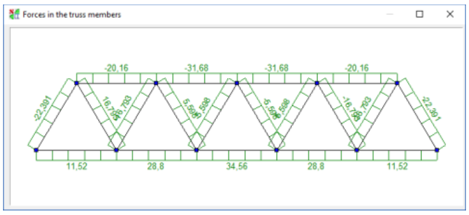


Figure 4.10.1-7. *The* **Forces in truss members** *information window*

Clicking the **Forces in the truss members for the current load case** button invokes an information window displaying a design model of the truss with a diagram of forces (Fig. 4.10.1-7).

Kristall (unlike SCAD) assumes that a truss is subjected to a nodal load. Thus the specified distributed load is not transferred onto the truss members; instead, it is assumed to be applied to some enclosing roof structure which performs the function of transferring the load onto the truss nodes.

As a rule, the top truss chord is subjected to a distributed load, and this fact is taken into account in the truss analysis performed by SCAD. The longitudinal force changes over the length in inclined members. SCAD calculates and displays maximum value of the force in the member. When converting the distributed load into the nodal loads, Kristall will display a value corresponding to the force in the middle of a SCAD finite element. Therefore the results generated by the two applications may differ.

It is possible to indicate the presence or absence of the dynamic loads on the truss. If all loads are static, the slenderness check of tension members is performed only in the vertical plane.

Clicking the **Calculate** button will display the value of K_{max} and indicate the type of check (strength, stability, slenderness) in which the maximum takes place. You can also browse all the other utilization factors of restrictions by clicking the **Factors** button.

The **Find** button enables you to switch to a mode that performs the purposeful search of cross-sections for the truss members and replaces the numbers of profiles selected by the user (the cross-section type and bracing conditions are not modified). The application switches to the next number of profile greater in area in the catalogue the cross-sections were initially selected from if during the checks of the considered member (e.g., a top chord) the value of the utilization factor of restrictions $K > 1$ has been detected, or it switches to the next number of profile smaller in area if $K < 1$. Such switches are performed until all checks give $K < 1$ and the replacement of the profile with its adjacent smaller one gives $K > 1$, i.e. until the smallest profile that satisfies the requirements of the design codes is found.

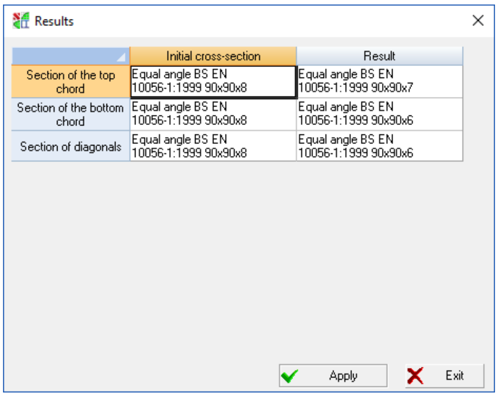


Figure 4.10.1-8. *The* **Results** *dialog box*

The searches through different groups of members (the top and bottom chords, diagonals, verticals) are mutually independent.

When the search is completed, a dialog with the recommendations on the selection of cross-sections appears on the screen (Fig. 4.10.1-8).

The user can reject these recommendations (using the **Exit** button) or accept them. In the latter case the **Apply** button is clicked, and all the recommended sections are transferred to perform the check analysis of the new structure.

If the maximum profile of the assortment was used in the search process and the value was still $K > 1$, the dialog will display the respective message and the **Apply** button will become inactive.

Note that the limitation of the deflection does not affect the results of the search. Moreover, the report document will contain a table with support reactions, total weight of the truss, and paint area.

4.10.2 Truss Member

The **Trusses** mode described above provides the most common but not complete set of truss structures. To analyze members of an arbitrary truss, the application contains a **Truss Member** mode which enables to analyze the load-bearing capacity or to find the section for a truss member (it is assumed that the static problem has been solved previously and the forces in the member are known). The following checks are performed in this mode:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.1	Sec. 8.1.1	Sec. 7.1
Stability in the truss plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability in the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Stability out of the truss plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability out of the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*; Sec. 7.14, Table 27*	Sec. 7.1, 7.2, 7.23	Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.10	Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.11	Sec. 1.4.3.1, 1.4.3.2, Table 1.4.3	Sec. 8.3.1, 8.3.2, Table 8.3	Sec. 9.1, 9.2, 9.3, Sec. 9.15
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.27*	Sec. 7.32, 7.33, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10	Sec. 7.3.8, Table 10; Sec. 7.3.11	Sec. 1.4.3.7, Table 1.4.4	Sec. 8.3.7, Table 8.4	Sec. 9.22, 9.23, 9.27
Pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2	Sec. 10.6
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9	Sec. 10.5-10.13
Stability of the curved member							
Slenderness	Sec. 6.1-6.4,6.16	Sec. 6.1-6.4, 6.15	Sec. 11.1.1-11.1.4, 11.4.1	Sec. 10.1.1-10.1.4, 10.4.1	Sec. 1.9.1.1-1.9.1.4, 1.9.4.1	Sec. 13.1.1-13.1.4, 13.4.1	Sec. 8.1-8.4, 8.18

Limitations

The following values of the service factor, γ_c, are used for finding and checking the truss members:

- 0,95 for chords, support diagonals, lattice members in tension, lattice members in compression with cross-shaped sections;
- 0,8 for lattice members in compression of tee section, the slenderness being greater than 60.

Truss members are calculated for longitudinal forces by default. A special checkbox enables to take into account the bending moments and shear forces as well. In this case the calculation is similar to that performed in the **Resistance of Sections** mode, but the values of effective length and limit slenderness are selected automatically in accordance with the recommendations of the codes for truss members.

Since the standard requirement to the limitation of the slenderness of a tension member is related to the limitation of its self-weight sag, the slenderness check of tension members is performed only in the vertical plane (for example, according to Note 1 to Table 33 of SP 16.13330).

The truss plane is assumed to be the vertical one in the **Truss Member** mode. Slenderness of the tension truss members out of the truss plane is not checked. Thus, the out-of-plane bracing of the bottom (tension) chord does not affect the result of its slenderness check.

If the user has to check the slenderness of the tension members out of the truss plane (for horizontal and inclined trusses), the **Resistance of Sections** mode can be used. When the calculation of a tension member is performed in the **Resistance of Sections** mode, both slenderness values are checked, because the very concept of a vertical plane is absent in this mode.

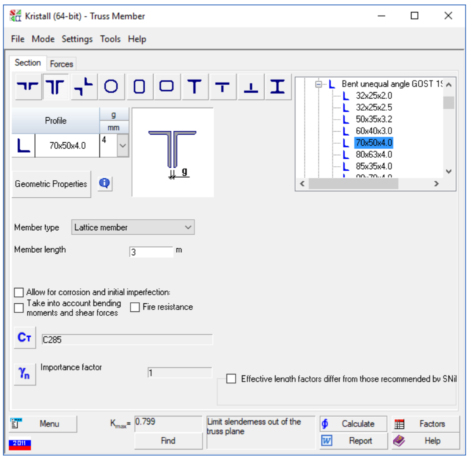


Figure 4.10.2-1. *The* **Section** *tab of the* **Truss Member** *dialog box*

The **Section** tab (Fig. 4.10.2-1) is used to assign a cross-section to a truss member. The sections are made from double equal or unequal angles arranged as a tee (the latter come in two variations) or a cross (of equal angles), or from round and rectangular pipes. The sections are selected from a database of rolled profiles and a gap between the angles is entered. The type and the length of the member are also specified. When a chord member is considered, you need to specify the panel length and the length between out-of-plane restraints. The following member types can be analyzed: a chord member, a lattice member, a support diagonal, a support vertical.

A separate tab is used to specify forces (longitudinal forces) in a member under various load cases. It is possible to indicate the presence or absence of the dynamic loads or crane loads on the truss. If all loads are static, the slenderness check of tension members is performed only in the vertical plane.

The **Web instability is forbidden** checkbox is used to perform the check of the section taking into account its postbuckling behavior (after the local buckling of the web). The checked checkbox enables to reject the postbuckling behavior of the section if the check indicates local buckling of the web.

If a truss member has additional bracing or weakening, you can use the **Effective length factors differ from those recommended by SNiP** checkbox to enter nonstandard effective length factors in the truss plane and out of the truss plane.

The mode enables to perform the check analysis taking the corrosion into account. Moreover, you can specify the values of initial imperfection for members made of double angles, and if equal angles are used, you can also analyze local defects such as notches and bents. To do it, check the **Allow for corrosion and initial imperfections** checkbox in the **Section** tab. This will add one more tab to the dialog, **Defects** (Fig. 4.10.2-2), where you have to specify the data on the thickness of the corrosion layer, the value of the initial imperfection both in the truss plane and out of the truss plane, the data on the local notches or bents.

Using the button, you can invoke the calculation of the corrosion layer thickness (Fig. 4.10.2-2).

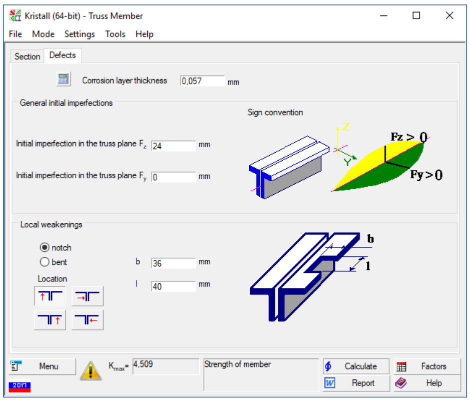


Figure 4.10.2-2. *The* **Defects** *tab of the* **Truss Member** *dialog box*

The behavior analysis of the damaged structure will be performed according to the recommendations of SNiP 2.08.01-85 and the Guide [5]. The analysis takes into account the possibility for the bar with an initial imperfection to experience spatial buckling, therefore the set of factors generated in the result of the analysis may not include the results of in-plane or out-of-plane stability checks, or the results of both of these checks.

Clicking the **Calculate** button will display the value of K_{max} and indicate the type of check (strength, stability, slenderness) in which this maximum took place. You can also browse all the other utilization factors of restrictions by clicking the **Factors** button.

The search is performed according to the same rules as in the **Trusses** mode.

4.10.3 Beams

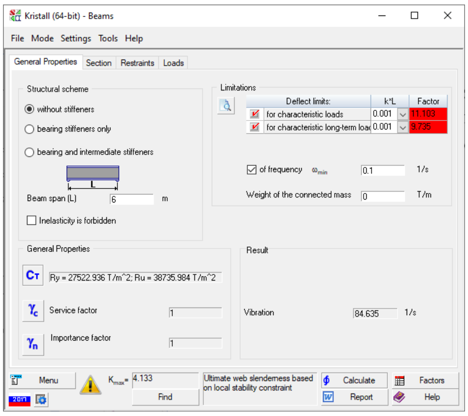


Figure 4.10.3-1. *The* **General Properties** *tab of the* **Beams** *dialog box*

This multitab dialog box enables to perform checks of beams made of rolled or welded I-beams, channels, rectangular pipes and welded box. The dialog contains five tabs: **General Properties**, **Section**, **Restraints**, **Loads**, and **Web Stiffeners**.

The following checks are performed for the beams:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of bearing stiffener	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 1.5.5.13	Sec. 9.5.13	Sec. 9.13
Bearing stiffener in bearing	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 1.5.5.13	Sec. 9.5.13	Sec. 9.13
Strength of girth weld	Sec. 11.5	Sec. 11.5	Sec. 15.1.19	Sec. 14.1.19	Sec. 1.12.1.19	Sec. 16.1.19	Sec. 13.5
Strength of bearing stiffener weld	Sec. 11.5	Sec. 11.5	Sec. 15.1.19	Sec. 14.1.19	Sec. 1.12.1.19	Sec. 16.1.19	Sec. 13.5
Strength under lateral force	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength under bending moment	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Bending strength taking into account plastic deformation	Sec. 5.18*	Sec. 5.18	Sec. 9.2.3	Sec. 8.2.3	Sec. 1.5.2.3	Sec. 9.2.3	Sec. 7.18
Lateral-torsional buckling under moment	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	Sec. 7.15
Lateral-torsional buckling taking into account plastic			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6	

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
deformation							
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10	Sec. 9.5.1–9.5.9	Sec. 8.5.1–8.5.9	Sec. 1.5.5.1–1.5.5.9	Sec. 9.5.1–9.5.9	Sec. 9.1 – 9.6; Sec. 9.10, 9.11
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.24, Table 30	Sec. 7.34	Sec. 9.5.14	Sec. 8.5.18	Sec. 1.5.5.14	Sec. 9.5.14	Sec. 9.24
Height to thickness ration of the beam web	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3	Sec. 9.5
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.14
Strength for local normal stresses under the concentrated force	Sec. 5.13	Sec. 5.13	Sec. 9.2.2	Sec. 8.2.2	Sec. 1.5.2.2	Sec. 9.2.2	Sec. 7.13
Deflection of beam							

In the case when the maximal normal stresses in the beam section exceed the design strength of steel, the local stability check of the beam webs and flanges is performed allowing for the development of plastic deformations.

If the web has such a slenderness that the element can be classified as an element with a flexible web, the application outputs the **Ratio between height and width of the web** factor with a value greater than 1,0. The calculation of elements with a flexible web is not implemented in the program due to the extremely limited scope of the method for calculating such structures (only for continuous beams bearing the static load).

Limitations

There is no analysis of welds attaching the web stiffeners and no check of minimum sizes of the welds for compliance with Table 38* of SNiP II-23-81* (Table 35 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.1 of DBN B.2.6-163:2010, Table 16.1 of DBN B.2.6-198:2014).

The procedure of the stability check of beams (stability of in-plane bending) implemented in SNIP "Steel Structures" requires information on the type of load and its application area (top or bottom chord). The codes provide recommendations only for a few particular cases, when all the loads are applied only to one chord. How to proceed in the cases when there are different loadings and moreover, bending moments are taken from the envelope, rather than from a single specific loading, remains unclear. Therefore, developers of the program (focused on the consideration of the loading in the general case) have assumed that in the calculation of the coefficient ψ we consider the case of a distributed load, without analyzing the particular cases given in Table 77 of SNIP II-23-81* (all further modifications of the codes have not made any changes to this problem).

However, a special dialog box can be invoked by clicking the button , where you can select one of the rules for calculating this factor provided by the codes (the user is responsible for the consequences of this selection), or use the **Default** button to return to the standard behavior of the program (described above).

The **General Properties** tab (Fig. 4.10.3-1) is used to specify the span of the beam. Moreover, the **Structural scheme** group contains radio buttons for specifying a system of web stiffeners:

- without stiffeners;
- bearing stiffeners only;
- bearing and intermediate stiffeners.

If a structure with intermediate stiffeners has been selected, you need to specify their spacing in the respective field. It does not have to be an exact submultiple of the beam span; the end segments of the beam will be adjusted to fit.

After checking the respective checkboxes, you can specify limitations of the absolute deflection value (for characteristic and/or characteristic long-term loads) or of the natural oscillation frequency in the **Limitations** group. In the latter case you can specify the weight of the connected mass which will be added to the own mass of the structure. The deflection limit for the beam is specified in fractions of its span length (it will be compared with the relative deflection under the loads with design values which correspond to the serviceability limit state). You can specify the deflection of the beam in fractions of the span expressed as $1/A$ where A is one of the most frequently used values (500, 750, etc.).

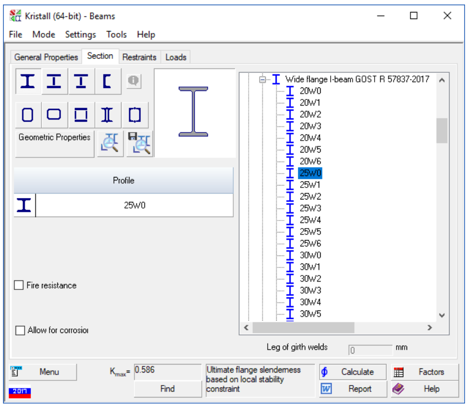


Figure 4.10.3-2. *The* **Section** *tab of the* **Beams** *dialog box*

Note that no special factor is introduced for the deflection in **Kristall** — the application only calculates and outputs the maximum deflection value.

In cases when the user for some reasons does not want to go beyond the elastic behavior, he can use the **Inelasticity is forbidden** checkbox provided in this tab.

The **Section** tab (Fig. 4.10.3-2) is used to assign a cross-section to the beam.

If the section is a rolled I-beam or a channel then it should be selected from the catalogues. If the beam cross-section is a welded I-beam, then you need to specify the sizes of its constituent sheets. The thickness of the sheets can be either taken from the suggested list (they comply with those in the assortment) or specified by the user.

This tab also contains a text field for specifying the leg of girth welds. This field becomes accessible when a welded I-beam is selected.

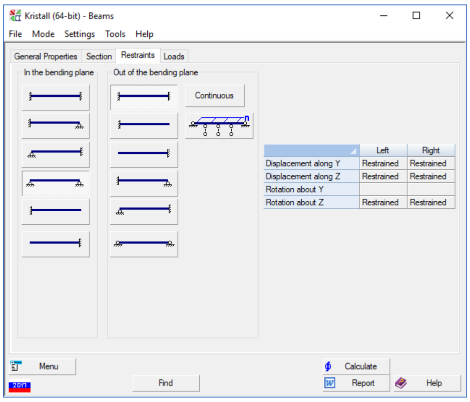


Figure 4.10.3-3. *The* **Restraints** *tab of the* **Beams** *dialog box*

The analysis can be performed with the account of corrosion like in the **Resistance of Sections** mode. The difference is that the built-in corrosion calculation module (invoked with the button) does not require you to specify the inclination of the member to the horizon. The **Restraints** tab (Fig. 4.10.3-3) contains two groups of buttons for specifying a system of beam restraints in and out of the bending plane. The selection within each group is performed independently by clicking a respective button. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

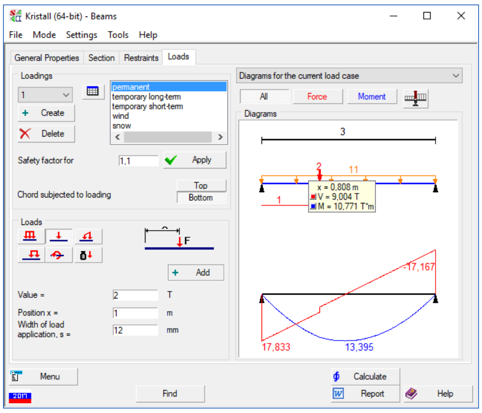


Figure 4.10.3-4. *The* **Loads** *tab of the* **Beams** *dialog box*

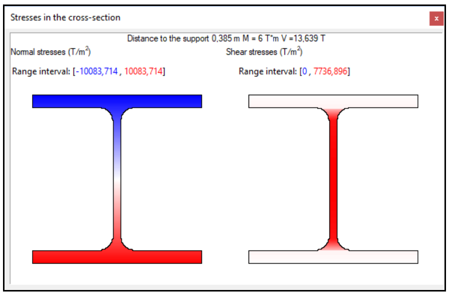


Figure 4.10.3-5. *Normal and shear stresses in the cross-section*

The **Loads** tab (Fig. 4.10.3-4) is used to specify the loads acting on the beam. This tab is nearly identical to that from Section 4.9.1. The difference is that an application area (top or bottom chord) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case. Moreover, you have to specify the width of load application for a "concentrated force". It should be noted that if you place the cursor in the window displaying the diagram and click the right mouse button, this will display a beam cross-section with isofields of normal and shear stresses depending on the specified parameters and corresponding to the section defined by the cursor position (Fig. 4.10.3-5).

The **Web Stiffeners** tab (Fig. 4.10.3-6), which appears if you select the respective structural scheme, is used to specify the sizes of the bearing stiffener and the leg of the weld that attaches this stiffener to the beam web. If the design of the beam with both bearing and intermediate stiffeners is selected, it is necessary to specify the spacing between the intermediate stiffeners in the **Spacing of stiffeners** tab, define their type, and specify their width and thickness in the respective fields.

The type of intermediate stiffeners (single-sided or double-sided) is selected using the respective buttons. The sizes of the stiffeners specified in millimeters are checked for compliance with the requirements of codes concerning the width and the thickness of the protruding part. It is assumed that the width of the bearing stiffener should be not less than that of the narrowest flange of the I-beam.

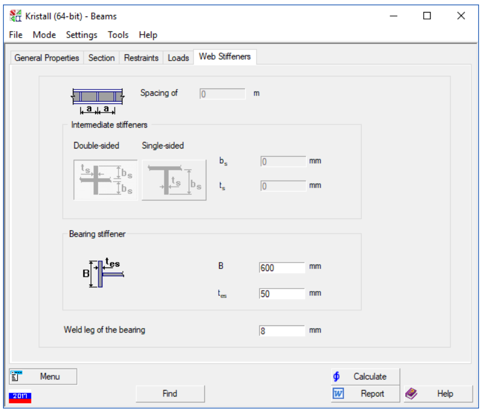


Figure 4.10.3-6. *The* **Web Stiffeners** *tab of the* **Beams** *dialog box*

The **Factors** and **Find** buttons (the latter is available only for the rolled-profile beams) enable you to analyze the calculation results or to perform the search of cross-sections. These modes are described in Section 4.10.1.

It should be noted that the search of sections will be based only on the strength and stability conditions, if you have not specified a deflection limitation in the **General Properties** tab. The limitation of the natural oscillation frequency does not affect the results of the search.

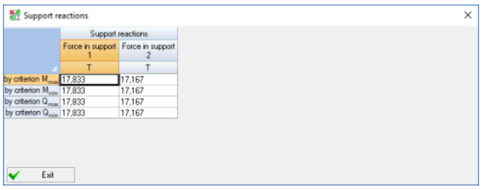
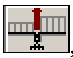


Figure 4.10.3-7. *The* **Support reactions** *dialog box*

Moreover, the report document will contain a table with design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 4.10.3-7), which can be invoked by clicking the respective button, , in the **Loads** tab.

4.10.4 Sin-Beams

The **Sin-Beams** mode enables to perform checks of the load-bearing capacity of beams made of welded corrugated I-beams. The dialog contains the following tabs: **General Properties** (Fig. 4.10.4-1), **Section** (Fig. 4.10.4-2), **Restraints** (Fig. 4.10.4-3), **Loads** (Fig. 4.10.4-4).

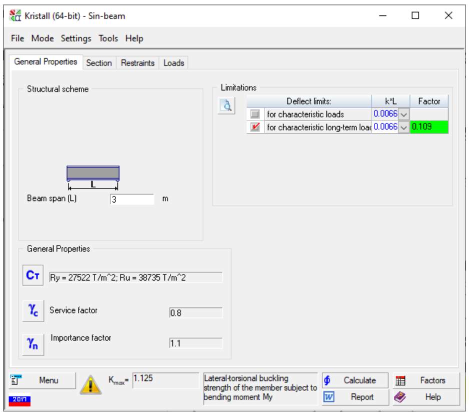
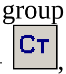
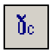
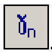


Figure 4.10.4-1. *The* **General Properties** *tab of the* **Sin-Beams** *dialog box*

The **Structural scheme** group of the **General Properties** tab (Fig. 4.10.4-1) is used to specify the span of a single-span beam made of a welded corrugated I-beam. The **General Properties** group contains buttons for accessing the reference modes **Steel** — , **Service Factor** —  and **Importance Factor** — . Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again.

After checking checkboxs in the **Limitations** group you can specify limitations of the absolute deflection value (for characteristic and/or characteristic long-term loads). The deflection limit for the beam is specified in fractions of its span length (it will be compared with the relative deflection under the loads with design values which correspond to the serviceability limit state). You can specify the deflection of the beam in fractions of the span expressed as $1/A$, where A is one of the most frequently used values (500, 750, etc.).

Note that no special factor is introduced for the deflection in the **Sin-Beams** mode — the application only calculates and outputs the maximum deflection value.

The following checks are performed for the Sin-beams:

Check	SP 294.1325800.2017	DBN B.2.6-198:2014
Strength under bending moment My	Sec. 20.6.3.1	Sec. 24.3.1
Strength under lateral force Qz	Sec. 20.6.3.1	Sec. 24.3.1
Strength under combined action of bending moment My and lateral force Qz	Sec. 20.6.3.3	Sec. 24.3.3
Stability of in-plane bending under bending moment My	Sec. 20.6.3.4	Sec. 24.3.4
Local stability of corrugated web under bending	Sec. 20.6.3.6	Sec. 24.3.6
General stability of corrugated web under bending	Sec. 20.6.3.6	Sec. 24.3.6
Local stability of chord overhang of beam with corrugated web under bending	Sec. 20.6.3.12	Sec. 24.3.12
Strength of corrugated web under action of lateral force	Sec. 20.6.4.5	Sec. 24.4.7
Local stability of chord overhang of beam with corrugated web under compression	Sec. 20.6.4.7	Sec. 24.4.9
Deflection of beam		

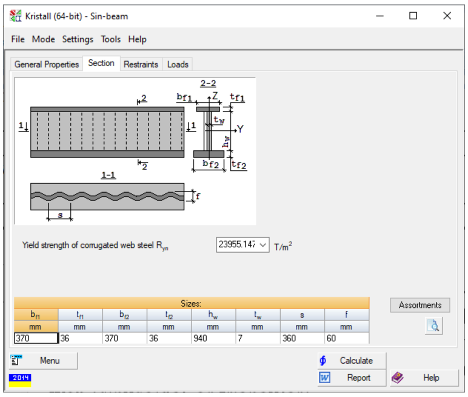


Figure 4.10.4-2. *The* **Section** *tab of the* **Sin-Beams** *dialog box*

The **Section** tab (Fig. 4.10.4-2) is used to specify the cross-sectional dimensions of a welded corrugated I-beam: the width and thickness of the lower and upper flanges of the I-beam, the web's height, the corrugated web thickness, corrugation spacing and amplitude. Here you can also set information about the shape of the web (sinusoidal or triangular).

The **Section** tab also contains the **Design strength of corrugated steel Ry** field for specifying different values of the design strength of steel for the flanges of the welded I-beam and for its corrugated web.

The **Assortment** button allows you to select the geometric dimensions of the section from variations of I-beams with corrugated walls.

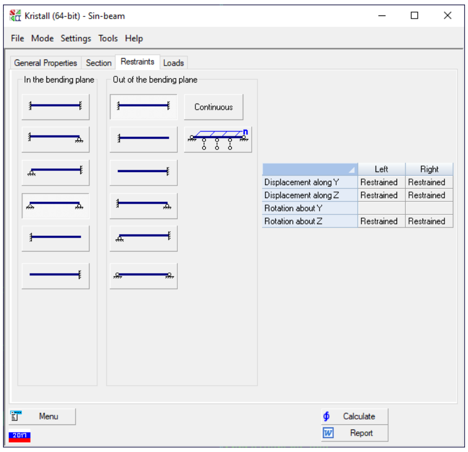


Figure 4.10.4-3. *The* **Restraints** *tab of the* **Sin-Beams** *dialog box*

The **Restraints** tab (Fig. 4.10.4-3) contains two groups of buttons **In the bending plane** and **Out of the bending plane**, for specifying a system of beam restraints in and out of the bending plane. The selection within each group is performed independently by clicking a respective button. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

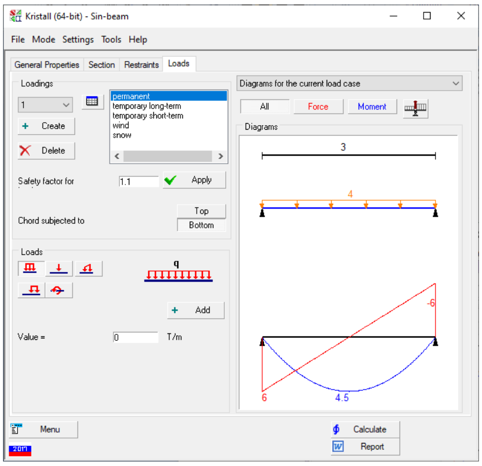


Figure 4.10.4-4. *The* **Loads** *tab of the* **Sin-Beams** *dialog box*

The **Loads** tab (Fig. 4.10.4-4) is used to specify the loads acting on the beam. This tab is nearly identical to that from Section 4.8.1. The difference is that an application area (top or bottom chord) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case.

Click the **Calculate** button to perform the calculation of the load-bearing capacity of the corrugated beam (Fig. 4.10.4-4). When the analysis is completed, the **Factors** button appears, which enables you to analyze the calculation results in the **Factors Diagram** dialog box. See Section 4.10.3

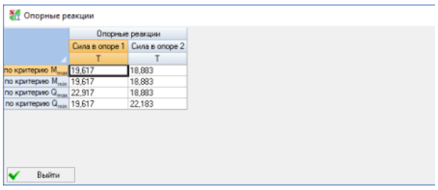
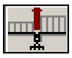


Figure 4.10.4-5. *The* **Support reactions** *dialog box*

Clicking the **Report** button generates a document which among other things includes a table with the design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 4.10.4-5), which can be invoked by clicking the respective button  in the **Loads** tab (Fig. 4.10.4-4).

4.10.5 Continuous Beams

This mode enables to perform checks of beam structures made of rolled I-beams, channels rectangular pipes, and of welded I-beams and welded box. The dialog box of this mode (Fig. 4.10.5-1) contains four tabs: **General Properties**, **Section**, **Loads**, and **Web Stiffeners**.

The following checks are performed for the continuous beams:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of bearing stiffener	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 1.5.5.13	Sec. 9.5.13	Sec. 9.13
Bearing stiffener in bearing	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 1.5.5.13	Sec. 9.5.13	Sec. 9.13
Strength of girth weld	Sec. 11.5	Sec. 11.5	Sec. 15.1.1 9	Sec. 14.1.19	Sec. 1.12.1.1 9	Sec. 16.1.1 9	Sec. 13.5
Strength of bearing stiffener weld	Sec. 11.5	Sec. 11.5	Sec. 15.1.1 9	Sec. 14.1.19	Sec. 1.12.1.1 9	Sec. 16.1.1 9	Sec. 13.5
Strength under lateral force	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength under bending moment	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Lateral-torsional buckling under moment	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	Sec. 7.15

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Lateral-torsional buckling taking into account plastic deformation			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6	
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10	Sec. 9.5.1–9.5.9	Sec. 8.5.1–8.5.9	Sec. 1.5.5.1–1.5.5.9	Sec. 9.5.1–9.5.9	Sec. 9.1 – 9.6; Sec. 9.10, 9.11
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.24, Table 30	Sec. 7.34	Sec. 9.5.14	Sec. 8.5.18	Sec. 1.5.5.14	Sec. 9.5.14	Sec. 9.24
Height to thickness ration of the beam web	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3	Sec. 9.5
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.14
Strength for local normal stresses under the concentrated force	Sec. 5.13	Sec. 5.13	Sec. 9.2.2	Sec. 8.2.2	Sec. 1.5.2.2	Sec. 9.2.2	Sec. 7.13
Deflection of beam							

In the case when the maximal normal stresses in the beam section exceed the design strength of steel, the local stability check of the beam webs and flanges is performed allowing for the development of plastic deformations.

If the web has such a slenderness that the element can be classified as an element with a flexible web, the application outputs the **Ratio between height and width of the web** factor with a value greater than 1,0. The calculation of elements with a flexible web is not implemented in the program due to the extremely limited scope of the method for calculating such structures (only for continuous beams bearing the static load).

Limitations

The procedure of the stability check of beams (stability of in-plane bending) implemented in SNIP "Steel Structures" requires information on the type of load and its application area (top or bottom chord). The codes provide recommendations only for a few particular cases, when all the loads are applied only to one chord. How to proceed in the cases when there are different loadings and moreover, bending moments are taken from the envelope, rather than from a single specific loading, remains unclear.

Therefore, developers of the program (focused on the consideration of the loading in the general case) have assumed that in the calculation of the coefficient ψ we consider the case of a distributed load, without analyzing the particular cases given in Table 77 of SNIP II-23-81* (all further modifications of the codes have not made any changes to this problem). However, a special dialog box can be invoked by clicking the button , where you can select one of the rules for calculating this factor provided by the codes (the user is responsible for the consequences of this selection), or use the **Default** button to return to the standard behavior of the program (described above).

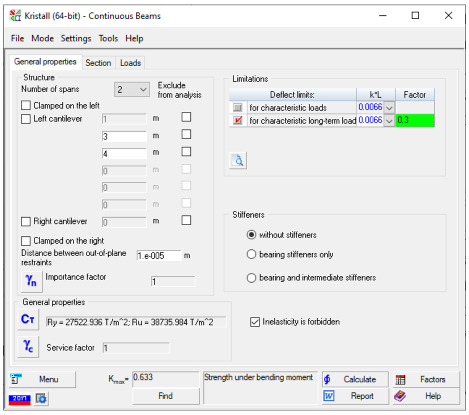


Figure 4.10.5-1. *The* **General Properties** *tab of the* **Continuous Beams** *dialog box*

The design of a multi-span beam is specified in the **Design solution** group of the **General Properties** tab. It is defined by the number of spans, their length and the presence or absence of cantilevers. If you specify stiff clamping (on the right and/or on the left), the respective cantilever can no longer be defined. This tab also contains a field for specifying the spacing between out-of-plane restraints of the beam compressed chord. It is assumed that this spacing is uniform over the whole beam length and that there are such restraints on all supports.

The respective checkboxes are used to model the stiff clamping on the end of the beam.

If you need to perform a check or find a cross-section in several or even in one span rather than in all spans, check the respective checkboxes and exclude spans which are not considered.

The **Stiffeners** group contains radio buttons for specifying a system of web stiffeners:

- without stiffeners;
- bearing stiffeners only;
- bearing and intermediate stiffeners.

When necessary, a limitation of the deflection can be imposed, similarly to the **Beams** mode.

In cases when the user for some reasons does not want to go beyond the elastic behavior, he can use the **Inelasticity is forbidden** checkbox provided in this tab.

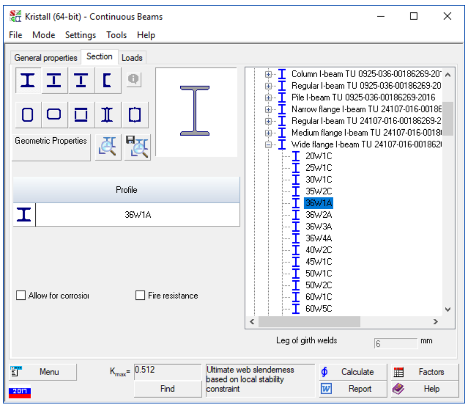


Figure 4.10.5-2. *The* **Section** *tab of the* **Continuous Beams** *dialog box*

The **Section** tab (Fig. 4.10.5-2) is similar to that of the **Beams** mode (see Sec. 4.10.3).

The **Loads** tab (Fig. 4.10.5-3) is used to specify the loads acting on the considered beam. This tab is nearly identical to that from Section 4.10.3. The difference is that the loads are specified for each span. The number of a span (or a cantilever) to which a load is applied is selected from the drop-down list. The position of the concentrated force is measured from the left edge of the span. Furthermore, if you place the cursor in the diagram field and click the right mouse button, this will display a beam cross-section with isofields of normal and shear stresses depending on the specified parameters and corresponding to the section defined by the cursor position (Fig. 4.10.5-5).

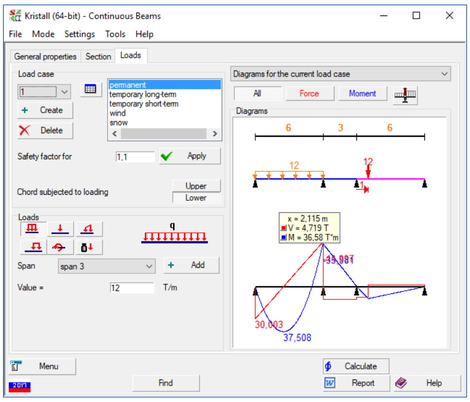


Figure 4.10.5-3. *The* **Loads** *tab of the* **Continuous Beams** *dialog box*

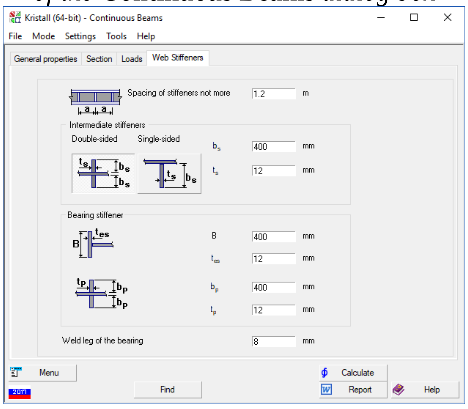
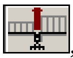


Figure 4.10.5-4. *The* **Web Stiffeners** *tab of the* **Continuous Beams** *dialog box*

The **Web Stiffeners** tab (Fig. 4.10.5-4) appears only in cases when you have selected a structural scheme with stiffeners in the **General Properties** tab. Text fields for specifying the parameters of different designs of stiffeners become accessible depending on the presence of intermediate and end supports.

Click the **Calculate** button to perform the calculation. When the analysis is completed, the **Factors** and **Find** buttons appear (the latter is available only for the rolled-profile beams) which enable you to analyze the calculation results or to perform the search of cross-sections. These modes are described in Section 4.10.3.

Moreover, the report document will contain a table with support reactions.

During the working session, values of the unfavorable combinations of the support reactions are displayed in the **Support reactions** dialog box (Fig. 4.10.3-7), which can be invoked by clicking the respective button, , in the **Loads** tab.

4.10.6 Columns

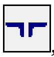
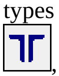
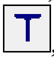
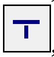
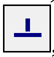
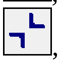
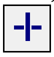
This mode enables to perform checks of columns and posts of solid (rolled or welded I-beams, round or rectangular pipes) and of lattice cross-section. The whole set of checks for strength, stability and slenderness is implemented in compliance with SNiP, SP and DBN. A planar loading is assumed, though checks are performed for two principal planes.

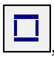
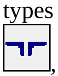
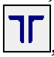
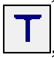
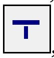
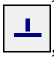
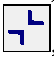
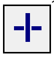
The slenderness checks use the values specified in the **Limit Slenderness** mode. The set of checks depends on the type of the member cross-section and the set of loads it is subjected to.

Check	**SNiP II-23-81***	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.1333 0**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
Strength under axial tension/compression	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.1
Excessive deformations of the tension fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3	Sec. 7.28

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength under action of bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec.7.12
Strength under action of bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12
Strength under action of lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength under action of lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.24, 7.25
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.14
Stability under compression in XoY (XoU) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability under compression in XoY (XoU) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Stability under compression in XoZ (XoV) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3
Stability under compression in XoZ (XoV) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.27, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5	Sec. 7.3, 9.20
Stability in the moment M_y plane under eccentric compression	Sec. 5.27*	Sec. 5.27	Sec. 10.2.9, 10.2.10	Sec. 9.2.9, 9.2.10	Sec. 1.6.2.9, 1.6.2.10	Sec. 10.2.9, 10.2.10	Sec. 7.3
Stability in the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.4.6	Sec. 9.2.2, 9.2.10, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.4.5	Sec. 7.27
Stability in the moment M_z plane under eccentric compression	Sec. 5.27*	Sec. 5.27	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2	Sec. 9.2.9, 9.2.10, 9.3.1, 9.3.2	Sec. 1.6.2.9, 1.6.2.10, 1.6.3.1, 1.6.3.2	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2	Sec. 7.27, 9.20
Stability in the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.1, 10.3.2, 10.4.5	Sec. 7.27
Stability under compression and bending in two planes	Sec. 5.34	Sec. 5.35	Sec. 10.2.9	Sec. 9.2.9	Sec. 1.6.2.9	Sec. 10.2.9	Sec. 7.27, 9.20
Stability under compression and bending in two planes (post-buckling behavior)	Sec. 5.34, 7.20*	Sec. 5.35, 7.30	Sec. 10.2.9, 10.4.6	Sec. 9.2.9, 9.2.10, 9.4.6	Sec. 1.6.2.9, 1.6.2.10, 1.6.4.5	Sec. 10.2.9, 10.2.10, 10.4.5	Sec. 7.34

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability out of the moment M_y plane under eccentric compression	Sec. 5.30-5.32	Sec. 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 9.2.4, 9.2.5, 9.2.8	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 7.34, 9.20
Stability out of the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.30-5.32, 7.20*	Sec. 5.30-5.32, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.5	Sec. 7.30-7.32
Stability out of the moment M_z plane under eccentric compression (sections of the following types , , , , , ,  are not checked)	Sec. 5.27*, 5.30-5.32	Sec. 5.27, 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.2	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 7.30-7.32, 9.20
Lateral-torsional buckling taking into account plastic deformation			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6	Sec. 7.27, 7.30-7.32
Stability out of the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.32, 7.20*	Sec. 5.27, 5.30-5.32, 7.30	Sec. 10.2.4,10.2.5,10.2.8, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2, 10.4.5	Sec. 7.27, 7.30-7.32, 9.20
Strength under axial tension/compression	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.15
Stability in compression of angle	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.1333 0	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Excessive deformations of the tension fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3	Sec. 9.1 – 9.7, Sec. 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18
Stability of in-plane bending (sections of the following types , , , , , , ,  are not checked)	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	Sec. 9.22, 9.23, Sec. 9.24, 9.27
Web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.23, 7.26, 7.27, 7.28	Sec. 8.3.2, Table 8; Sec. 8.3.10 ; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20 ; Sec. 10.4.3; Sec. 10.4.9	Sec. 7.3.2, Table 9; Sec. 7.3.11 ; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22 ; Sec. 9.4.3; Sec. 9.4.9	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ; Sec. 1.6.4.5	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ; Sec. 10.4.5	Sec. 10.6
Flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.27*	Sec. 7.32, 7.33, 7.34, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10 ; Sec. 9.5.14; Sec. 10.4.7, Table 21 ; Sec. 10.4.9	Sec. 7.3.8, Table 10; Sec. 7.3.11 ; Sec. 8.5.18; Sec. 9.4.7, Table 23 ; Sec. 9.4.9	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14 ; Sec. 1.6.4.8, Table 1.6.4 ; Sec. 1.6.4.7	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ; Sec. 10.4.7	Sec. 10.5-10.13
Pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2	Sec. 9.5
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1 -1.10.2.9	Sec. 14.2.1-14.2.9	Sec. 7.3-7.6
Height to thickness ration of the beam web	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8. 5.3	Sec. 1. 5.5.3	Sec. 9. 5.3	Sec. 7.3-7.6

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
General stability of a build-up member under axial compression in XoY plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5	Sec. 7.27, 7.30-7.32
General stability of a build-up member under axial compression in XoZ plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5	Sec. 9.1-9.7, 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18
Stability out of the moment M_z plane	Sec. 5.27*, 5.30-5.32	Sec. 5.27, 5.30-5.32	Secs. 10.2.4, 10.2.5,10.2.8, 10.3.1, 10.3.2	Secs. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.2	Secs. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2	Secs. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 9.22, 9.23, 9.24; Sec. 9.27
Chord web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10, 7.23, 7.26, 7.27, 7.28	Sec. 8.3.2, Table 8; Sec. 8.3.10 ; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20 ; Sec. 10.4.9	Sec. 7.3.2, Table 9; Sec. 7.3.11 ; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22 ; Sec. 9.4.9	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ; Sec. 1.6.4.5	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ; Sec. 10.4.5	Sec. 7.8, 7.9, 7.36
Chord flange slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.27*	Sec. 7.32, 7.33, 7.34, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10 ; Sec. 9.5.14; Sec. 10.4.7, Table 21 ; Sec. 10.4.9	Sec. 7.3.8, Table 10; Sec. 7.3.11 ; Sec. 8.5.18; Sec. 9.4.7, Table 23 ; Sec. 9.4.9	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14 ; Sec. 1.6.4.8, Table 1.6.4 ; Sec. 1.6.4.7	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ; Sec. 10.4.7	Sec. 7.8, 7.9, 7.36
Resistance of a batten to the lateral force	Sec. 5.8, 5.9, 5.36	Sec. 5.8, 5.9, 5.36	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.2.7, 7.2.8, 9.3.7	Sec. 1.4.2.7, 1.4.2.8, 1.6.3.7	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.12
Resistance of a batten to bending	Sec. 5.8, 5.9, 5.36	Sec. 5.8, 5.9, 5.38	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.2.7, 7.2.8, 9.3.7	Sec. 1.4.2.7, 1.4.2.8, 1.6.3.7	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.12
Strength of chord under bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength of chord under bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1	Sec. 7.12, 7.18
Strength of chord under lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3	Sec. 7.24, 7.25, 7.33

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength of chord under lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3	Sec. 7.24, 7.25, 7.33
Strength of chord under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.24, 7.25, 7.33
Strength of chord under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.3, 7.6
Strength of chord under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3	Sec. 7.3, 7.6, 9.20
Stability of chord under compression in XoY plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4	Sec. 7.3, 7.6
Stability of chord under compression in XoY plane (post-buckling)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5	Sec. 7.3, 7.6, 9.20
Stability of chord under compression in XoZ plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4	Sec. 7.27, 7.33, 7.35
Stability of chord under compression in XoZ plane (post-buckling)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5	Sec. 7.27, 7.33, 7.35, 9.20

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of chord in the moment M_y plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27, 5.33, 5.35	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5	Sec. 7.27, 7.33, 7.35
Stability of chord in the moment M_y plane under eccentric compression (post-buckling)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.2, 9.2.10, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.3.3-10.3.5, 10.4.5	Sec. 7.27, 7.33, 7.35, 9.20
Stability of chord in the moment M_z plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27, 5.33, 5.37	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5	Sec. 7.24, 7.25
Stability of chord in the moment M_z plane under eccentric compression (post-buckling)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.2-10.3.4, 10.3.6, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1-9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.2-1.6.3.5, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.2-10.3.5, 10.4.5	Sec. 7.27, 7.30-7.33, 7.35
Bending of chord in two principal planes	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1	Sec. 7.27, 7.30-7.33, 7.35, 9.20
Stability of chord out of the moment M_y plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5	Sec. 7.27, 7.30-7.33, 7.35
Stability of chord out of the moment M_y plane under eccentric compression (post-buckling)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5	Sec. 7.27, 7.30-7.33, 7.35, 9.20

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Stability of chord out of the moment M_z plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5	Sec. 7.1, 7.6
Stability of chord out of the moment M_z plane under eccentric compression (post-buckling)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5	Sec. 7.3, 7.6
Strength of chord under tension	Sec. 5.1, 5.6	Sec. 5.1, 5.6	Sec. 8.1.1, 8.2.1	Sec. 7.1.1, 7.2.1	Sec. 1.4.1.3, 1.4.2.1	Sec. 8.1.3, 8.2.1	Sec. 7.3, 7.6, 9.20
Stability of chord under compression	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3-1.4.2.4	Sec. 8.1.3, 8.2.3-8.2.4	Sec. 7.28
Stability of chord under compression (post-buckling)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3-1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3-8.2.4, 8.3.5	Sec. 7.15
Excessive deformations of the tension chord fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3	Sec. 7.8, 7.10
Stability of in-plane bending of the chord	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1	Sec. 7.8, 7.10
Strength of lattice	Sec. 5.10	Sec. 5.8, 5.10	Sec. 8.2.9	Sec. 7.2.9	Sec. 1.4.2.9	Sec. 8.2.9	Sec. 7.3, 7.8, 7.10
Stability of lattice under compression	Sec. 5.10, 5.3	Sec. 5.8, 5.10, 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3	Sec. 7.3, 7.8, 7.10
Slenderness in XoY plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1	Sec. 8.18
Slenderness in XoZ plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1	Sec. 8.18

Limitations

When determining the relative eccentricity of members under compression and bending the design codes recommend taking the design moment as the moment in the section, which is located in a particular area of the bar. This area is determined depending on the boundary conditions of the bar, on which the program has no information. Therefore, the value of the moment maximal along the length of the element is used.

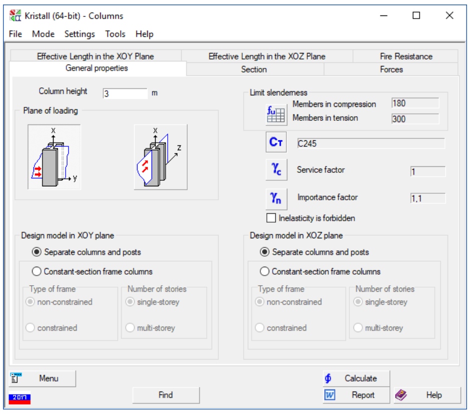


Figure 4.10.6-1. *The* **General Properties** *tab of the* **Columns** *dialog box*

The dialog box of the **Columns** mode contains five tabs: **General Properties**, **Section**, **Forces**, **Effective Length in the XOY Plane**, **Effective Length in the XOZ Plane**.

The **General Properties** tab (Fig. 4.10.6-1) contains a text field for entering the column height and two buttons for selecting the load plane (an orientation of the deformation plane). For frame structures, the load plane is defined by the way a column is incorporated into a planar frame.

This tab also contains radio buttons for specifying a design model according to which the application should calculate the effective length for each of the principal planes.

For members carrying the longitudinal force and the bending moment, design standards for steel structures suggest two possible strength checks:

- taking into account the possible elastoplastic behavior;
- in the absence of the inelastic behavior.

The possibility of the elastoplastic behavior is limited by a number of conditions, such as the absence of the direct action of dynamic loads.

In the case when there is a direct action of dynamic loads, as well as in cases when the user for some other reasons does not want to go beyond the elastic behavior, he can use the **Inelasticity is forbidden** checkbox provided in this tab.

The **Web instability is forbidden** checkbox is used to perform the check of the section taking into account its postbuckling behavior (after the local buckling of the web). The checked checkbox enables to reject the postbuckling behavior of the section if the check indicates local buckling of the web.

The **Section** tab (Fig. 4.10.6-2) enables you to select a cross-section for the column and to specify its properties. Rolled profiles can be selected from the database.

The properties of welded sections are entered into the respective text fields for specifying the thickness and the width of the sheets. Buttons for selecting the lattice type and the text fields for specifying the respective data are used for the lattice cross-sections. Sections of the lattice members are selected from the catalogue of equal or unequal angles.

Particular members of the lattice cross-section can be selected using the respective buttons.

The *length between restraints out of the bending plane* has to be specified (this value will be used in the analysis of stability of in-plane bending).

If the transverse stiffeners can be installed for the given cross-sections, you can use the **Stiffeners** checkbox, thus indicating that the stiffeners are installed, and specify their spacing. If this spacing is greater than the length of the element, the local stability analysis of the web is performed as for a web without stiffeners. If the web has such a slenderness that the element can be classified as an element with a flexible web, the application outputs the **Ratio between height and width of the web** factor with a value greater than 1,0. The calculation of elements with a flexible web is not

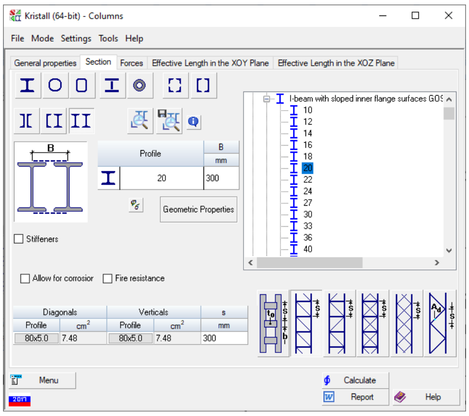


Figure 4.10.6-2. *The* **Section** *tab of the* **Columns** *dialog box*

implemented in the program due to the extremely limited scope of the method for calculating such structures (only for continuous beams bearing the static load).

The analysis can be performed with the account of corrosion like in the **Resistance of Sections** mode. The difference is that the built-in corrosion calculation module (invoked with the button) does not require you to specify the inclination of the member to the horizon.

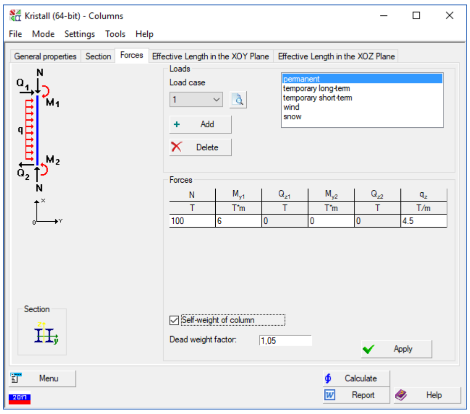


Figure 4.10.6-3. *The* **Forces** *tab of the* **Columns** *dialog box*

The **Forces** tab (Fig. 4.10.6-3) is used to specify all the loads for each load case simultaneously. The general conditions of equilibrium are satisfied for these forces and moments. In particular, shear forces Q_1 and Q_2, as well as the nodal moments M_1 and M_2 should be taken from the results of the analysis of the system as a whole. The conditions of equilibrium are as follows:

$$Q_1 - Q_2 + qL = 0,$$

$$M_2 - M_1 - Q_1L - qL^2/2 = 0.$$

It should be noted that a positive longitudinal force corresponds to compression in this mode.

Clicking the button will open the **Preview** dialog box with diagrams of N, M_y and Q_z. Clicking the **Apply** button will perform the calculation of the lacking force factors on the basis of the conditions of equilibrium.

It should be noted that all the loads act in the XOY plane or in the XOZ plane (the X axis is oriented along the bar). The plane is selected in the **General Properties** tab.

The **Effective Length in the XoY Plane** and **Effective Length in the XoZ Plane** tabs are equivalent to those described in Section 4.8.4 and implement the same capabilities, except for the rules for calculation of the effective lengths in compliance with Eurocode 3. The tabs enable to specify a column configuration and all the parameters necessary for the calculation of the effective lengths. These lengths are calculated for a fragment of a frame system located in the load plane.

If the data on the effective length in one of the planes are specified in the information mode for a frame system, it should be noted that **Kristall** itself calculates the effective length factor based on the same algorithms that are used in the **Effective Lengths** mode. The height of the column and the moment of inertia are automatically selected from the data specified in the **General Properties** tab. Rules for selecting the moment of inertia are as follows:

		Effective length in plane	
		XoY	XoZ
Load Plane	XoY	I_z	
	XoZ		I_y

It should be noted that for cases highlighted in grey in the above table, the codes do not provide any procedures for determining the effective length (the load plane should coincide with the plane of the frame structure).

The **Factors** and **Find** buttons (the latter is available only for the rolled-profile columns) enable you to analyze the calculation results or to perform the search of cross-sections. These modes are described in Section 4.10.1.

4.10.7 Base Plates

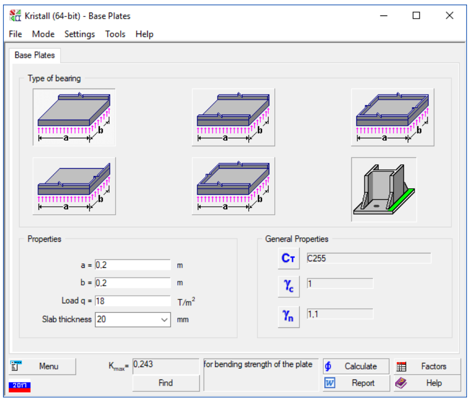


Figure 4.10.7-1. *The* **Base Plates** *dialog box*

This mode enables to select the necessary thickness of a base plate under a column. Parts of the base plate divided by the column end, wing plates and stiffeners are considered.

The dialog contains five possible design models of base plates supported along one, two, three, or four sides (edges). Once you have selected a design model, specify the dimensions of the sides and the load caused by the foundation pressure.

The **Find** mode (Fig. 4.10.7-1) will calculate the needed thickness of the plate after you click the **Calculate** button, while the **Check** mode will ask you for the thickness and calculate the value of K_{max}. To perform a detailed analysis, you can switch between the **Find** and **Check** modes repeatedly.

4.10.8 Sheet Structures

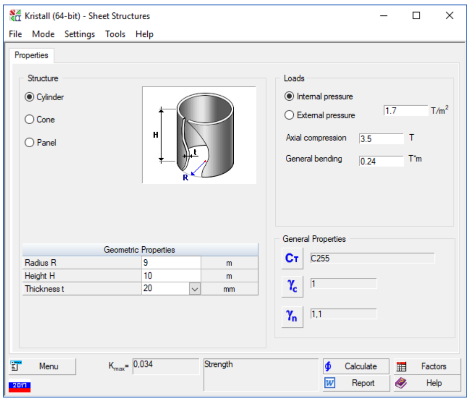


Figure 4.10.8-1. *The* **Sheet Structures** *dialog box*

This mode enables to determine the utilization factors of restrictions for strength and stability of sheet structures of one of three types: circular cylindrical or conical shells, or cylindrical panels.

The structure type is selected using the respective radio buttons in the **Structure** group. The initial data include the geometric properties and values of loads (Fig. 4.10.8-1). To start the calculation, click the **Calculate** button.

The list of checks of the load-bearing capacity for the sheet structures

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Strength	Sec. 8.1-8.3	Sec. 8.1-8.3	Sec. 12.1.1-12.1.3	Sec. 11.1.1-11.1.3	Sec. 1.10.1.1-1.10.1.3	Sec. 14.1.1-14.1.3	Sec. 10.1-10.3
Stability	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9	Sec. 10.5-10.13

4.10.9 Castellated Beams

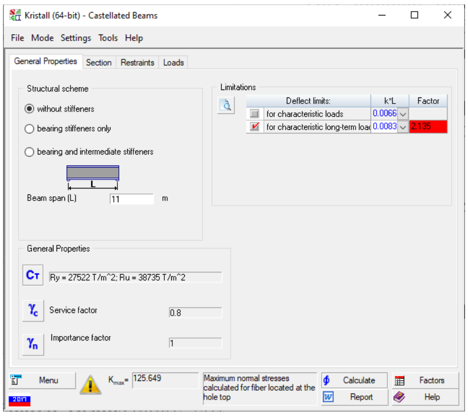


Figure 4.10.9-1. *The* **General Properties** *tab of the* **Castellated Beams** *dialog box*

This multitab dialog box enables to check the load-bearing capacity of a castellated beam made of rolled I-sections. The dialog contains five tabs: **General Properties**, **Section**, **Restraints**, **Loads**, and **Web Stiffeners**.

The **General Properties** tab (Fig. 4.10.9-1) contains the **Beam span (L)** field where you have to specify the geometric length of the castellated beam.

Clicking this button  invokes the dialog box where you can select a steel grade for the castellated beam. Buttons  and  are used to specify the service factor and the importance factor for the castellated beam.

The following checks are performed for the castellated beams:

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330 2011	SP 294.1325800 2017	DBN B.2.6-198:2014	ShNK 2.03.05-13
Maximum normal stresses calculated for the extreme fiber of the top chord	Sec. 19.2	Sec. 19.2	Sec. L.5.2	Sec. M.5.2	Sec. 20.5.2	Sec. 23.2	Sec. 21.2
Maximum normal stresses calculated for the fiber above the hole	Sec. 19.2	Sec. 19.2	Sec. L.5.2	Sec. M.5.2	Sec. 20.5.2	Sec. 23.2	Sec. 21.2
Maximum normal stresses calculated for the extreme fiber of the bottom chord	Sec. 19.2	Sec. 19.2	Sec. L.5.2	Sec. M.5.2	Sec. 20.5.2	Sec. 23.2	Sec. 21.2
Maximum normal stresses calculated for the fiber below the hole	Sec. 19.2	Sec. 19.2	Sec. L.5.2	Sec. M.5.2	Sec. 20.5.2	Sec. 23.2	Sec. 21.2
Maximum shear stresses calculated for the support section of the beam	Sec. 19.2	Sec. 19.2	Sec. L.5.2	Sec. M.5.2	Sec. 20.5.2	Sec. 23.2	Sec. 21.2
Stability of in-plane bending	Sec. 19.3	Sec. 19.3	Sec. L.5.3	Sec. M.5.3	Sec. 20.5.3	Sec. 23.3	Sec. 21.3
Local stability of web	Sec. 7.18*, 19.5	Sec. 7.28, 19.5	Sec. 8.3.2, L.5.5	Sec. 7.3.2, M.5.5	Sec. 7.3.2 SP 16.13330.2017, Sec. 20.5.5	Sec. 23.5	Sec. 9.17, 21.5
Ratio of the beam compressed flange width to effective height of the beam web compressed part	Sec. 7.18*, 19.5	Sec. 7.18, 19.5	Sec. 8.3.2, L.5.5	Sec. 7.3.2, M.5.5	Sec. 7.3.2 SP 16.13330.2017, Sec. 20.5.5 SP 294.1325800.2017	Sec. 23.5	Sec. 9.17, 21.5
Stability of bearing stiffener	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 8.5.17 SP 16.13330.2017	Sec. 9.5.13	Sec. 9.13
Bearing stiffener in bearing	Sec. 7.12	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17	Sec. 8.5.17 SP 16.13330.2017	Sec. 9.5.13	Sec. 9.13
Strength of bearing stiffener weld	Sec. 11.5	Sec. 11.5	Sec. 15.1.19	Sec. 14.1.19	Sec. 14.1.19 SP 16.13330.2017	Sec. 16.1.19	Sec. 13.5
Beam deflection limit	Sec. 19.6	Sec. 19.6	Sec. L.5.6	Sec. M.5.6	Sec. M.5.6 SP 16.13330.2011	Sec. 23.6	Sec. 21.6

The **General Properties** tab (Fig. 4.10.9-1) contains radio buttons in the **Structural scheme** group for specifying a system of web stiffeners of the castellated beam:

- without stiffeners;
- bearing stiffeners only;
- bearing and intermediate stiffeners.

After checking the checkboxs, you can specify limitations of the absolute deflection value (for characteristic and/or characteristic long-term loads) in the **Limitations** group. The deflection limit for the castellated beam is specified in fractions of its span length. The actual value of the deflection of the castellated beam under the loads corresponding to the serviceability limit state will be compared with the deflection limit.

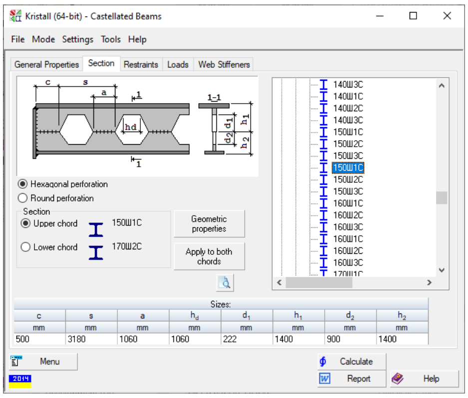


Figure 4.10.9-2. *The* **Section** *tab of the* **Castellated Beams** *dialog box*

The **Section** tab (Fig. 4.10.9-2) is used to assign a cross-section to the castellated beam.

The section of a castellated beam is made up of rolled I-beams, which are generally different for the top and bottom chords. Rolled I-beams can be selected from the respective assortments given on the right in the tab.

Moreover, the user has to specify the sizes of the holes, their spacing and their arrangement along the beam web. With the help of the corresponding radio buttons, you can choose between hexagonal and round perforations.

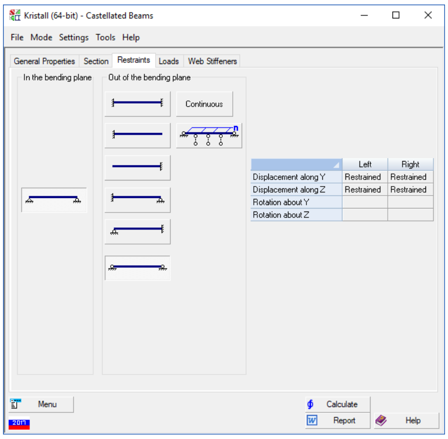


Figure 4.10.9-3. *The* **Restraints** *tab of the* **Castellated Beams** *dialog box*

The **Restraints** tab (Fig. 4.10.9-3) contains two groups of buttons for specifying a system of beam restraints in and out of the bending plane. The selection within each group is performed independently by clicking a respective button.

Only single-span castellated beams simply supported in the bending plane are implemented in this mode due to the methodological limitations of the standard analysis of such structures. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

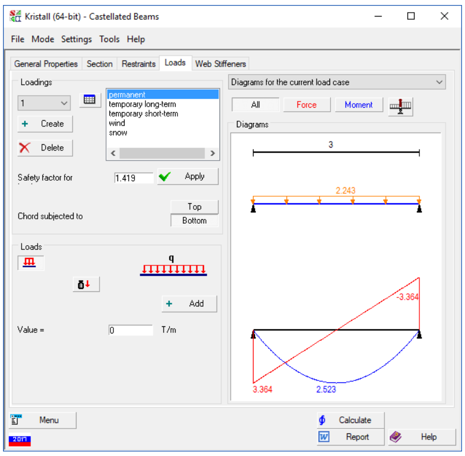


Figure 4.10.9-4. *The* **Loads** *tab of the* **Castellated Beams** *dialog box*

The **Loads** tab (Fig. 4.10.9-4) is used to specify the loads acting on the castellated beam. Only uniformly distributed loads on the castellated beam and the dead weight are implemented in this mode due to the methodological limitations of the standard analysis of such structures. Application area (top or bottom chord of the castellated beam) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case.

The diagrams of internal forces and moments are displayed in the **Diagrams** field in this tab.

The **Web Stiffeners** tab (Fig. 4.10.9-5) appears only in cases when you have selected a structural scheme with stiffeners in the **General Properties** tab (Fig. 4.10.9-1).

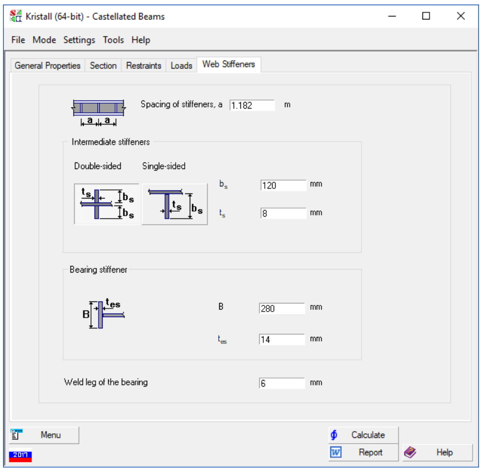


Figure 4.10.9-5. *The* **Web Stiffeners** *tab of the* **Castellated Beams** *dialog box*

This tab is used to specify the sizes (width and thickness) of the bearing stiffener and the leg of the fillet weld that attaches this stiffener to the castellated beam web. It is assumed that the width of the bearing stiffener should be not less than that of the narrowest flange of the I-beam.

If the design of the beam with both bearing and intermediate stiffeners is selected, it is necessary to specify the spacing between the intermediate stiffeners in the **Spacing of stiffeners** tab, define their type, and specify their width and thickness in the respective fields. The type of intermediate stiffeners (single-sided or double-sided) is selected using the respective buttons.

Note that the specified spacing of stiffeners must be a multiple of the distance between the centers of the holes of the castellated beam! If the beam span is not a multiple of the spacing of intermediate stiffeners, the intermediate stiffeners are arranged along the length of the beam by adjusting the extreme segments.

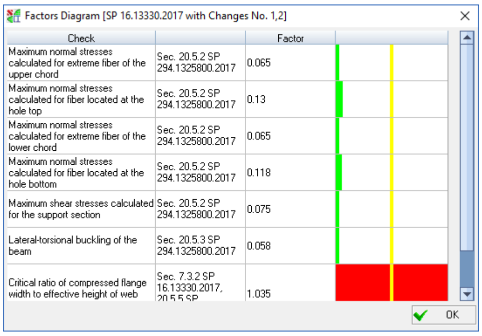


Figure 4.10.9-6. *The* **Factors Diagram** *dialog box of the* **Castellated Beams** *mode*

Clicking the **Factors** button invokes the **Factors Diagram** dialog box (Fig. 4.10.9-6), which displays the results of the load-bearing capacity checks of the castellated beam.

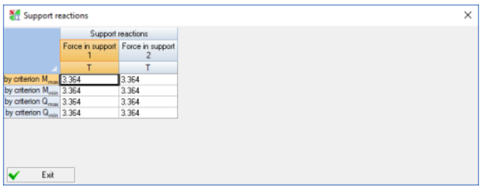
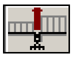


Figure 4.10.9-7. *The* **Support reactions** *dialog box*

Clicking the **Report** button generates a report document which, among other things, contains a table with design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 4.10.9-7), which can be invoked by clicking the respective button  in the **Loads** tab.

Limitations

There is no analysis of welds attaching the web stiffeners and no check of minimum sizes of the welds for compliance with Table 38* of SNiP II-23-81* (Table 35 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.1 of DBN B.2.6-163:2010, Table 16.1 of DBN B.2.6-198:2014).

When the calculation is performed according to SP 16.13330.2017, the deflections of the castellated beam are not determined by the formula (171) of Sec. 20.5.6, since this formula is applicable only for castellated beams with regular hexagonal holes 0.667*h* high, where *h* is the total height of the beam. In this case the application implements a more general approach provided by Sec. M.5.6 of SP 16.13330.2011.

4.11 Appendix

4.11.1 Design Codes the Requirements of Which are Implemented in Kristall

Mode	References to sections of standards or codes						
	SNiP II-23-81*	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
Steel	Table 50* Table 51* Table 51, *b* of SNiP 53-01-96	Table 50* Table 51* Table 51, *b* of SNiP 53-01-96	Sec. 6.1, 6.2, Annex C, Table C1, C2, C5-C8	Sec. 5.1,5.2, Annex. C, Table C1, C2, C5-C8	Annex F., Table F.1	Annex D., Table D.1	Annex E., Table E.3, E.4
Assortment of Rolled Profiles	GOST 26020-83; GOST 8239-89 GOST 8240-89 GOST 8509-93; GOST 8510-86*	GOST 26020-83; GOST 8239-89 GOST 8240-89 GOST 8509-93; GOST 8510-86*	GOST 26020; GOST 8239 GOST 8240 GOST 8509; GOST 8510	GOST 26020; GOST 8239 GOST 8240 GOST 8509; GOST 8510	GOST 26020; GOST 8239 GOST 8240 GOST 8509; GOST 8510	GOST 26020, GOST 8239, GOST 8240, GOST 8509, GOST 8510	GOST 26020-83, GOST 8239-89, GOST 8240-89, GOST 8509-93, GOST 8510-86*
Bolts	Table 57* GOST 1759.4-87	Table 57* GOST 1759.4-87	Sec. 6.5, Annex D, Table D3,D6, D8, D9	Sec. 5.5, Annex D, Table D3, D6, D8, D9	Annex C	Annex A	Table 57*, GOST 1759.4-87
Limit Slenderness	Table 19* Table 20*	Table 19* Table 20*	Sec. 11.4	Sec. 10.4	Table 1.9.9, 1.9.10	Table 13.9, 13.10	
Service Factor	Table 6*	Table 6*	Sec. 5.3.2	Sec. 4.3.2	Sec. 1.12.3.4 Table 1.1.1, 1.12.4	Sec. 16.3.4, Table 16.1, 16.4	Annex F, Table F.1
Effective Lengths	Sec. 6.1–6.4 Sec. 6.5–6.6 Table 71, *a* Table 17, *a* Annex F of EN 1993-1-1	Sec. 6.1–6.4 Sec. 6.5–6.6 Table 71, *a* Table 17, *a* Annex F of EN 1993-1-1	Sec. 11.1, 11.2, 11.4, Annex O, Table O1, O2, O4	Sec. 10.1, 10.2, 11.4, Annex I, Table I1, I2, I4	Sec. 1.9.1–1.9.4 Table 1.9.1, 1.9.2, 1.9.8, 1.9.9	Sec. 13.1–13.4, Table 13.1, 13.2, 13.8, 13.9	Sec. 8.1–8.4; Sec. 8.5, 8.9, Sec. 17.2

Mode	References to sections of standards or codes						
	SNiP II-23-81*	**SNiP RK 5.04-23-2002**	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
Resistance of Sections	Sec. 5.1, 5.3–5.6, 5.8–5.10, 5.12, 5.14*, 5.15, 5.18*, 5.24, 5.25, 5.27*, 5.28, 5.30–5.32, 5.34; Sec. 6.15, 6.16; Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*; Sec. 7.20*; Sec. 7.22*, 7.23*, Table 29*; Sec. 7.24, Table 30; Sec. 7.27; Sec. 8.5–8.13	Sec. 5.1, 5.3–5.6, 5.8–5.10, 5.12, 5.14, 5.15, 5.18, 5.24, 5.25, 5.27, 5.28, 5.30–5.32, 5.35; Sec. 6.14, 6.15; 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.24, 7.26, 7.27, 7.28, 7.30; 7.32, 7.33, 7.34, 7.37; 8.5–8.13	Sec. 8.1.1, 8.1.3–8.1.5; Sec. 8.2.7–8.2.9; Sec. 8.3.2, Table 8; Sec. 8.3.5; Sec. 8.3.7, Table 9; Sec. 8.3.10; Sec. 9.2.1, 9.2.3, 9.4.1, Sec. 9.5.1–9.5.9, Sec. 9.5.14; Sec. 10.1.1, 10.1.3, 10.2.4, 10.2.5, 10.2.8–10.2.10, 10.3.2, Sec. 10.4.2, Table 20; Sec. 10.4.3, 10.4.6, 10.4.7, Table 21; Sec. 10.4.9; Sec. 11.4.1; Sec. 12.2.1–12.2.8	Sec. 7.1.1, 7.1.3–7.1.5; Sec. 7.2.7–7.2.9; Sec. 7.3.2, Table 9; Sec. 7.3.6, Sec. 7.3.8, Table 10; Sec. 7.3.11; Sec. 8.2.1, 8.2.3, 8.4.1, Sec. 8.5.1–8.5.9; Sec. 8.5.18; Sec. 9.1.1, 9.1.3, Sec. 9.2.2, 9.2.4, 9.2.5, 9.2.8–9.2.10, Sec. 9.3.2, Sec. 9.4.2, Table 22; Sec. 9.4.3, 9.4.6, Sec. 9.4.7, Table 23; Sec. 9.4.9; Sec. 10.4.1; Sec. 11.2.1–11.2.9	Sec. 1.4.1.3, 1.4.1.5; Sec. 1.4.2.7–1.4.2.9; Sec. 1.4.3.2, Table 1.4.3; Sec. 1.4.3.5, 1.4.3.7, Table 1.4.4; Sec. 1.5.2.1, 1.5.2.3, 1.5.4.1; Sec. 1.5.5.1–1.5.5.9; Sec. 1.5.5.1 4; Sec. 1.6.1.1, 1.6.1.3, Sec. 1.6.2.2, 1.6.2.4, 1.6.2.5, 1.6.2.8–1.6.2.10; Sec. 1.6.3.2 Sec. 1.6.4.2, Table 1.6.3; Sec. 1.6.4.5, 1.6.4.7, 1.6.4.8, Table 1.6.4; Sec. 1.9.4.1; Sec. 1.10.2.1–1.10.2.9	Sec. 8.1.3, 8.1.5; Sec. 8.2.7–8.2.9; Sec. 8.3.2, Table 8.3; Sec. 8.3.5, Sec. 8.3.7, Table 8.4; Sec. 9.2.1, 9.2.3, Sec. 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.14 ; Sec. 10.1.1 , 10.1.3; Sec. 10.2.2 , 10.2.4, 10.2.5, 10.2.8–10.2.10; Sec. 10.3.2 , Sec. 10.4.2 , Table 10.3; Sec. 10.4.5 , 10.4.7, 10.4.8, Table 10.4; Sec. 13.4.1 ; Sec. 14.2.1–14.2.9	Sec. 7,1, 7.3–7.6, 7.8–7.10, 7.12, 7.14, 7.15, 7.18, 7.24, 7.25, 7.27, 7.28, 7.30–7.32, 7.34; Sec. 8.18; Sec. 9.1. 9.2, 9.3, 9.4, 9.5 – 9.7, 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18; Sec. 9.20, 9.22, 9.23, 9.24, 9.27; Sec. 10.5–10.13
Bolted Connections	Sec. 5.1, 5.12, 5.25* Sec. 11.7	Sec. 5.1, 5.12, 5.25, 11.7	Sec. 9.2.1, 10.1.1, Sec. 15.2.9	Sec. 8.2.1, 9.1.1; Sec. 14.2.9	Sec. 1.5.2.1, 1.6.1.1, Sec. 1.12.2.9	Sec. 9.2.1, 10.1.1, Sec. 16.2.9	Sec. 7.1, 7.12, 7.25 Sec. 13.7
Friction Connections	Sec. 5.12, 5.25*, Sec. 11.13*	Sec. 5.12, 5.25, 11.13	Sec. 9.2.1, 10.1.1, Sec. 15.3.3	Sec. 8.2.1, 9.1.1, Sec. 14.3.3	Sec. 1.5.2.1, 1.6.1.1, Sec. 1.12.3.3	Sec. 9.2.1, 10.1.1, Sec. 16.3.3	Sec. 7.12, 7.25; Sec. 13.13
Welded Connections	Sec. 11.2*, 11.3*, 11.5, 12.8	Sec. 11.2, 11.3, 11.5, 12.8	Sec. 15.1.7, 15.1.9, 15.1.14–15.1.19	Sec. 14.1.7, 14.1.9, 14.1.14–14.1.19	Sec.1.1.1.16, Table 1.12.1	Sec.16.1.16 , Table 16.1	Sec. 13.2
Trusses	Sec. 5.1, 5.3; Sec. 6.1–6.4, 6.16; Sec. 7.14, Table 27*; Sec. 7.1, 7.2*; Sec. 7.14,	Sec. 5.1, 5.3; 6.1–6.4, 6.15, 7.23, 7.1, 7.2, 7.23, 7.30, 7.32, 7.33, 7.36, 7.37, 8.5-8.13	Sec. 8.1.1, 8.1.3; Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.5;	Sec. 7.1.1, 7.1.3; Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.6,	Sec. 1.4.1.1, 1.4.1.3; Sec. 1.4.3.1, 1.4.3.2, Table 1.4.3; Sec. 1.4.3.5, 1.4.3.7,	Sec. 8.1.1, 8.1.3; Sec. 8.3.1, 8.3.2, Table 8.3; Sec. 8.3.5, 8.3.7,	Sec. 7.1, 7.3; Sec. 8.1–8.4, 8.18; Sec. 9.15; Sec. 9.1, 9.2, 9.3; Sec. 9.15, 9.20, 9.22,

Mode	References to sections of standards or codes						
	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
	Table 27*; Sec. 7.20*, 7.22*, 7.23*, Table 29*, Sec. 7.26*, 7.27*; Sec. 8.5-8.13		Sec. 8.3.7, Table 9; Sec. 8.3.10; Sec. 11.1.1–11.1.4, 11.4.1; Sec. 12.2.1-12.2.8	Sec. 7.3.8, Table 10; Sec. 7.3.11; Sec. 10.1.1–10.1.4, 10.4.1; Sec. 11.2.1–11.2.9	Table 1.4.4; Sec. 1.9.1.1–1.9.1.4, 1.9.4.1; Sec. 1.10.2.1–1.10.2.9	Table 8.4; Sec. 13.1.1–13.1.4, 13.4.1; Sec. 14.2.1–14.2.9	9.23, 9.26, 9.27; Sec. 10.5-10.13
Truss Member	Sec. 5.1, 5.3; Sec. 6.1–6.4, 6.16; Sec. 7.14, Table 27*; Sec. 7.1, 7.2*; Sec. 7.14, Table 27*; Sec. 7.20*, 7.22*, 7.23*, Table 29*, Sec. 7.26*, 7.27*; Sec. 8.5–8.13	Sec. 5.1, 5.3, 6.1–6.4, 6.15; 7.23, 7.1, 7.2, 7.23, 7.30, 7.32, 7.33, 7.36, 7.37, 8.5–8.13	Sec. 8.1.1, 8.1.3; Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.5; Sec. 8.3.7, Table 9; Sec. 8.3.10; Sec. 11.1.1–11.1.4, 11.4.1; Sec. 12.2.1–12.2.8	Sec. 7.1.1, 7.1.3; Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.6, Sec. 7.3.8, Table 10; Sec. 7.3.11; Sec. 10.1.1–10.1.4, 10.4.1; Sec. 11.2.1–11.2.9	Sec. 1.4.1.1, 1.4.1.3; Sec. 1.4.3.1, 1.4.3.2, Table 1.4.3; Sec. 1.4.3.5, 1.4.3.7, Table 1.4.4; Sec. 1.9.1.1–1.9.1.4, 1.9.4.1; Sec. 1.10.2.1–1.10.2.9	Sec. 8.1.1, 8.1.3; Sec. 8.3.1, 8.3.2, Table 8.3; Sec. 8.3.5, 8.3.7, Table 8.4; Sec. 13.1.1–13.1.4, 13.4.1; Sec. 14.2.1–14.2.9	Sec. 7.1, 7.3; Sec. 8.1–8.4, 8.18; Sec. 9.15; Sec. 9.1, 9.2, 9.3; Sec. 9.15, 9.20, 9.22, 9.23, 9.26, 9.27; Sec. 10.5-10.13
Beams	Sec. 1.10–1.12 Sec. 5.12, 5.14*, 5.15 Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.12, Sec. 7.24, Table 30; Sec. 11.5	Sec. 1.10–1.12 СНиП 2.01.07-85*; Sec. 5.12, 5.14, 5.15 7.1, 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.12, 7.34, 11.5	Sec. 6.2–6.5 SP 20.13330.2011; Sec. 9.2.1, 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.13, 9.5.14; Sec. 15.1.19	Sec. 6.2–6.5 SP 20.13330.2011; Sec. 8.2.1, 8.4.1; Sec. 8.5.1–8.5.9; Sec. 8.5.17, 8.5.18; Sec. 14.1.19	Sec. 1.5.2.1, 1.5.4.1; Sec. 1.5.5.1–1.5.5.9; Sec. 1.5.5.13, 1.5.5.14; Sec. 1.12.1.19	Sec. 9.2.1, 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.13, 9.5.14; Sec. 16.1.19	Sec. 1.10–1.12 KMK 2.01.07-96; Sec. 7.12, 7.14, 7.15 Sec. 9.1, 9.2, 9.3, 9.4, 9.5 – 9.7, 9.10, 9.11; Sec. 9.13, 9.24; Sec. 13.5
Continuous Beams	Sec. 1.10–1.12 Sec. 5.12, 5.14*, 5.15 Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.12, Sec. 7.24, Table 30; Sec. 11.5	Sec. 1.10–1.12 СНиП 2.01.07-85*; Sec. 5.12, 5.14, 5.15, 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.12, 7.34, 11.5	Sec. 6.2–6.5 SP 20.13330.2011; Sec. 9.2.1, 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.13, 9.5.14; Sec. 15.1.19	Sec. 6.2–6.5 SP 20.13330.2011; Sec. 8.2.1, 8.4.1; Sec. 8.5.1–8.5.9; Sec. 8.5.17, 8.5.18; Sec. 14.1.19	Sec. 1.5.2.1, 1.5.4.1; Sec. 1.5.5.1–1.5.5.9; Sec. 1.5.5.13, 1.5.5.14; Sec. 1.12.1.19	Sec. 9.2.1, 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.13, 9.5.14; Sec. 16.1.19	Sec. 1.10–1.12 KMK 2.01.07-96; Sec. 7.12, 7.14, 7.15; Sec. 9.1, 9.2, 9.3, 9.4, 9.5 – 9.7, 9.10, 9.11; Sec. 9.13, 9.24; Sec. 13.5
Columns	Sec. 5.1, 5.3–5.6, 5.8–5.10, 5.12, 5.14*, 5.15, 5.18*, 5.24, 5.25, 5.27*, 5.28, 5.30–5.32, 5.34;	Sec. 5.1, 5.3–5.6, 5.8–5.10, 5.12, 5.14, 5.15, 5.18, 5.24, 5.25, 5.27, 5.28, 5.30–5.32, 5.35, 6.14, 6.15, 7.1. 7.2,	Sec. 8.1.1, 8.1.3–8.1.5; Sec. 8.2.7–8.2.9; Sec. 8.3.2, Table 8; Sec. 8.3.5; Sec. 8.3.7,	Sec. 7.1.1, 7.1.3–7.1.5; Sec. 7.2.7–7.2.9; Sec. 7.3.2, Table 9; Sec. 7.3.6, Sec. 7.3.8,	Sec. 1.4.1.3, 1.4.1.5; Sec. 1.4.2.7–1.4.2.9; Sec. 1.4.3.2, Table 1.4.3; Sec. 1.4.3.5, 1.4.3.7,	Sec. 8.1.3, 8.1.5; Sec. 8.2.7–8.2.9; Sec. 8.3.2, Table 8.3; Sec. 8.3.5, Sec. 8.3.7,	Sec. 7.1, 7.3–7.6, 7.8–7.10, 7.12, 7.14, 7.15, 7.18, 7.24, 7.25, 7.27, 7.28, 7.30–7.32, 7.34; Sec. 8.18;

Mode	References to sections of standards or codes						
	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
	Sec. 6.15, 6.16; Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*; Sec. 7.20*; Sec. 7.22*, 7.23*, Table 29*; Sec. 7.24, Table 30; Sec. 7.27; Sec. 8.5–8.13	7.3, 7.4–7.6, 7.9, 7.10, 7.23, 7.26, 7.27, 7.28, 7.30; 7.32, 7.33, 7.34, 7.37; 8.5–8.13	Table 9; Sec. 8.3.10; Sec. 9.2.1, 9.2.3, 9.4.1, Sec. 9.5.1–9.5.9, Sec. 9.5.14; Sec. 10.1.1, 10.1.3, 10.2.4, 10.2.5, 10.2.8–10.2.10, 10.3.2, Sec. 10.4.2, Table 20; Sec. 10.4.3, 10.4.6, 10.4.7, Table 21; Sec. 10.4.9; Sec. 11.4.1; Sec. 12.2.1–12.2.8	Table 10; Sec. 7.3.11; Sec. 8.2.1, 8.2.3, 8.4.1, Sec. 8.5.1–8.5.9; Sec. 8.5.18; Sec. 9.1.1, 9.1.3, Sec. 9.2.2, 9.2.4, 9.2.5, 9.2.8–9.2.10, Sec. 9.3.2, Sec. 9.4.2, Table 22; Sec. 9.4.3, 9.4.6, Sec. 9.4.7, Table 23; Sec. 9.4.9; Sec. 10.4.1; Sec. 11.2.1–11.2.9	Table 1.4.4; Sec. 1.5.2.1, 1.5.2.3, 1.5.4.1; Sec. 1.5.5.1–1.5.5.9; Sec. 1.5.5.14; Sec. 1.6.1.1, 1.6.1.3, Sec. 1.6.2.2, 1.6.2.4, 1.6.2.5, 1.6.2.8–1.6.2.10; Sec. 1.6.3.2 Sec. 1.6.4.2, Table 1.6.3; Sec. 1.6.4.5, 1.6.4.7, 1.6.4.8, Table 1.6.4; Sec. 1.9.4.1; Sec. 1.10.2.1–1.10.2.9	Table 8.4; Sec. 9.2.1, 9.2.3, Sec. 9.4.1; Sec. 9.5.1–9.5.9; Sec. 9.5.14; Sec. 10.1.1, 10.1.3; Sec. 10.2.2, 10.2.4, 10.2.5, 10.2.8–10.2.10; Sec. 10.3.2, Sec. 10.4.2, Table 10.3; Sec. 10.4.5, 10.4.7, 10.4.8, Table 10.4; Sec. 13.4.1; Sec. 14.2.1–14.2.9	Sec. 9.1, 9.2, 9.3, 9.4, 9.5 – 9.7, 9.11, 9.11; Sec. 9.15, 9.16, 9.17, 9.18; Sec. 9.20, 9.22, 9.23; Sec. 9.24, 9.27; Sec. 10.5–10.13
Local Stability	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18, Table 27*; Sec. 7.22*, 7.23*, Table 29*; Sec. 7.24, Table 30	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10, 7.23, 7.26, 7.27, 7.28, 7.32, 7.33, 7.34	Sec. 8.3.2, Table 8; Sec. 8.3.7, Table 9; Sec. 9.5.1–9.5.9; Sec. 9.5.14; Sec. 10.4.2, Table 20; Sec. 10.4.3, Sec. 10.4.7, Table 21;	Sec. 7.3.2, Table 9; Sec. 7.3.8, Table 10; Sec. 8.5.1–8.5.9; Sec. 8.5.18; Sec. 9.4.2, Table 22; Sec. 9.4.3, Sec. 9.4.7, Table 23; Sec. 11.2.2	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.1–1.5.5.9; Sec. 1.5.5.14; Sec. 1.6.4.2, Table 1.6.3; Sec. 1.6.4.8, Table 1.6.4	Sec. 8.3.2, Table 8.3; Sec. 8.3.7, Table 8.4; Sec. 9.5.1–9.5.9; Sec. 9.5.14; Sec. 10.4.2, Table 10.3; Sec. 10.4.8, Table 10.4; Sec. 14.2.2	Sec. 9.1, 9.2, 9.3, 9.4, 9.5 – 9.7, 9.10, 9.11; Sec. 9.15, 9.16, 9.17, 9.18; Sec. 9.22, 9.23, 9.24
Base Plates	Sec. 5.12	Sec. 5.12	Sec. 9.6.2	Sec. 8.6.2	Sec. 1.10.1.1-1.10.1.3, 1.10.2.1-1.10.2.9	Sec. 14.1.1-14.1.3, 14.2.1-14.2.9	Sec. 7.12
Envelopes	Sec. 1.10–1.12 SNiP 2.01.07-85*	Sec. 1.10–1.12 SNiP 2.01.07-85*	Sec. 6.2–6.5 SP 20.13330.2011	Sec. 6.2–6.5 SP 20.13330.2011			Sec. 1.10–1.12 KMK 2.01.07 -96
Materials for	Sec. 2.2*, 3.4	Sec. 2.2, 3.4	Sec. 6.4, Annex D, Table D1,	Sec. 5.4, Annex D, Table D1,	Annex G	Annex E	Sec. 5.7, Table D.5

Mode	References to sections of standards or codes						
	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Welding			D2	D2			
Sheet Structures	Sec. 8.1-8.3, 8.5–8.12	Sec. 8.1-8.3, 8.5–8.12	Sec. 12.1.1–12.1.3, 12.2.1, 12.2.3–12.2.7	Sec. 11.1.1–11.1.3, 11.2.1, 11.2.3–11.2.7	Sec. 1.10.1.1-1.10.1.3, 1.10.2.1-1.10.2.9	Sec. 14.1.1-14.1.3, 14.2.1-14.2.9	Sec. 10.1-10.3, 10.5–10.12
Castellated Beam	Sec. 7.12, 7.18*, 11.5, Sec. 19.2, 19.3, 19.5, 19.6	Sec. 7.12, 7.28, 11.5 Sec. 19.2, 19.3, 19.5, 19.6	Sec. 8.3.2, 9.5.13, 15.1.19 Sec. L.5.2, L.5.3, L.5.5, L.5.6	Sec. 7.3.2, 8.5.17, 14.1.19, Sec. M.5.2, M.5.3, M.5.5, M.5.6 SP 16.13330.2011, Sec. 7.3.2, 8.5.17, 14.1.19 SP 16.13330.2017, Sec. 20.5.2, 20.5.3, 20.5.5 SP 294.1325800.2017	–	Sec. 9.5.13, 16.1.19, Sec. 23.2, 23.3, 23.5, 23.6	Sec. 9.13, 9.18, 13.5, Sec. 21.2, 21.3, 21.5, 21.6

4.11.2 On Formula (49) from SNiP II-23-81*

Formula (49) is totally acceptable in application to rectangular cross-sections with a characteristic angular point A (Fig. 4.11.2-1), where conditional stresses $M_x/(c_x W_{xn,\min} R_y \gamma_c)$ and $M_y/(c_y W_{yn,\min} R_y \gamma_c)$ are summed.

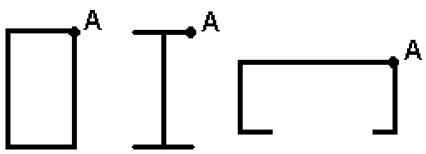


Figure 4.11.2-1. Cross-sections with an angular point

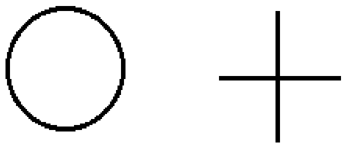


Figure 4.11.2-2. Cross-sections without angular points

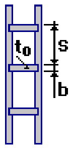
It is impossible to sum the stresses in sections without such an angular point (Fig. 4.11.2-2). Formula (50) of SNiP II-23-81* is better in this sense. It allows to choose between coordinates x and y of various cross-section points.

This problem exists in SP and DBN as well.

The said difference between the considered cases was noticed by L.B. Katznelson (Central Res. Inst. for Build. Structures). Authors of the application are very grateful to him for initiating a discussion of this and many other details of the design code implementation.

4.11.3 On Formula (14) from SNiP II-23-81*

A book by S.P. Timoshenko [8, c. 257–269] studies the approximate formulas like (14) in SNiP II-23-81*. These studies have shown that the formulas of such type have limited applicability. Unfortunately, SNiP II-23-81* does not provide any recommendations on the practical use of Formula (14). As a result, its formal application entails the following paradox. The stability is ensured for a column under central compression, the cross-section of which is an I-beam, while a column the cross-section of which consists of double I-beams with a lattice like



loses its stability at certain relationships between the lattice sizes. To avoid such a mishap, **Kristall** takes the effective height of the chord as the least of two values: a value obtained by Formula (14) and the actual height of the lattice column.

This problem exists in SP and DBN as well.

4.11.4 Analysis of Custom Sections

SNiP, SP and DBN regulate the checks only for a limited set of cross-section shapes. However, the practice of structural design often requires using the section shapes not defined by SNiP II-23-81*. This situation can be handled with applications included in the **SCAD Office** package in one of the following ways:

Create a file with the required section shape using Section Builder, Consul or TONUS (see [1]). Use the Resistance of Sections mode (see Section 4.9.1) of Kristall to analyze the behavior of the created section. It should be noted that SNiP, SP and DBN do not provide any recommendations for the determination of some parameters for the analysis of custom sections (section shape factor η, coefficients α and β according to Table 10, etc.). Therefore, the analysis is conservative and uses the most unfavorable values of these parameters. The check for the stability of in-plane bending is not performed at all due to the assumption that the possibility for this mode of buckling to occur is excluded by the appropriate restraints. SNiP, DBN and other similar design codes regulate the analysis of sections under shear forces according to the formulas that are difficult to adapt to the case of an arbitrary section. Therefore, when analyzing the strength of an arbitrary section under shear forces, the software uses the Eurocode approach, which compares the average tangential stresses, calculated as the ratio of the shear force to the shear area, and the design shear strength.

1. In this case the problem of selecting the design strength of steel $\mathbf{R_y}$, the value of which depends on the thickness of the section, appears when checking the sections. The value of the design strength $\mathbf{R_y}$ will be taken according to the following rules for arbitrary sections with an explicitly specified steel grade:

- for sections created by **Consul** — the minimum design strength value of the steel of the given grade;
- for sections created by **Tonus** — design strength value of the thickest steel strip (if this thickness is not found in the respective table of the design code, the minimum design strength value of the steel of the given grade will be taken);
- for sections created by **Section Builder** — design strength value of the thickest steel plate or rolled profile (if this thickness is not found in the respective table of the design code, the minimum design strength value of the steel of the given grade will be taken); if the section is made from structural steel, the data from the codes for structural steel will be used, otherwise — the data from the codes for sheet and plate steel.

2. Use **SEZAM** (see [1]) to find an equivalent section and replace a nonstandard cross-section with a similar standard one (for example, with an I-beam or a hollow section).

4.11.5 On Seismic Actions

The concept of “seismic load” is absent from some of the **Kristall** modes (*Columns*, *Beams*, *Continuous Beams*, *Trusses*). This is due to the fact that special information on the logical relations between the loadings (which is actually realized in the DCF mode of SCAD) has to be specified to take seismicity into account correctly.

It should be noted that Section 2.14 of SNiP II-7-81* *Construction In Seismic Regions* requires using an additional service factor which is greater than one (see Table 7 of SNiP II-7-81*). When a structure is being analyzed for simultaneous action of multiple loads (which is nearly always the case for building structures), one of which is seismic, SNiP formally requires to use this factor even in cases when the fraction of the seismic action is small in comparison with other (long-term) loads. A requirement like this may lead to “dangerous results”.

The user can always take seismicity into account by, for example, setting the service factor equal to 1.4 (according to Table 7 of SNiP II-7-81*) in the **Kristall** settings. Without these “auxiliary” operations the analysis will be conservative.

4.11.6 Accounting for the Constrained Torsion

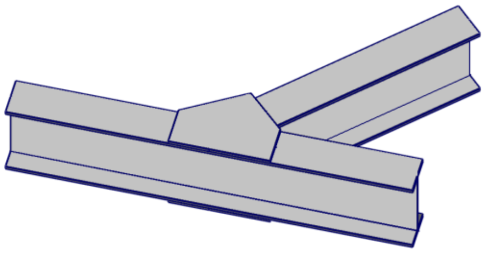
With the release of the **SCAD Office 21.1**, questions concerning the implementation of the SP 16.13330 sections on the accounting for the restrained torsion have arisen in many forums on the Internet. Here is one of them:

> In the result of analysis (for example, of bars) the following values are still calculated: N, M_z, M_y, $M_к$, Q_z, Q_y.
> What about accounting for the bimoment for any occurring restraints in the spatial structure and any elements? How will you implement the requirement, for example, of Sec. 8.2.1 of SP 16 to consider the additional stresses at the constrained torsion?
> It does not say that this formula applies only to thin-walled bars. SP16 is not yet implemented for any elements and types of cross-sections.
> But most importantly, it is simply impossible.

The author of the question has rightly observed, "... most importantly, it is simply impossible".

Indeed, as follows from the technical theory of thin-walled bars, a generalized displacement in the form of warping (loss of flatness) of the bar cross-section corresponds to the bimoment *B*. For a system composed of thin-walled bars, when several elements oriented in different ways in space are connected in a joint, you need to be able to record the compatibility conditions of warping or the equilibrium conditions of bimoments determined in the end sections of the bars. Such conditions can easily be formulated only in the case of a joint of two elements lying on one line, and only if their cross-sections are identical (continuous thin-walled beam). In the general case a detailed analysis of the joint structure is required and it goes beyond the standard analysis of the bar system.

Since the bimoment is an action the static equivalent of which is equal to zero, the equilibrium conditions in the joint will be satisfied at arbitrary values of the bimoments in the end sections of the bars. How to compare the values of warping in the end sections remains unclear. Let’s consider one of the simplest joint (see figure). I would be interesting to find out what warping conditions will be listed here.



The procedure of introducing the bimoment into SP 16.13330 is of particular interest.

For the bars, working in an elastic stage (sections of the 1st class) a strength check according to the following formula (43) is suggested:

$$\frac{M_x y}{I_{xn} R_y \gamma_c} \pm \frac{M_x y}{I_{yn} R_y \gamma_c} \pm \frac{B\omega}{I_\omega R_y \gamma_c} \le 1.$$

The authors did not bother to indicate what is denoted by I_ω and ω. Obviously, these are the sectorial moment of inertia and the sectorial area. The latter depends on the coordinates of the considered point, which that should have been mentioned. However, if the particular point of the cross-section with *x* and *y* coordinates is considered at a particular loading which has caused the internal forces M_x, M_y, B, what do the symbols ± mean in this formula?

It is recommended to check the stability of I-beams of the 1st class according to the following formula:

$$\frac{M_x}{\varphi_b W_{xc} R_y \gamma_c} + \frac{M_x}{W_y R_y \gamma_c} + \frac{B}{W_\omega R_y \gamma_c} \le 1,$$

where the value of sectorial moment of resistance W_ω is used without indicating the section point for which it is calculated. It seems that the following formula should be used here $W_\omega = I_{\omega n} / \omega(x, y)$, doesn't it?

Even more questions arise when looking at the formula (105), which is suggested for checking the strength of the elements reaching the limit state in the elastic-plastic stage,

$$\left(\frac{|N|}{A_n R_y \gamma_c} \right)^n + \frac{|M_x|}{c_x W_{xn,\min} R_y \gamma_c} + \frac{|M_x|}{c_x W_{xn,\min} R_y \gamma_c} + \frac{|B|}{W_{\omega n,\min} R_y \gamma_c} \le 1,$$

or at formula (106) for the elements which reach the limit state not at the yield point

$$\left(N/A_n \pm M_x y/I_{xn} \pm M_x y/I_{yn} \pm B\omega/I_{\omega n} \right) / \left(R_y \gamma_c \right) \le 1.$$

The very structure of the formula (105), which determines the surface of interaction between separately calculated ultimate values of the section resistance $A_n R_y \gamma_c$, $c_x W_{xn.\min} R_y \gamma_c$, $c_y W_{yn.\min} R_y \gamma_c$ and $W_{\omega n,\min} R_y \gamma_c$, raises doubts. The fact that the section works in the elastic-plastic stage is taken into account by introducing the increasing factors c_x and c_y to the values of the bending section modulus. However this is not done in relation to the warping section modulus. It turns out that the ultimate value of the resistance to the bimoment at an elastic-plastic constrained torsion is equal to its elastic value. Well-known researches of A.I. Strelbitskaya have shown that it is not true.

Finally, the bimoments arise at a constrained torsion together with the moments of constrained torsion. However, the codes do not state that it is necessary to check a cross-section for torsion (both non-constrained and constrained). What is the cause of such an inconsistency remains unclear.

4.12 References

[1] V.Karpilovsky, E.Kriksunov, A.Perelmuter, M.Perelmuter. Creation of Sections and Calculation of Their Geometric Properties. — Kiev, "Compass", 2001.— 96 p.

[2] A.I. Kikin, A.A. Vasiliev, B.N. Koshutin, V.Y.Uvarov, Y.L.Volberg. Increase of Metal Structures Durability for Industrial Buildings. — Moscow, "Stroyizdat", 1984.

[3] V.P.Koroliov. Theoretical principles of engineering analysis of steel structures for corrosion resistance and durability: Res. papers / Donetsk State Academy for Civil Engineering and Architecture.— Issue 1-95. Donetsk, 1995.— 108 p.

[4] A.V.Perelmuter, L.A.Gildengorn. On classification of steel structures / Structural mechanics and analysis. —1990. —N3. — pp. 67–70.

[5] Guide on designing of residential buildings.— Issue 3. Designs of residential buildings (to SNiP 2.08.01-85) / Centr. Resrch. Inst. for Resident. Eng., State Committee for Architecture. Moscow, "Stroyizdat", 1989. 304 p.

[6] Guide to design of reinforcing for steel structures (to SNiP II-23-81*).— Moscow, "Stroyizdat", 1989.

[7] Guide to design of steel structures (to SNiP II-23-81*) / Kucherenko Centr. Res. Inst. for Building Structures, USSR State Committee for Construction, Moscow, 1989. – 148 p.
[8] S.P.Timoshenko. Stability of bars, plates, and shells.— Moscow, "Nauka".— 1971.— 808 p.
[9] E.V. Gorochov, V.P.Korolev, A.A. Balkena / Durability of Steel Structures Under Reconstruction.— Rotterdam: Brookfield, 1999.— 305 p.

5. Kristall-Eurocode 3

Kristall-Eurocode 3 enables to perform checks of members and joints of steel structures for compliance with requirements of Eurocode 3. It can be useful for experts who develop design documentation ordered by European companies. *EN 1993-1-1. Eurocode 3: Design of Steel Structures. — Part 1.1: General rules and rules for buildings* and *EN 1993-1-5. Eurocode 3: Design of Steel Structures. — Part 1.5: Plated structural elements* define checks of members of steel structures, and *EN 1993-1-8. Eurocode 3: Design of Steel Structures. — Part 1.8: Design of joints* is used for the checks of joints of steel structures.

Eurocode 3 is supplemented by the national application documents (NAD) which at some points clarify the general provisions of EN 1993-1-1, EN 1993-1-5 and EN 1993-1-8. All the design national peculiarities defined by the national bodies in charge of safety are taken into account in the application.

5.1 Main Window

The main window of the application (Fig. 5.1-1) contains buttons for selecting a working mode. The **Beams, Columns** and **Bracing** modes enable to perform a complex check of particular structural members of steel structures, and the **Trusses** mode enables to perform all the necessary strength and stability checks of the truss members for compliance with all the code requirements for the truss structure, beginning with creating the design load combinations.

Figure 5.1-1. *The main window*

The **Resistance of Sections**, **Bolted Connections**, and **Welded Connections** modes implement particular operations for the determination of the resistances of cross-sections of bar elements of steel structures and their joints, as defined by EN 1993-1-1, EN 1993-1-5 and EN 1993-1-8.

The auxiliary modes **Envelopes**, **Critical Moment, Effective Lengths,** and **Geometric Properties** implement some frequently used stages of analysis which may be of particular interest.

The information modes **Steel**, **Assortment of Rolled Profiles**, and **Bolts** are used to browse through the grades of steel and the corresponding catalogues of rolled steel and pipes, assortments of rolled profiles, and properties of bolts (mechanical properties, proof loads, and minimum ultimate tensile loads).

When you invoke any of these modes, a multi-tab dialog box appears, which contains the controls common for all working modes. These include the **File**, **Mode**, **Settings**, **Tools**, and **Help** menus. The **Help** and **Exit** buttons perform functions common for a Windows application: providing reference information and finishing the current session respectively. Other buttons are described below. The **Menu** button switches from any mode to the main window. Design codes can be selected from the respective list. Information on the selected code is displayed in the bottom left corner of the active mode window.

5.2 Settings

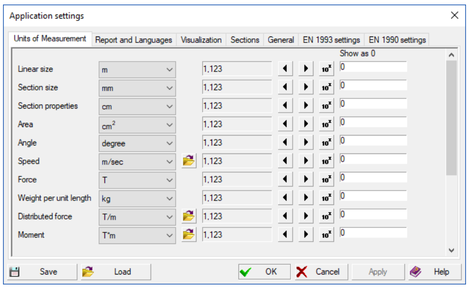
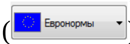


Figure 5.2-1. *The* **Application Settings** *dialog box*

The **Application Settings** dialog box (Fig. 5.2-1) contains the following tabs: **Units of Measurement**, **Report and Languages**, **Visualization**, **Sections**, **General**, and two more tabs **EN 1993 Settings** and **EN 1990 Settings**. The first five tabs are similar to those of **Kristall** (SNiP) described above.

The **EN 1990 Settings** tab (Fig. 5.2-3) enables to select loading combination factors in compliance with EN 1990.

The **EN 1993 Settings** tab (Fig. 5.2-2) enables to select the partial safety factors in compliance with EN 1993-1-1 or in compliance with the National Annexes to Eurocode (NAD). Moreover, this tab enables to select the method for checking the general stability of the elements under bending and compression (select the respective Annex A or B of EN 1993-1-1), limit the execution of standard checks assuming that the deformations of steel are elastic, specify the parameter ρ_{lim} allowing for the effect of buckling of the section elements on the rigidity of the structure as a whole, select the method for allowing for shear lag for the ultimate limit state, specify the parameters η for checking the shear stability of webs, and to define the set of the used classes of bolts (see Fig. 5.2-2).

If the values of the partial safety factors and other standard parameters of the calculation are specified in compliance with the recommendations of EN 1993-1-1, then the row with the EU flag (), which is selected by default, should be selected from the drop-down list. If you want to use the National Annexes, select the row with a flag of the respective state. The numerical values of the respective partial safety factors and the parameters of the standard calculation will be displayed in the table in the **EN 1990 Settings** tab.

If the values of the partial safety factors or the parameters of the standard calculation are specified by the user, then you have to select the **Other** item in the drop-down list. Thus you can use the application for custom, nonstandard situations, and also when any modifications are made in NAD.

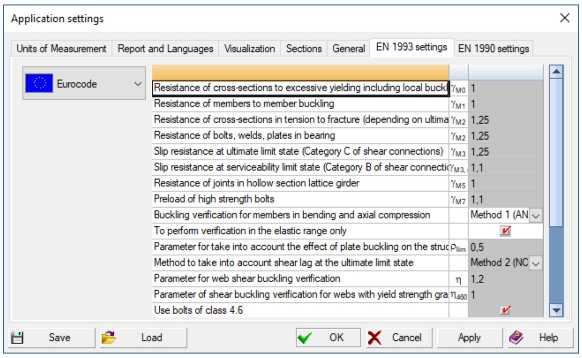


Figure 5.2-2. *The* **EN 1993 Settings** *tab of the* **Application Settings** *dialog box*

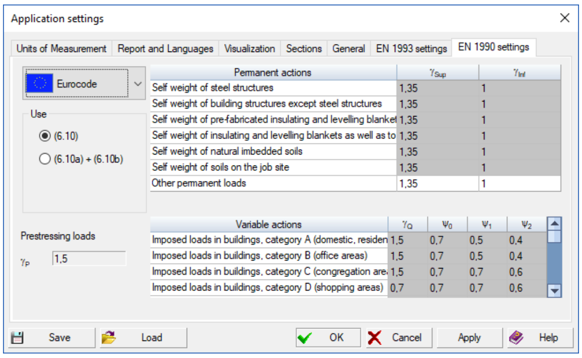
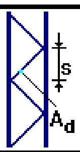
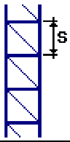
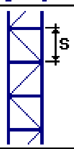
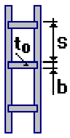


Figure 5.2-3. *The* **EN 1990 Settings** *tab of the* **Application Settings** *dialog box*

5.3 Creating Cross-Sections

The actions of selection of a cross-section for structural members are similar to those described in Section 4.3. The only difference is a different set of connecting lattices for compound sections. A list of the connecting lattices and limitations of their sizes are given in Table 5.3-1.

Table 5.3-1. Limitations of lattice sizes

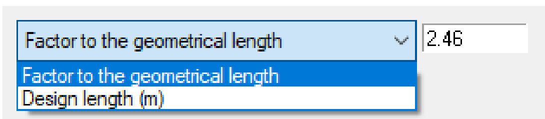
Design	Limitations	Design	Limitations
	$s \geq 2h$ $A_d < A$		$s \geq h$ $A_v < A$ $A_d < A$
	$s \geq h$ $A_d < A$		$s \geq b$

Notes:

h is a distance between the axes of chords;
A is a chord area;
A_d is a diagonal area;
A_v is a vertical area.

5.4 Specifying the Effective Lengths

It is necessary to specify the data on the effective lengths of structural members in some modes. In many cases you can specify it either in the form of the effective length factor or the effective length, which can be selected from the drop-down list next to the text field:



5.5 Fire Resistance

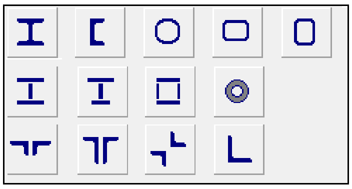
The fire resistance analysis of individual unprotected elements of steel structures is implemented in **Kristall**. The check is performed by the critical temperature method according to EN 1993-1-2:2005. The method assumes that when the structure is heated by a fire, the temperature of the steel is uniformly distributed throughout the structure and its critical value is determined by the reserve of strength of the considered element. This reserve of strength is characterized by the load-bearing capacity utilization factor $\mu_{0,}$ the physical meaning of which corresponds to the coefficient K_{max}. Taking into account μ_0 the critical temperature is calculated by the following formula:

$$\theta_{a,cr} = 30,19\ln\left(\frac{1}{0,967\mu_0^{3,833}} - 1\right) + 482.$$

The program determines the value of the load-bearing capacity utilization factor K_{max} from the combination of loads which includes only the values of constant and long-term loads at $t = 0$ (quasi-permanent combination).

Quasi-permanent combinations are calculated automatically in the *Columns* and *Beams* modes. The user has to check the **Fire** checkbox in the *Resistance of Sections* mode for the loadings corresponding to the quasi-permanent combinations. When importing data from the files of type rsu2 the **Fire** checkboxes will be checked automatically.

The fire resistance analysis is implemented in the following modes: *Resistance of Sections, Columns, Beams* for the following types of sections:



The critical temperature, time to reach this temperature, and the air and steel temperature-time curves are obtained in the result of the analysis. Moreover, the software outputs the reduced thickness (the ratio of the cross-sectional area to the heated perimeter) and the proper fire resistance limit (the time it takes to reach the temperature of 500°C, which causes significant change in strength and deformability — see Table 6 of the *Manual for determining the limits of fire resistance of building structures, fire hazard parameters of materials. Procedure for the design of fire protection.* M. 2013).

The results are output in the **Fire Resistance** tab (Fig. 5.5-1)

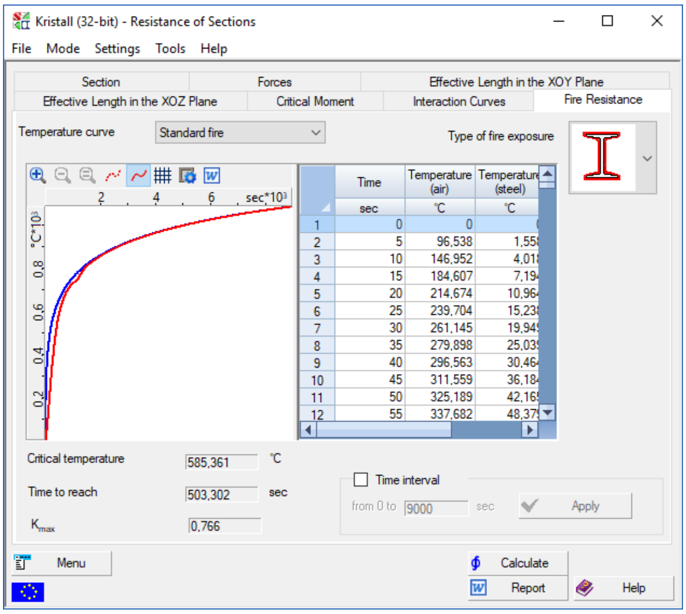


Figure 5.5-1. *The* **Fire Resistance** *tab*

The analysis is performed for one of the following types of the structural heating (Fig. 5.5-2):

- standard fire;
- hydrocarbon fire;
- external fire;
- smoldering fire.

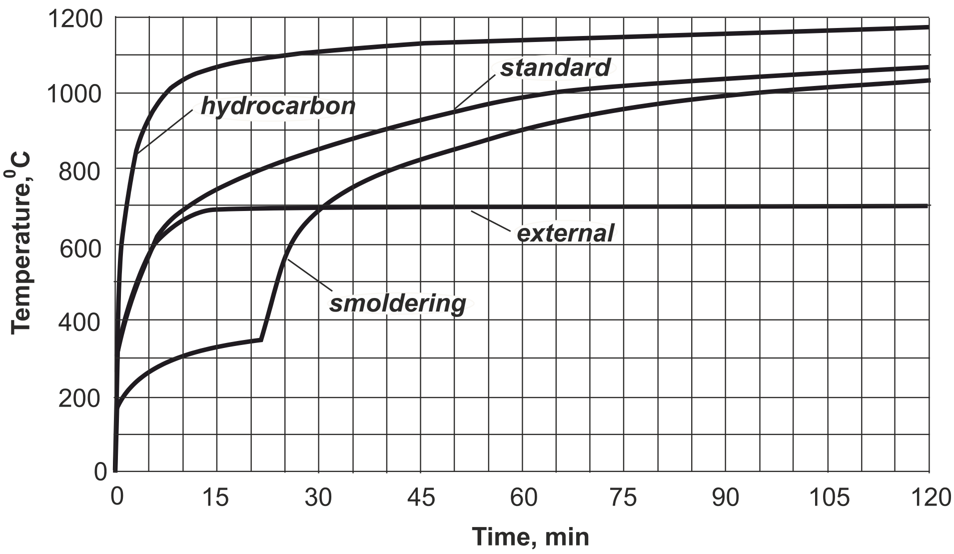


Figure 5.5-2. *Temperature-time curves*

The type of fire is selected from the respective drop-down list in the **Fire Resistance** tab. The analysis can be performed for the following types of fire exposure:

- heating from four sides for all cross-sections;
- heating from three sides for all sections except for angles and hollow sections.

The temperature increase of steel at heating is calculated taking into account the change in the thermal conductivity and the specific heat of steel (Fig. 5.5-3).

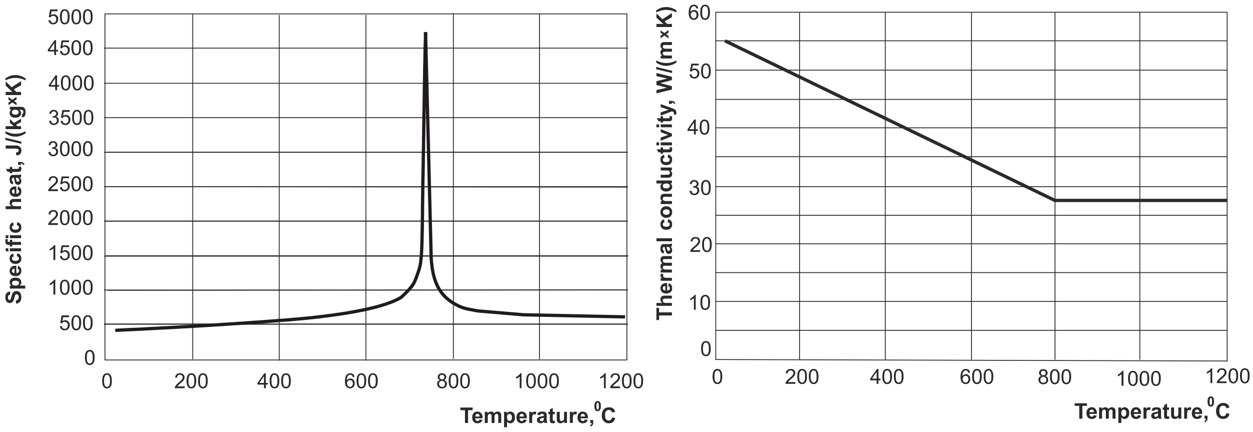


Figure 5.5-3. *Variation of the thermal properties of steel*

5.6 Addvanced Settings

In many modes there is a button  next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional restrictions for selecting steel sections (Fig. 5.6-1).

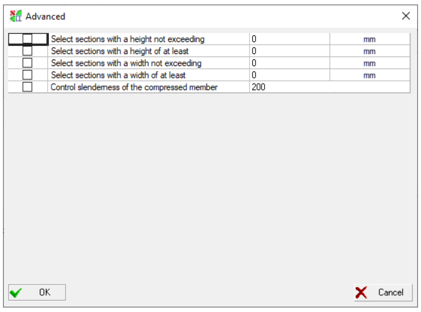


Figure 5.6-1. *The* **Additional Settings** *dialog box*

In particular, you can specify the restrictions on the height and width of a section.

Slenderness of compressed members

This marker is intended for the program to analyze the slenderness of compressed members. After activating it, you can specify the value of the slenderness limit (default is 200).

5.7 Reference Modes

5.7.1 Steel

The **Steel** mode can be used as a reference one to obtain data on steels of different grades and on the rolled steels and pipes made from them (in this case it is invoked from the main window), and also to assign the steel grade when checking the members of steel structures. In the latter case this mode is invoked from the design one.

A two-tab dialog box **Steel** (Fig. 5.7.1-1) is used in the reference mode. Its first tab **Structural Steels and Plates** contains information on steels used for structural steels and plates, and the second one **Pipes** (Fig. 5.7.1-2) contains information on steels used for pipes.

Each tab contains a list of steel grades and a table with reference data which includes the data on the thickness of the rolled steel or of the pipe walls, as well as the characteristic values of the yield strength (f_y) and the ultimate strength (f_u), corresponding to the steel grade selected from the list. A standard corresponding to the selected steel grade is indicated under the list of steel grades.

When the **Steel** mode is invoked from the design modes by clicking the CT button, a third tab, **User Defined Steel**, (Fig. 5.7.1-3), is added. It enables to specify the values of the yield and ultimate strength different from the standard ones.

To use the selected steel grade or the specified parameters of steel in the check of the load-bearing capacity, click the **Apply** button.

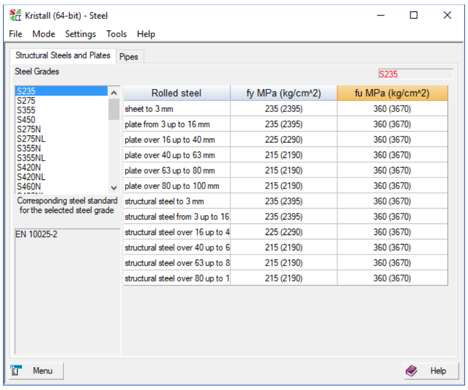


Figure 5.7.1-1. *The* **Structural Steels and Plates** *tab of the* **Steel** *dialog box*

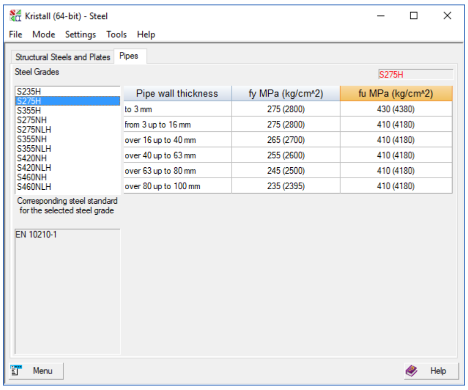


Figure 5.7.1-2. *The* **Pipes** *tab of the* **Steel** *dialog box*

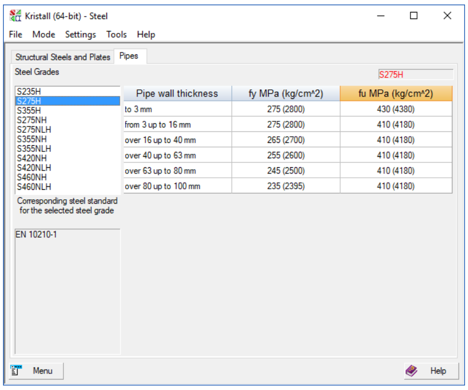


Figure 5.7.1-3. *The* **User Defined Steel** *tab of the* **Steel** *dialog box*

5.7.2 Assortment of Rolled Profiles

This mode enables you to browse through steel profile assortments available in the database of the application and is described in the Section 4.7.2.

5.7.3 Bolts

This information mode provides the reference data on the mechanical properties of bolts, and also on the proof loads, and minimum ultimate tensile loads according to EN ISO 898-1:1999-12 (see Fig. 5.7.3-1 and 5.7.3-2).

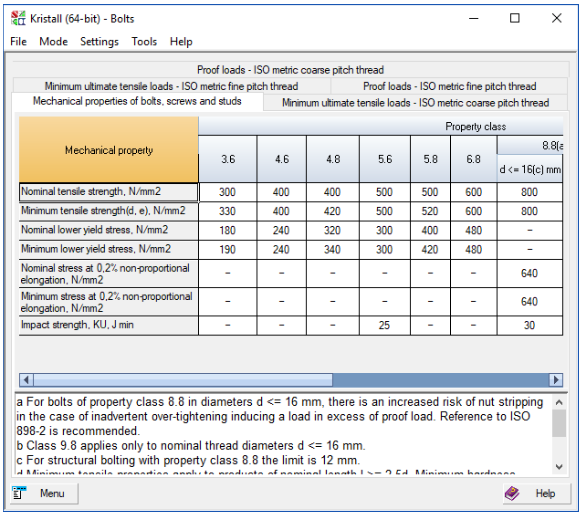


Figure 5.7.3-1. *The* **Mechanical Propertis of Bolts, Screws and Studs** *tab of the* **Bolts** *dialog box*

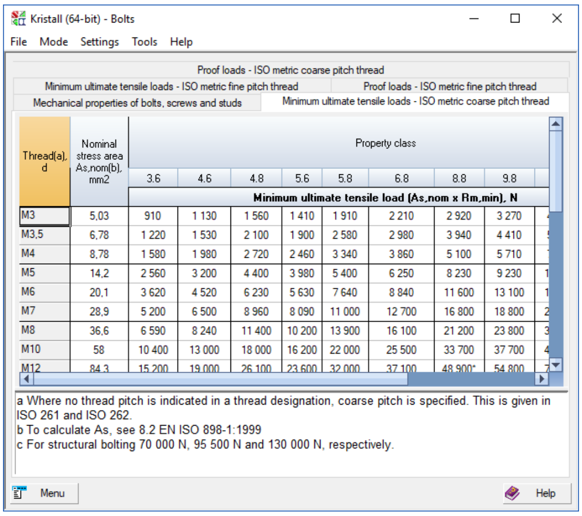


Figure 5.7.3-2. *The* **Minimum Ultimate Loads** *tab of the* **Bolts** *dialog box*

5.8 Auxiliary Modes

5.8.1 Resistance of Sections

The procedure of determining the load-bearing capacity of bar elements of steel structures is implemented in the **Resistance of Sections** mode. The types of cross-sections implemented in this mode are given in Fig. 5.8.1-1. The whole set of strength and stability checks according to the respective sections of EN 1993-1-1:2005 and EN 1993-1-5:2005 is implemented.

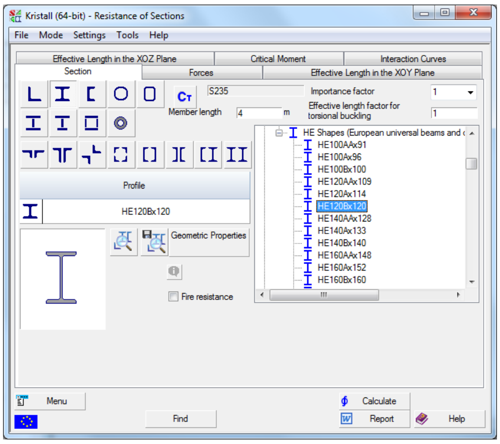


Figure 5.8.1-1. *The* **Section** *tab of the* **Resistance of Sections** *dialog box*

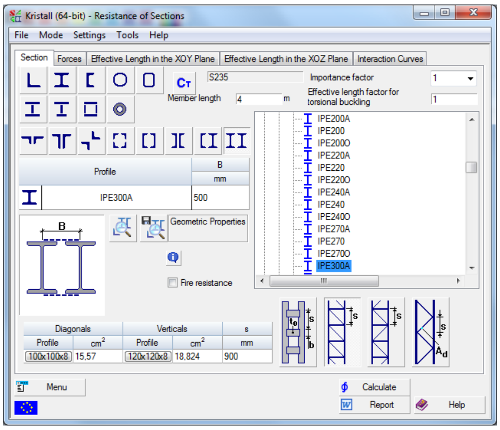


Figure 5.8.1-2. *The* **Section** *tab* for selecting properties of the connecting lattice

Moreover, this mode enables to perform the selection of sections from rolled profiles or compound sections from rolled profiles. The search problem has a restricted formulation as a focused search in the list of possible designs (list of the assortment of rolled steel profiles).

The dialog box contains six tabs: **Section, Forces, Effective Length in the XoY Plane, Effective Length in the XoZ Plane, Critical Moment, Interaction Curves**.

The **Section** tab (Fig. 5.8.1-1) contains a set of buttons for selecting a cross-section type. The section with dimensions is displayed for welded, compound, and lattice sections, and text fields for entering these dimensions appear. Units of measurement which should be used to specify the sizes of the section are also indicated here.

For lattice sections, the tab has buttons for selecting the type of a connecting lattice. Clicking one of them will display a detailed picture of the lattice with sizes the values for which are entered in the respective text fields (Fig. 5.8.1-2).

The **Geometric Properties** button is used to invoke a window that displays the values of the geometric properties calculated for the created section.

The selected section can be saved (with the **Save** button) in the **Custom Sections** directory which can be accessed by clicking the button .

This tab also contains the  button, which enables to select steel. The procedure of the steel selection is described in the Section 5.7.1.

The length of the structural member as well as design length factor for torsional buckling have to be also specified on the **Section** tab in the corresponded fields.

Moreover, the importance factor which will be further multiplied by the values of all internal forces for all design combinations of loadings acting in the considered section has to be specified in the **Importance factor** field. If the values of the internal forces for the section have been obtained based on the results of the analysis of the system accounting for the importance factor (for example, when the design values of the applied loads were obtained taking into account this factor), the value equal to one has to be selected in the **Importance factor** drop-down list.

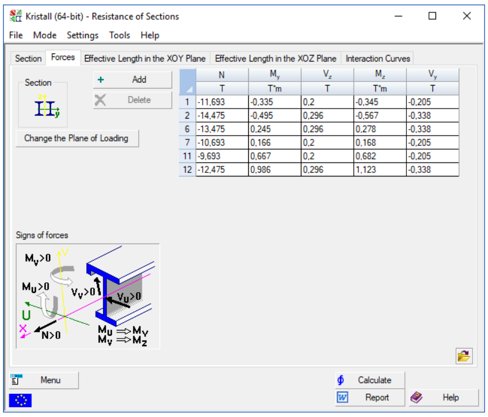


Figure 5.8.1-3. *The* **Forces** *tab of the* **Resistance of Sections** *dialog box*

The **Forces** tab (Fig. 5.8.1-3) is used to specify forces acting in the cross-section of the member. It displays a cross-section with the principal axes of inertia and the positive directions of forces.

Checks for the general case of loading, i.e. for the action of a longitudinal force, two bending moments, and two shear forces acting in the principal planes of inertia, are implemented for all the given types of cross-sections. The checks are performed for any number of loadings. The table of internal forces (each line of which corresponds to a certain loading) can be specified by importing the design combinations of forces from **SCAD** by clicking the button .

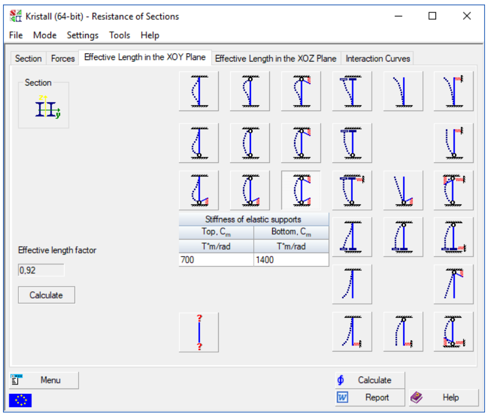


Figure 5.8.1-4. *The* **Effective Length** *tab of the* **Resistance of Sections** *dialog box*

The **Effective Length in the … Plane** tabs (Fig. 5.8.1-4) are intended to specify the design length factor for flexural buckling in the corresponded planes. They contain a group of buttons depicting design models of a compressed bar member different from one another in their combinations of boundary conditions (free end, hinge, elastic support, elastic clamping, clamped).

If the boundary conditions include an elastic clamping or an elastic support, a table for entering the data on the stiffness of the respective restraint will appear.

Factors $k = l / L$ describe the ratio of the effective length, l, to the geometric length of the bar, L**,** in different planes. They are displayed in the respective fields of results.

Clicking the button will enable you to enter any desired values for k and confirm your choice by clicking the **Apply** button. In all other cases this field is inaccessible.

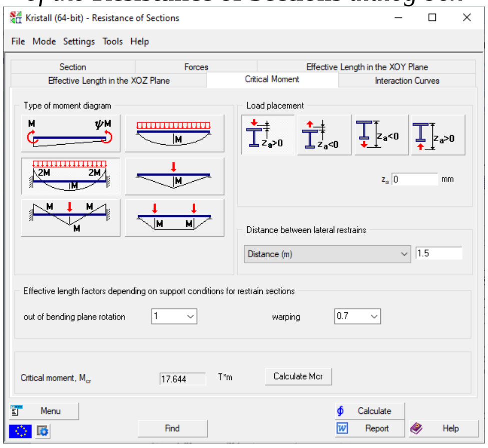


Figure 5.8.1-5. *The* **Critical Moment** *tab of the* **Resistance of Sections** *dialog box*

The **Critical Moment** tab (Fig. 5.8.1-5) enables to calculate the critical bending moment for beams which is further used in the check of the stability of in-plane bending. The calculation of the critical moment is performed using the analytical solution given in the Annex of the French NAD of EN 1993-1-1 depending on the character of the bending moment diagram, load application with respect to the shear center of the section, and also depending on the asymmetry of the section with respect to the axis of the greater stiffness. The calculations are performed only for the cross-sections symmetric with respect to the axis of the lower stiffness at their bending with respect to the axis of the greater stiffness. Moreover, EN 1993-1-1 limits the use of this check method for the sections which are characterized by a significant rotation of the principal axes of inertia of the "effective" section with respect to the principal axes of inertia of the gross cross-section.

The respective limitations are introduced in the software as well.

If a different cross-section type is selected, this tab will not appear at all, and it is assumed that the stability of in-plane bending mode is ensured by the appropriate restraints.

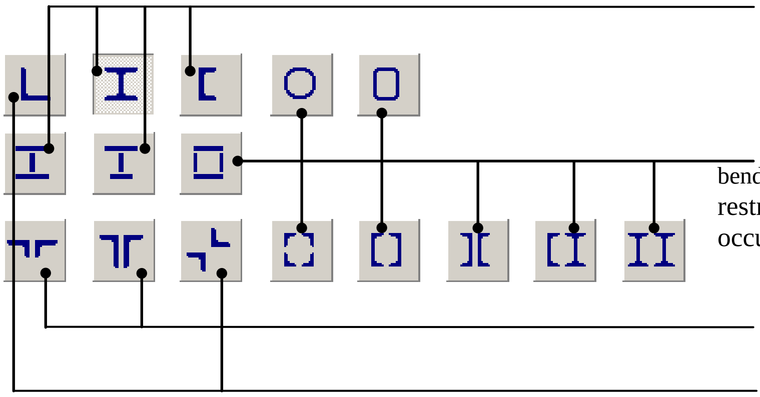
In addition, to calculate the critical moment, two effective length factors should be specified. The factor k refers to end rotation on plan and the factor k_w refers to end warping. Recommendations for assigning the coefficients k, k_w for some special cases can be found in the books

- L. Gardner, *Stability of Steel Beams and Columns: In Accordance with Eurocodes and the UK National Annexes*, Steel Construction Institute, 2011.
- N.Boissonnade, R.Greiner, J.P.Jaspart, J.Lindner, *Rules for member stability in EN 1993-1-1 : Background documentation and design guidelines*, ECCS European Convention for Constructional Steelwork, 2006.

To determine the critical moment, specify a moment diagram type by selecting it from six options given in the Annex of the French NAD of EN 1993-1-1. If the first option is selected, you should also specify the parameter ψ which characterizes the ratio of the end moments. The design length factor for lateral-torsional buckling has to be specified in the **l/L for a compressed flange** field. Moreover, the distance from the load application point to the top chord level, z_a, should be specified. The sign of z_a is assigned according to a schematic given in this tab.

Clicking the **Calculate M$_{cr}$** button will perform the calculation of the critical moment, M_{cr}.

Further actions depend on the section type according to a schematic shown in Fig. 5.8.1-6 containing information on how to consider M_{cr}, in sections of various types.



Calculation is performed

It is assumed that the stability of in-plane bending is provided by the respective restraints, or the in-plane buckling does not occur

Calculation is performed

For these sections, EN 1993-1-1 does not provide any recommendations on the determination of the critical moment

Figure 5.8.1-6. A response of the **Critical Moment** mode to the section type

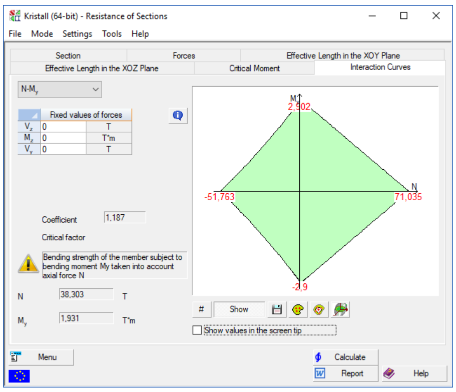


Figure 5.8.1-7. *The* **Interaction Curves** *tab of the* **Resistance of Sections** *dialog box*

Clicking the **Calculate** button will output the result of the calculation – K_{max} value (maximum of the detected utilization factors of restrictions). The message about the type of the check (strength, stability, local stability, etc.) in which this maximum took place will appear in the **Critical factor** field. The **Selection** button enables to perform an automatic selection of the section implemented as a focused search in the list of the assortment of rolled steel profiles.

The **Interaction Curves** tab (Fig. 5.8.1-7) enables to plot the curves enclosing an area of the section load-bearing capacity under various pairs of forces which can arise in the considered section.

To plot the interaction curves, select a pair of forces from the drop-down list and click the **Show** button (for details see Section 4.9.1).

The set of checks in accordance with EN 1993-1-1 is defined by the type of internal forces acting in the design section of the element, type of cross-section of the element, and the class of the section. The following classes of sections are implemented in the **Resistance of Sections** mode:

- 2 class of sections – steel sections with elastic-plastic behavior;
- 3 class of sections – steel sections with elastic behavior, the section elements do not lose local stability;
- 4 class of sections – sections the elements of which lose local stability.

One of the characteristic differences in the calculations of the load-bearing elements of steel structures according to EN 1993-1-1 is the possibility of their operation after the local buckling of the compressed elements of the section. When calculating the geometric properties of cross-sections in the **Resistance of Sections** mode the concept of the "effective width" is used, which is implemented in the form of the Winter's equation with the respective modifications related to the consideration of the gradient of stresses in the compressed elements of the section, and the bearing conditions according to EN 1993-1-5.

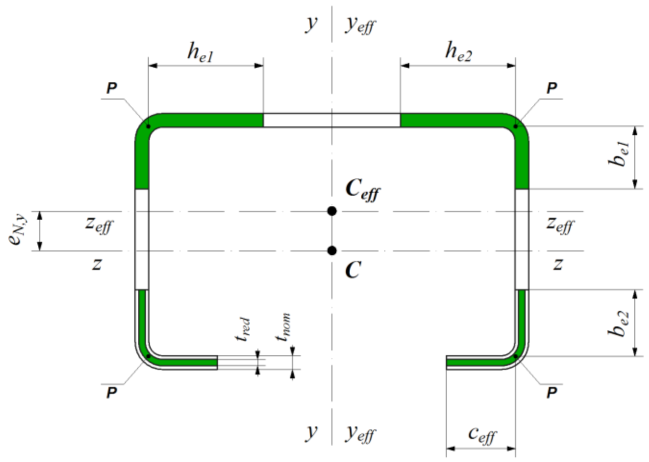


Figure 5.8.1-8. "Effective" cross-section

The general algorithm of the standard calculation of the bar structural elements includes the calculation of the "effective" width of the compressed section elements accounting for the local buckling (Fig. 5.8.1-8). If the position of the center of mass of the effective cross-section does not coincide with the position of the center of mass of the gross cross-section, the displacement of the center of mass is taken into account in the subsequent checks. This displacement is taken into account by increasing the bending moments acting in the considered design section.

The list of checks performed in the **Resistance of Sections** mode is given in Table 5.8.1-1. This table also lists sections of EN 1993-1-1 and EN 1993-1-5 which define these checks (a full list of results of the checks is given in the report document).

Table 5.8.1-1. A list of checks of sections for compliance with EN 1993-1-1 and EN 1993-1-5

Factor	EN 1993-1-1	EN 1993-1-5
Strength under action of longitudinal tensile force N	6.2.3 (6.5), (6.6)	
Strength under action of longitudinal compressive force N	6.2.4 (6.9)…(6.11)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of bending moment M_y	6.2.5 (6.12)…(6.15)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of bending moment M_z	6.2.5 (6.12)…(6.15)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of lateral force V_z	6.2.6 (6.17)…(6.20)	
Strength under action of lateral force V_y	6.2.6 (6.17)…(6.20)	
Strength under combined action of bending moment M_z and lateral force V_y	6.2.5 (6.12)…(6.15), 6.2.8 (6.29), (6.30)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under combined action of bending moment M_y and lateral force V_z	6.2.5 (6.12)…(6.15), 6.2.8 (6.29), (6.30)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under combined action of bending moment M_y and longitudinal force N	6.2.5 (6.12)…(6.15), 6.2.9 (6.36)…(6.44)	4.4 (4.1)...(4.4), Table 4.1, 4.2

Factor	EN 1993-1-1	EN 1993-1-5
Strength under combined action of bending moments M_y, M_z, longitudinal force N and lateral forces V_z and V_y	6.2.5 (6.12)…(6.15), 6.2.9 (6.36)…(6.44), 6.2.10 (6.45)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Stability at flexural buckling about the y-y axis under action of longitudinal compressive force N	6.3.1 (6.46)…(6.51)	
Stability at flexural buckling about the z-z axis under action of longitudinal compressive force N	6.3.1 (6.46)…(6.51)	
Stability at torsional-flexural buckling under action of longitudinal compressive force N	6.3.1 (6.46)…(6.49), 6.3.1 (6.52), (6.53)	
Stability of in-plane bending under action of bending moment M_y	6.3.2 (6.54)...(6.56)	
Stability of in-plane bending under action of bending moment M_z	6.3.2 (6.54)...(6.56)	
Stability under combined action of longitudinal force and bending moments	6.3.3 (6.61), (6.62)	
Stability under combined action of longitudinal force and bending moments taking into account in-plane bending	6.3.3 (6.61), (6.62)	
Stability of the web under action of lateral force V_z		5.2 (5.1), (5.2), 5.3 (5.3)...(5.6), 5.4 (5.8), (5.9), 5.5 (5.10)
Stability of the web under action of lateral force V_y		5.2 (5.1), (5.2), 5.3 (5.3)...(5.6), 5.4 (5.8), (5.9), 5.5 (5.10)

Only the cross-section of a member is checked.

The **Resistance of Sections** mode does not provide the following checks:

- weakened sections with bolt holes;
- members of connecting lattices in the lattice cross-sections;
- stability of webs reinforced by stiffeners for I-beam, channel, and hollow section members.

5.8.2 Bolted Connections

The **Bolted Connections** mode enables to determine the load-bearing capacity of bolted connections. Structural designs of bolted connections implemented in this mode are the most common ones in the engineering practice.

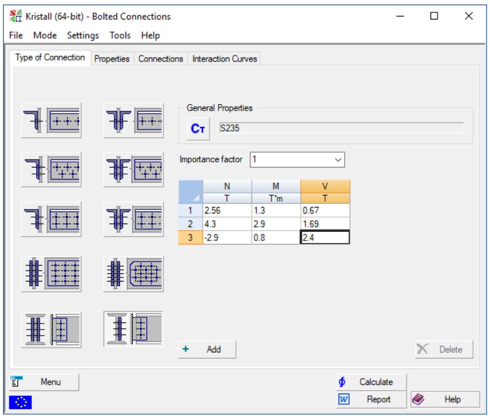


Figure 5.8.2-1. *The* **Type of Connection** *tab of the* **Bolted Connections** *dialog box*

The **Type of Conncetion** tab (Fig. 5.8.2 -1) contains a group of buttons for selecting a type of the bolted connection, and a table for entering data on the forces acting in it.

This tab also contains the  button, which enables to select steel. The procedure of the steel selection is described in the Section 5.7.1.

Moreover, the importance factor which will be further multiplied by the values of all internal forces for all design combinations of loadings acting in the connection has to be specified in the **Importance factor** field (it is assumed that the point of application of forces is the center of mass of the section). If the values of the internal forces for the connection have been obtained based on the results of the analysis of the system accounting for the importance factor (for example, when the design values of the applied loads were obtained taking into account this factor), the value equal to one has to be selected in the **Importance factor** drop-down list.

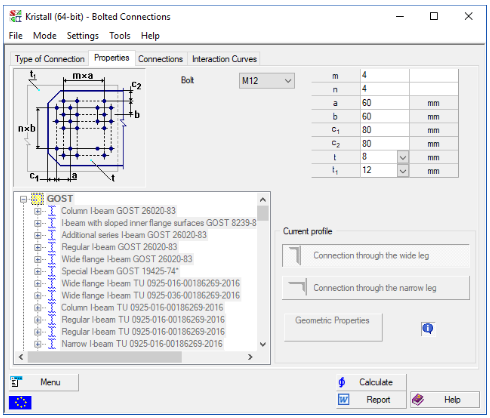


Figure 5.8.2-2. *The* **Properties** *tab of the* **Bolted Connections** *dialog box*

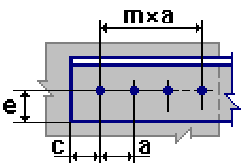
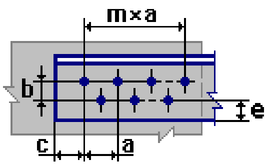
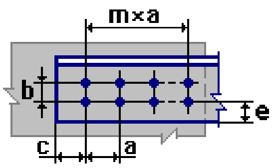
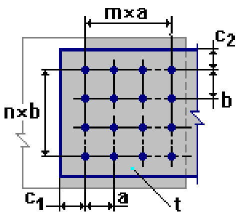
The **Properties** tab (Fig. 5.8.2-2) contains text fields for entering the values of the parameters of a structural design. In cases when a connection between rolled profiles (angle or I-beam) is considered, the database of the rolled profiles can be accessed.

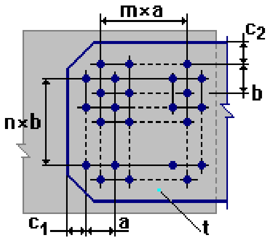
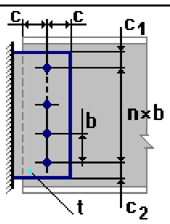
At the analysis of bolted connections the diameter of a bolt is selected from the drop-down list.

The initial data are checked for compliance with Sections 3.5(1) and 3.5(2), and Table 3.3 of EN 1993-1-8, which define the arrangement of bolts. The diameter of the bolt hole, d_0, is assumed to be larger than the bolt diameter for bolts from M12 to M14 by 1 mm, for bolts from M16 to M24 – by 2 mm, and for bolts M27 and larger – by 3 mm.

As a rule, there is no check for members to which angles and gusset plates are attached in bolted connections. An exception is the attachment of an I-beam to a supporting structure (the lowest button in the group).

Table 5.8.2-1. Bolted connections

Joint	Checks	Designations
	$e + 5(d_0 - 3\ mm)/3 \le b_L - t_L$ $4\ t_L + 40\ mm \ge c \ge 1{,}2\ d_0$ $\min(14\ t_L;\ 200\ mm) \ge a \ge 2{,}2\ d_0$ $(b_L - t_L)/2 \ge 1{,}5\ d_0$ $e \ge 1{,}2\ d_0$	d_0 is the diameter of a bolt hole; t_L is the angle thickness; b_L is the angle width;
	$e + b + 5(d_0 - 3\ mm) / 3 \le b_L - t_L$ $b \ge 2{,}2\ d_0$; $e \ge 1{,}2\ d_0$ $4\ t_L + 40\ mm \ge c \ge 1{,}2\ d_0$ $\min(14\ t_L;\ 200\ mm) \ge a \ge 2{,}2\ d_0$ $\min(14\ t_L;\ 200\ mm) \ge b \ge 2{,}4\ d_0$ $(b_L - t_L - b) / 2 \ge 1{,}5\ d_0$	
	$e + b + 5(d_0 - 3\ mm) / 3 \le b_L - t_L$ $b \ge 3{,}0\ d_0$; $e \ge 1{,}2d_0$ $4\ t_L + 40\ mm \ge c \ge 1{,}2\ d_0$ $\min(14\ t_L;\ 200\ mm) \ge a \ge 2{,}2\ d_0$ $\min(14\ t_L;\ 200\ mm) \ge b \ge 2{,}4\ d_0$ $(b_L - t_L - b)/2 \ge 1{,}5\ d_0$	
	$4\ t + 40\ mm \ge c_1 \ge 1{,}2\ d_0$ $4\ t + 40\ mm \ge c_2 \ge 1{,}5\ d_0$ $\min(14\ t;\ 200\ mm) \ge a \ge 2{,}2\ d_0$ $\min(14\ t;\ 200\ mm) \ge b \ge 2{,}4\ d_0$	t is the gusset plate thickness.

Joint	Checks	Designations
	$4\ t + 40\ mm \geq c_1 \geq 1{,}2\ d_0$ $4\ t + 40\ mm \geq c_2 \geq 1{,}5\ d_0$ $\min(14\ t;\ 200\ mm) \geq a \geq 2{,}2\ d_0$ $\min(14\ t;\ 200\ mm) \geq b \geq 2{,}4\ d_0$	
	$c \geq 1{,}2\ d_0$ $c_1 \geq 1{,}5\ d_0$ $c_2 \geq 1{,}5\ d_0$ $\min(14\ t;\ 200\ mm) \geq b \geq 2{,}2\ d_0$ $c_1 + c_2 + n\ b \leq h_w$	

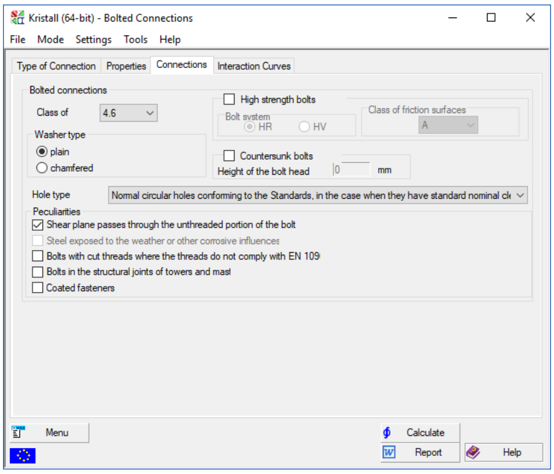


Figure 5.8.2-3. *The* **Connections** *tab of the* **Bolted Connections** *dialog box*

The **Connections** tab (Fig. 5.8.2-3) enables to take into account the characteristics of bolted connections. You can specify the class of bolts, type of washer, type of hole, etc. In the case when high-strength bolts with controlled tightening are used in the bolted connections, the class of the friction surface has to be specified as well.

The initial data are verified according to the rules given in Table 5.8.2-1.

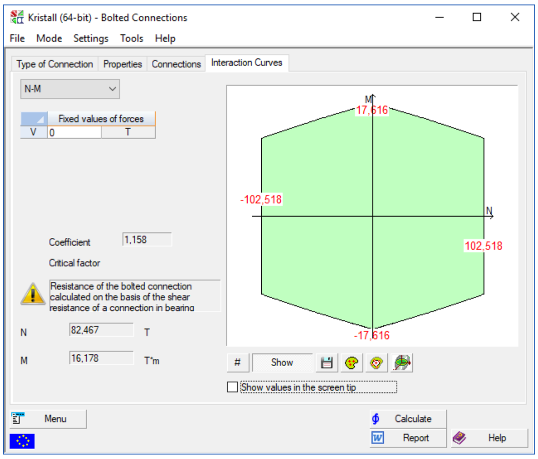


Figure 5.8.2-4. *The* **Interaction Curves** *tab of the* **Bolted Connections** *dialog box*

After you have finished entering the initial data, click the **Calculate** button to perform the analysis of your connection.

Interaction curves are plotted in the respective tab (Fig. 5.8.2-4) according to the rules described in Section 4.9.1.

Lists of checks for bolted connections are given in the Table 5.8.2-2. This table also indicates the sections of EN 1993-1-1 and EN 1993-1-8 which define these checks (a full list of results of the checks is given in the report document).

Table 5.8.2-2. A list of resistance checks for bolted connections

Factor	EN 1993-1-8	EN 1993-1-1
Bearing of angle	3.6.1, Table 3.4	
Shear strength of bolts	3.6.1, Table 3.4	
Bearing of flange	3.6.1, Table 3.4	
Bearing of web	3.6.1, Table 3.4	
Strength in net section of flange		6.2.3(4) (6.8), 6.2.5(4) (6.16)
Strength in net section of web		6.2.3(4) (6.8), 6.2.5(4) (6.16)

5.8.3 Welded Connections

The **Welded Connections** mode enables to determine the load-bearing capacity of welded connections. Structural designs of welded connections implemented in this mode are the most common ones in the engineering practice.

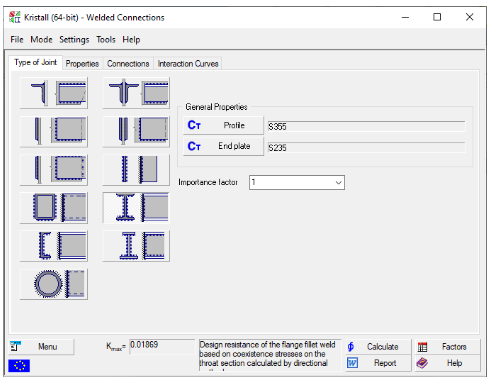


Figure 5.8.3-1. *The* **Type of Joint** *tab of the* **Welded Connections** *dialog box*

The **Type of Joint** tab (Fig. 5.8.3-1) contains a group of buttons for selecting a type of the welded connection.

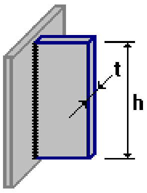
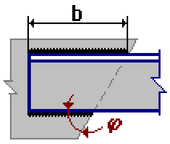
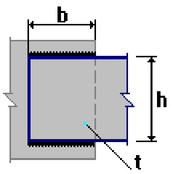
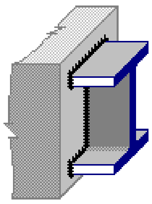
This tab also contains the  buttons, which enables to select steels for welding elements. The procedure of the steel selection is described in the Section 5.7.1.

Moreover, the importance factor which will be further multiplied by the values of all internal forces for all design combinations of loadings acting in the connection has to be specified in the **Importance factor** field (it is assumed that the point of application of forces is the center of mass of the section). If the values of the internal forces for the connection have been obtained based on the results of the analysis of the system accounting for the importance factor, the value equal to one has to be selected in the **Importance factor** drop-down list.

Welded connections are checked for compliance with

Sections 4.5.1(1) and 4.5.1(2), and if the requirements of these sections are violated, a message warning that the connection cannot be used due to the fact that its weld is too short is generated. The initial data are verified according to the rules given in Table 5.8.3-1.

Table 5.8.3-1. Welded connections

Joint	Checks	Designations
	h > 20 mm c_w < 1,2 t	t_L is the angle thickness; b_L is the width of an abutting leg; c_w is the weld leg; t_w is the thickness of an I-beam wall
	b > 40 mm 30° ≤ φ ≤ 90° c_w < 1,2 t	t is the gusset plate thickness
	b > 20 mm c_w < 1,2 t h / b > 0,1	
	c_w < 1,2 t_w	

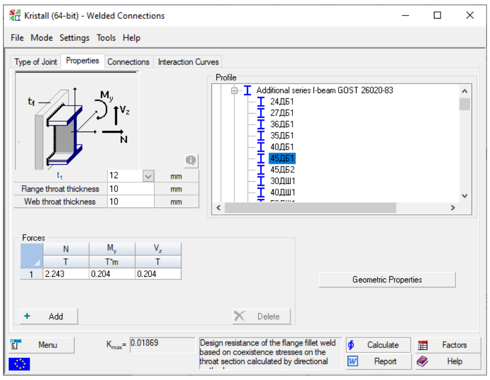


Figure 5.8.3-2. *The* **Properties** *tab of the* **Welded Connections** *dialog box*

The **Properties** tab (Fig. 5.8.3-2) contains text fields for entering the values of the parameters of a structural design. In cases when a connection between rolled profiles (angle or I-beam) is considered, the database of the rolled profiles can be accessed. For unequal angles, you can use radio buttons to specify which leg (wide or narrow) is used to attach the angle.

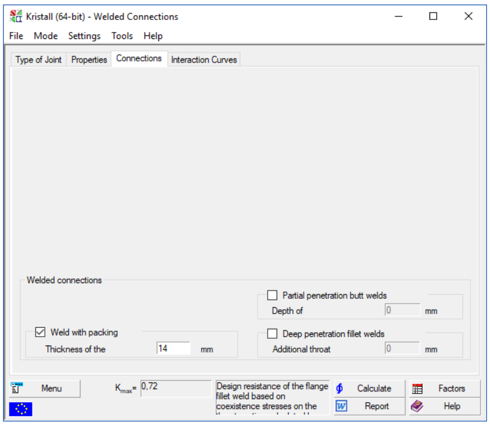


Figure 5.8.3-3. *The* **Connections** *tab of the* **Welded Connections** *dialog box*

The **Connections** tab enables to take into account the characteristics of welded connections, i.e. partial penetration of butt welds or deep penetration of fillet welds (Fig. 5.8.3-3). You can specify the class of bolts, type of washer, type of hole, etc. In the case when the welded connections are performed with gaskets, its thickness has to be specified.

Interaction curves are plotted in the respective tab according to the rules described in Section 4.9.1.

Lists of checks for welded connections are given in the Table 5.8.3-2. This table also indicates the sections of EN 1993-1-1 and EN 1993-1-8 which define these checks (a full list of results of the checks is given in the report document).

Table 5.8.3-2. A list of resistance checks for welded connections

Factor	EN 1993-1-8	EN 1993-1-1
Strength in net section of angle		6.2.3(4) (6.8), 6.2.5(4) (6.16)
Full strength of fillet weld	4.5.3.2 (4.1)	
Strength of fillet weld in the direction perpendicular to the weld axis	4.5.3.2 (4.1)	

5.8.4 Envelopes

This dialog box enables to solve a particular problem of determination of unfavorable combinations of multiple loads applied to flexural members.

This mode enables to generate the design combinations in accordance with the Sections 6.4.3.2…6.4.3.4 of EN 1990.

The combinations of loads for the established or transitional design situations (fundamental combinations) are created using the formula (6.10) of EN 1990 or as an alternative suggested by EN 1990, formulas (6.10a) and (6.10b) of EN 1990. The user has to select the variant in the **EN 1990 Settings** tab of the **Application Settings** dialog box (see Sec. 5.2).

The combination factors ψ_i used when generating the design combinations, as well as the factors $\gamma_{G,inf}$, $\gamma_{G,sup}$ and γ_Q are taken according to Annex A1 of EN 1990. The values of these factors are also specified in the **EN 1990 Settings** tab of the **Application Settings** dialog box (see Sec. 5.2) and can be changed by the user by selecting the **Other** option in the drop-down list.

It should be noted that the variable loads implicitly include a zero load (it enables to describe the case of absence of all variable loads). Therefore, when calculating maximum values (e.g., bending moments) we take the greatest of the positive moments and zero, and when calculating minimum values we take the least of the negative moments and zero.

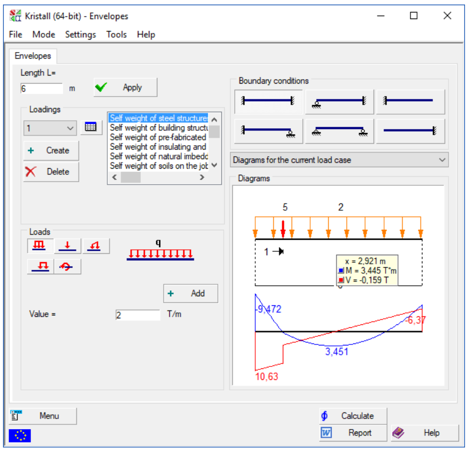


Figure 5.8.4-1. *The* **Envelopes** *dialog box*

The **Envelopes** dialog box (Fig. 5.8.4-1) is used to specify a beam span. Select one of the following boundary conditions using buttons on the top right: clamped at both ends; simply supported; hinged at one end and clamped at the other; or a cantilever beam.

Before entering a new load case (including the first one), click the **Create** button in the **Loadings** group. Then select a load case type (Permanent, Live loads of different categories, Snow loads, Other temporary etc.). The type of load case defines combination factors which will be used with the loads of this load case when determining the design combinations of load cases for the ultimate and serviceability limit states.

Loads are specified using five buttons which indicate the load type (distributed over the span, concentrated force, concentrated moment, trapezoid, and evenly distributed over a part of the span). Clicking the respective button will open text fields for entering the load parameters:

- for distributed loads — the load intensity;
- for a concentrated force — the value of the force and its position in the span;
- for a trapezoid load — value of the load at the beginning of the application area, distance from the beginning of the bar to the beginning of the application area, width of the load application area, value of the load at the end of the application area;
- for a load distributed over a part of the span — value of the load, distance from the beginning of the bar to the beginning of the application area, width of the load application area;
- for a concentrated moment — the value of the moment and its position in the span.

After specifying the properties of the load, click the **Apply** button.

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Loadings** list. The **Delete** button is used to delete a load case. To delete particular loads from one load case, you can use a table of loads which is invoked by clicking the respective button — .

Figure 5.8.4-2. *The* **Envelopes** *dialog box*

Once you click the **Apply** button, an image of the current loading is displayed in the **Diagrams** field with the superimposed diagrams of the bending moments and shear forces underneath. Once you have entered all the loadings, you can view the values of the extreme moments and their corresponding shear forces, as well as the extreme shear forces and their corresponding bending moments. To obtain the envelope diagrams select the respective items from the drop-down list above the **Diagrams** field (Fig. 5.8.4-2).

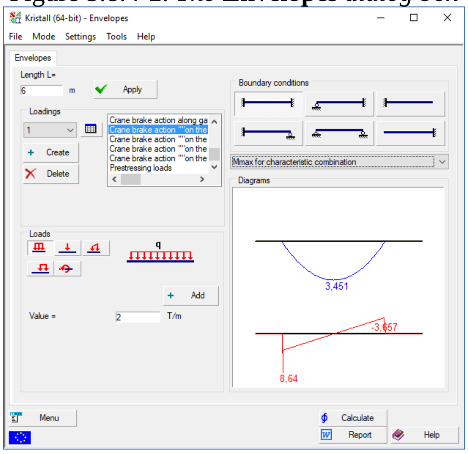


Figure 5.8.4-3. Digitization of the moment diagram

It should be noted that the diagrams of moments and shear forces can not display two different values in the same cross-section of the beam. Therefore, in cases when there is a leap of values (in a moment diagram under a concentrated moment, or in a shear force diagram under a concentrated force), it cannot be shown explicitly. Instead, values in two different but very close cross-sections will be displayed. This may produce a seeming violation of equilibrium.

If you place the mouse pointer in the diagram field, values of the moment and shear force in a particular cross-section corresponding to the position of the pointer will be displayed (Fig. 5.8.4-3).

5.8.5 Critical Moment

The **Critical Moment** mode enables to solve the problem of determination of an elastic critical bending moment for a flexural member which is further used in the check of the stability of in-plane bending. The calculation of the critical moment is performed using the analytical solution given in the Annex of the French NAD of EN 1993-1-1 depending on the character of the bending moment diagram, load application with respect to the shear center of the section, and also depending on the asymmetry of the section with respect to the axis of the greater stiffness. The calculations are performed only for the cross-sections symmetric with respect to the axis of the lower stiffness at their bending with respect to the axis of the greater stiffness.

The dialog box contains two tabs: **Properties** and **Critical Moment**.

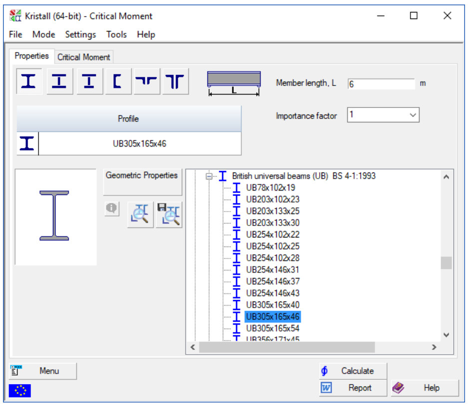


Figure 5.8.5-1. *The* **Properties** *tab of the* **Critical Moment** *dialog box*

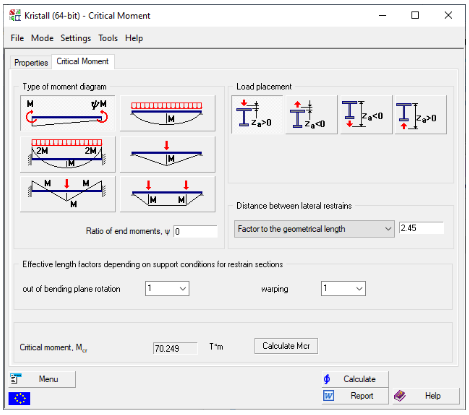


Рис. 5.8.5-2. *The* **Critical Moment** *tab of the* **Critical Moment** *dialog box*

The **Properties** tab (Fig. 5.8.5-1) is used to specify the cross-section of the member and its span length. The cross-section can be selected from the standard rolled profile database or from the custom welded profile database. The latter profiles can be also defined according to the rules described in the Section 5.2.

The **Critical Moment** tab (Fig. 5.8.5-2) is used to specify the type of the bending moment diagram by selecting it from six options given in the Annex of ENV 1993-1-1.

The calculation of the critical moment is performed using the analytical solution given in ENV 1993-1-1, depending on the character of the bending moment diagram, different boundary conditions of the bar element, restraints against warping, load application with respect to the shear center of the section, and also depending on the asymmetry of the section with respect to the axis of greater stiffness. Moreover, the calculation of the critical moment takes into account the pre-buckling curvature in accordance with [1].

For the linear diagram, the ratio of end moments, ψ, has to be specified. Since the type of the moment diagram is specified, the restraints of the member in the bending plane are defined as well. Out-of-plane restraints are defined by the ratio of the effective length, *l*, to the length of the span, *L*, specified for the compressed chord.

In addition, to calculate the critical moment, two effective length factors should be specified. The factor *k* refers to end rotation on plan and the factor k_w refers to end warping. Recommendations for assigning the coefficients *k*, k_w for some special cases can be found in the books

- L. Gardner, *Stability of Steel Beams and Columns: In Accordance with Eurocodes and the UK National Annexes*, Steel Construction Institute, 2011.
- N.Boissonnade, R.Greiner, J.P.Jaspart, J.Lindner, *Rules for member stability in EN 1993-1-1 : Background documentation and design guidelines*, ECCS European Convention for Constructional Steelwork, 2006.

The distance from the load application point to the top chord level, z_a, should be specified. The sign of z_a is assigned according to a schematic given in this tab.

Clicking the **Calculate** button will display the value of the critical moment, which is the result of the analysis in this mode.

It should be noted that EN 1993-1-1 limits the use of this check method for the sections which are characterized by a significant rotation of the principal axes of inertia of the “effective” section with respect to the principal axes of inertia of the gross cross-section. The respective limitations are introduced in this mode as well.

5.8.6 Geometric Properties

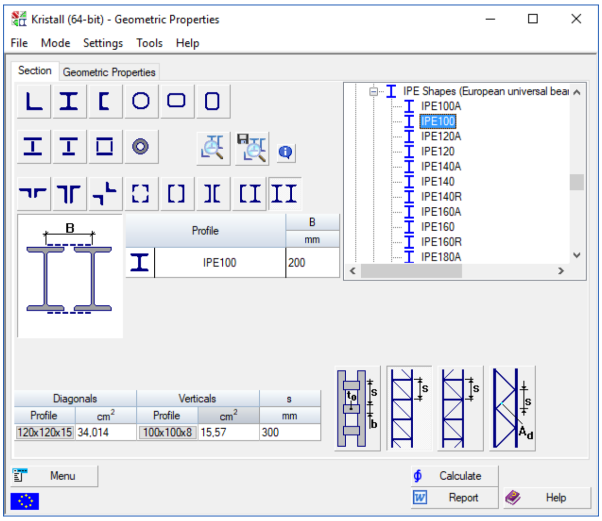


Figure 5.8.6-1. *The* **Section** *tab of the* **Geometric Properties** *mode*

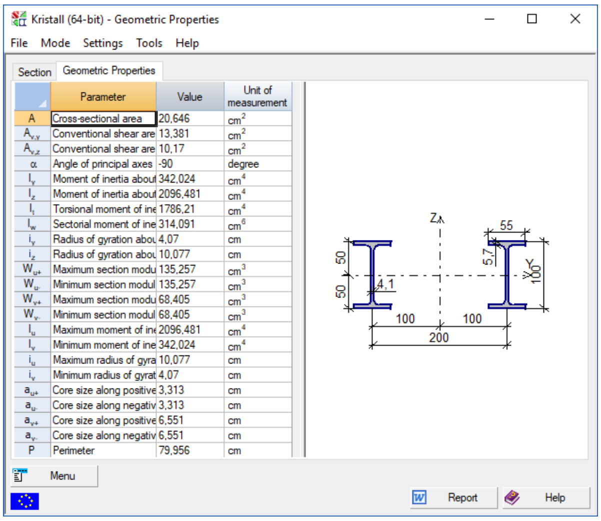


Figure 5.8.6-2. *The* **Geometric Properties** *tab of the* **Geometric Properties** *mode*

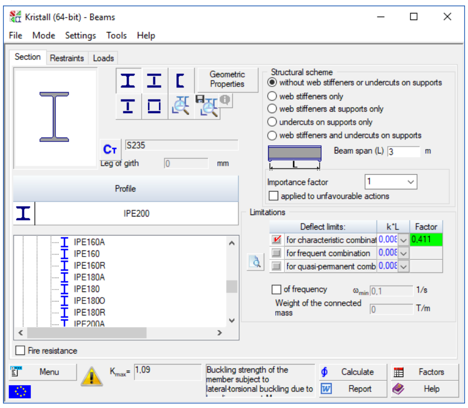
This mode enables to calculate the geometric properties of cross-sections in the same way as described in Section 5.8.1. The initial data are entered in the **Section** tab (Fig. 5.8.6-1). Results are displayed in the form shown in Fig. 5.8.6-2 in the **Geometric Properties** tab.

5.9 Design of Structural Members

To select a mode for the design of particular structural members, click the respective button in the main window. The application enables to perform the analysis of the following structural members: **Beams**, **Columns**, **Bracing** and **Trusses**.

5.9.1 Beams

a



The **Beams** mode enables to perform the check of a beam structure made of a rolled I-section profile or welded I-section and hollow section without longitudinal and transverse stiffeners reinforcing the web or with transverse stiffeners. The dialog box contains the following tabs: **General**, **Restraints**, **Loads**, **Web Stiffeners**, **End Undercut**.

The **General** tab (Fig. 5.9.1-1) contains a set of interface elements which are used to specify the initial data for performing the checks of the beam for the two limit states.

A rolled profile can be selected in the same way as described in Section 4.3 (see Fig. 5.9.1-1, *a*). If the beam cross-section type is an I-section or a hollow section, it is necessary to specify the sizes of the beam cross-section: height h_w and thickness t_w of the beam web, width b_f and thickness t_f of the flange (see Fig. 5.9.1-1, *b*).

The **Structural scheme** group contains radio buttons for selecting (when necessary) the transverse web stiffeners and the design of the beam end — either with or without end undercuts.

b

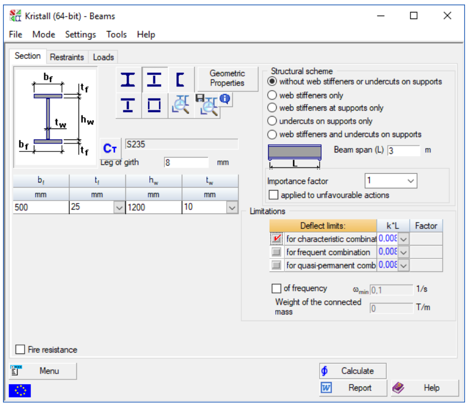


Figure 5.9.1-1. *The* **General** *tab of the* **Beams** *dialog box* (*a* – for a rolled I-beam, *b* – for a welded I-beam)

There are also text fields for entering the span length and the leg of girth welds (for welded sections).

This tab also contains the  button, which enables to select steel. The procedure of the steel selection is described in the Section 5.7.1.

The importance factor which will be further multiplied by all design values of the loads applied to the beam has to be specified in the **Importance factor** field.

The **Limitations** group of interface elements enables to specify the limitations of the beam deflection for the design combinations of loads of the serviceability limit state.

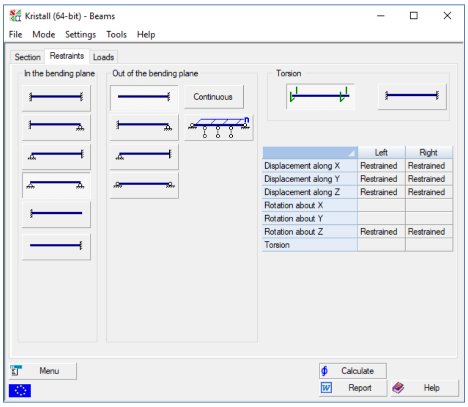


Figure 5.9.1-2. *The* **Restraints** *tab of the* **Beams** *dialog box*

The **Restraints** tab (Fig. 5.9.1-2) contains three groups of buttons for specifying a system of beam restraints in and out of the bending plane and against warping. The selection is performed by clicking a respective button. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

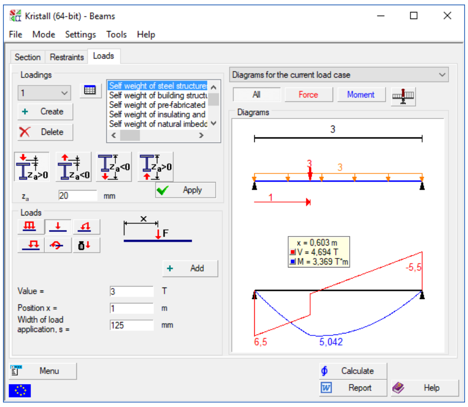


Figure 5.9.1-3. *The* **Loads** *tab of the* **Beams** *dialog box*

The **Loads** tab (Fig. 5.9.1-3) is used to specify the loads acting on the beam. This tab is nearly identical to that from Section 5.8. The difference is that the position of the application point of the considered loads with respect to the top chord of the beam has to be specified, which is assumed to be the same for all components of the current load case. When describing a concentrated load in addition to its value and position with respect to the left end of the beam, it is also necessary to specify the conditional width of the application of such a load. Furthermore, if you place the cursor in the diagram field and click the right mouse button, this will display a beam cross-section with isofields of normal and shear stresses depending on the specified parameters and corresponding to the design section defined by the cursor position (Fig. 5.6.1-3).

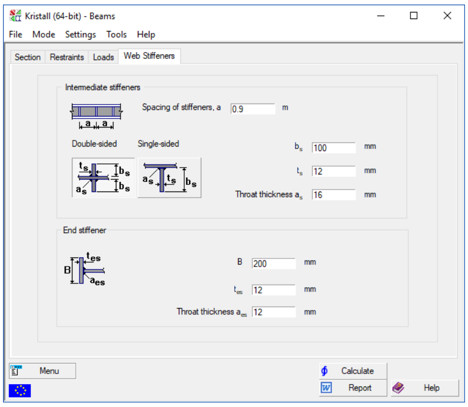


Figure 5.9.1-4. *The* **Web Stiffeners** *tab of the* **Beams** *dialog box*

The **Web Stiffeners** tab (Fig. 5.9.1-4) is used to specify data on the intermediate and bearing stiffeners. The tab is accessible in cases when one of the design options with stiffeners is selected in the **Structural scheme** group of the **Section** tab. The **Beams** mode provides the designs of double-sided and single-sided intermediate stiffeners welded to the beam web by fillet welds. It is necessary to specify the width and thickness of the intermediate and/or bearing stiffener, spacing of intermediate stiffeners, and the thickness of the fillet welds which are used to weld the stiffeners to the beam web.

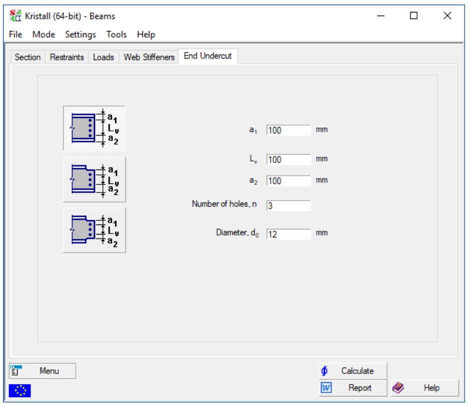


Figure 5.9.1-5. *The* **End Undercut** *tab of the* **Beams** *dialog box*

The **End Undercut** tab (Fig. 5.9.1-5) enables to specify data on the weakening of the beam cross-section in the area attaching to the supports. This tab is accessible in cases when one of the design options with end undercuts is selected in the **Structural scheme** group of the **Section** tab. A configuration of the end fixations is selected using the respective buttons, and then the respective sizes should be entered. Note that you specify the diameter of *bolt holes* rather than those of *bolts*.

A list of checks performed in this mode is given in the Table 2.6.1-1. This table also indicates the sections of EN 1993-1-1, EN 1993-1-5 and EN 1993-1-8 which define these checks (a full list of results of the checks is given in the report document).

Moreover, the report document will contain a table with design combinations of support reactions.

Table 5.9.1-1. A list of checks for beams according to EN 1993-1-1, EN 1993-1-5 and EN 1993-1-8

Factor	EN 1993-1-1	EN 1993-1-5	EN 1993-1-8
Strength under action of lateral force V_z	6.2.6 (6.17)...(6.21)		
Strength under action of bending moment M_y	6.2.5 (6.12)...(6.15)	4.4 (4.1)...(4.4), Table 4.1, 4.2	
Strength under combined action of bending moment M_y and lateral force V_z	6.2.5 (6.12)...(6.15), 6.2.8 (6.29)	4.4 (4.1)...(4.4), Table 4.1, 4.2	
Stability of in-plane bending	6.3.2 (6.54)...(6.57)		
Shear strength of web for a group of holes in the beam end fixation			3.10.2 (3.9), (3.10)
Strength of web under lateral force V_z		5.2 (5.1), (5.2), 5.3 (5.3)...(5.6), 5.4 (5.8), (5.9), 5.5 (5.10)	
Strength of web under local transverse load		6.1 (Fig. 6.1), 6.2 (6.1), (6.2), 6.3, 6.4 (6.3)…(6.5), 6.5 (6.8)…(6.13), 6.6 (6.14)	
Strength of intermediate stiffener from the condition of providing the stability under longitudinal force	6.3.1.1 (6.46)...(6.48), 6.3.1.2 (6.49), 6.3.1.3 (6.50), (6.51), 6.3.1.4 (6.52), (6.53)	9.1(2), 9.1(3); 4.4 (4.1)...(4.4), Table 4.1, 4.2	
Strength of end bearing stiffener from the condition of providing the stability under longitudinal force	6.3.1.1 (6.46)...(6.48), 6.3.1.2 (6.49), 6.3.1.3 (6.50), (6.51), 6.3.1.4 (6.52), (6.53)	9.1(2), 9.1(3); 4.4 (4.1)...(4.4), Table 4.1, 4.2	
Strength of intermediate stiffener from the condition of providing the stability under combined action of longitudinal force and bending moment	6.3.3 (6.61), (6.62)	9.1(2), 9.1(3); 4.4 (4.1)...(4.4), Table 4.1, 4.2	
Strength of intermediate transverse stiffeners providing a rigid reinforcement of the beam web		9.2.1 (9.1)	
Strength of welded connection with fillet welds between an intermediate stiffener and a beam web under a longitudinal force			4.5.3.2 (4.1)
Strength of welded connection with fillet welds between an intermediate stiffener and a beam web under a longitudinal force and a bending moment			4.5.3.2 (4.1)
Strength of welded connection with fillet welds between an end bearing stiffener and a beam web under a longitudinal force			4.5.3.2 (4.1)
Stability of compressed beam flange		8(1), (8.1)	

5.9.2 Columns

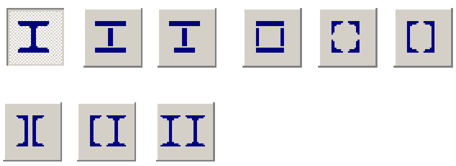


Figure 5.9.2-1. Types of column cross-sections

This mode enables to perform a check of a column structure the cross-section of which is selected from the types given in Fig. 5.9.2-1.

The dialog box contains the following tabs: **General Properties**, **Section**, **Forces**, and **Effective Lengths (or Effective Length in the XoY Plane** and . **Effective Length in the XoZ Plane)**.

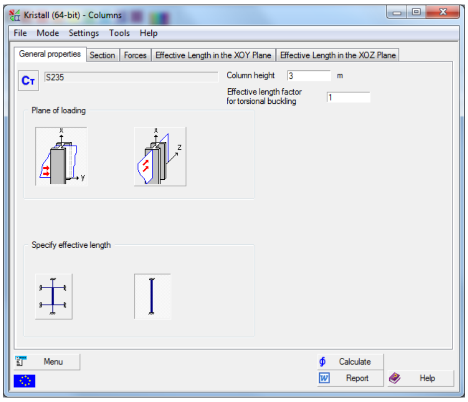


Figure 5.9.2-2. *The* **General Properties** *tab of the* **Columns** *dialog box*

The **General Properties** tab (Fig. 5.9.2-2) contains the **Column height** field where you have to specify the height of the column, and the  button, which enables to select steel. The procedure of the steel selection is described in the Section 5.7.1.

The **Load Plane** buttons are used to define an orientation of the deformation plane. All loads applied to the column are assumed to lie in that plane.

Buttons of the **Specify effective length** group enable to select a method for calculating the effective length of the column: either as a simple bar or a frame member, according to Annex E of EN 1993-1-1.

The design length factor for torsional buckling have to be also specified in the corresponded field.

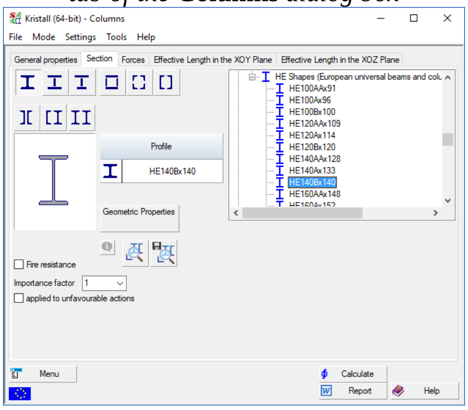


Figure 5.9.2-3. *The* **Section** *tab of the* **Columns** *dialog box*

The **Section** tab (Fig. 5.9.2-3) enables you to select a cross-section for the column and to specify its properties. Rolled profiles can be selected from the database. The properties of welded sections are entered into the respective text fields. These actions have been described earlier.

There are buttons for selecting the lattice type and text fields for entering their properties for the cross-sections with lattices (Fig. 5.9.2-4, *a*). According to recommendations of Sec. 6.4 of EN 1993-1-1, parallel lattices are oriented in the same way. For a cross-section from four angles, you have to use the respective buttons to specify the mutual arrangement of the lattices in adjacent faces (Fig. 5.9.2-4, *b*).

Moreover, the importance factor which will be further multiplied by the values of all internal forces has to be specified in the **Importance factor** field.

a *b*

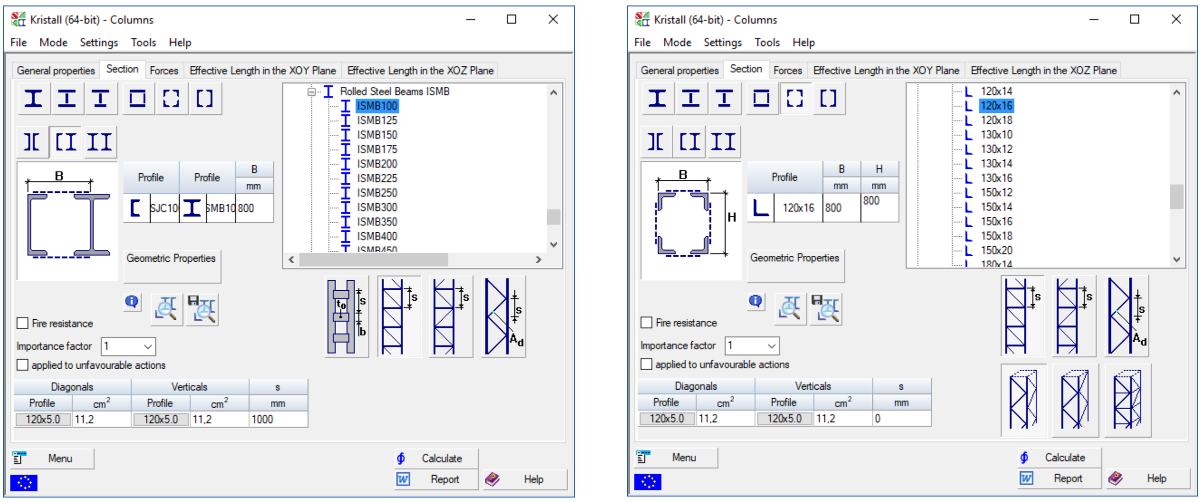


Figure 5.9.2-4. *Entering data on lattices*

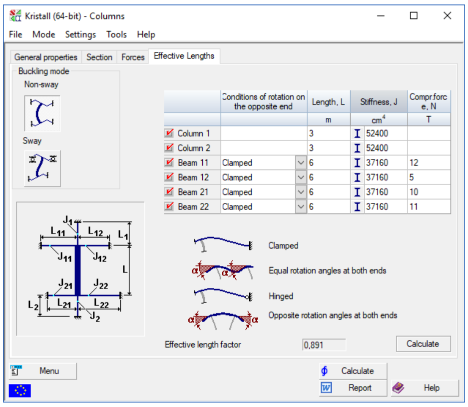


Figure 5.9.2-5. *The* **Effective Lengths** *tab of the* **Columns** *dialog box*

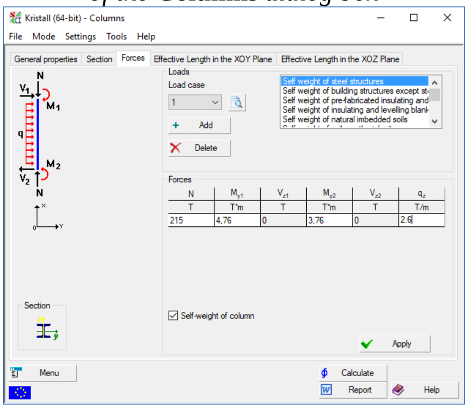


Figure 5.9.2-6. *The* **Forces** *tab of the* **Columns** *dialog box*

The **Effective Lengths** tab (Fig. 5.9.2-5) is used to specify the configuration of the part of the frame with the column, and to enter the properties needed to calculate the effective lengths. The lengths are calculated for a fragment of a frame structure located in the load plane which has been specified in the **General Properties** tab. The analysis requires data on the stiffness properties of the structural members adjacent to the column.

Once you click the **Calculate** button, the respective field will display the value of l / L. It is assumed that if the column was restrained in the other plane, the value would be $l / L = 1$.

If you have selected **Simple posts** as a method for calculating the effective length in the **General Properties** tab, the **Effective Length in the XoY (XoZ) Plane** tabs similar to those described in Sec. 4.8.4 (Fig. 4.8.4-2) will appear.

The **Forces** tab (Fig. 5.9.2-6) is similar to that described in the Section 4.10.6.

Clicking the button will display the diagrams of forces.

It should be noted that, unlike the **Resistance of Sections** mode, the positive longitudinal force here is always assumed to be compressive.

5.9.3 Bracing

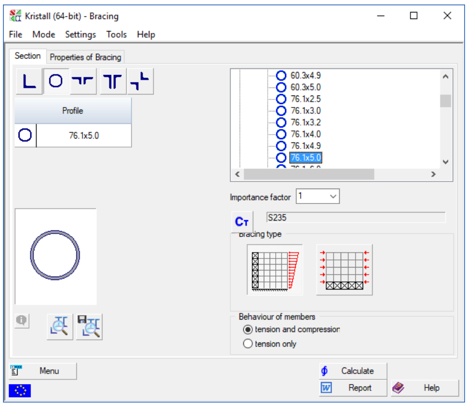


Figure 5.9.3-1. *The* **Section** *tab of the* **Bracing** *dialog box*

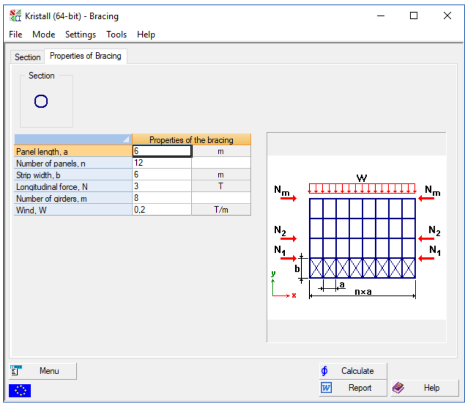


Figure 5.9.3-2. *The* **Properties of Bracing** *tab* (constraints over the roof are selected)

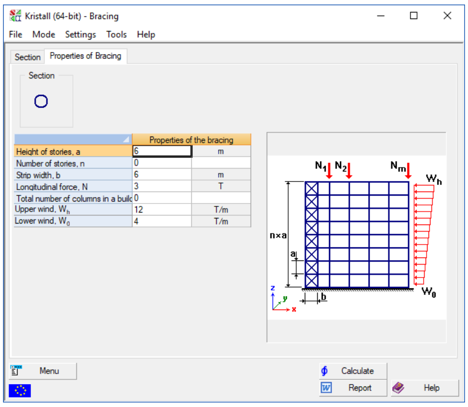


Figure 5.9.3-3. *The* **Properties of Bracing** *tab* (vertical braces are selected)

The **Bracing** mode enables to perform the strength and stability checks of bracing for the action of wind loads transferred onto this bracing, and for the action of equivalent forces. The equivalent forces can be used to allow for the possible imperfections of the braced structure, in compliance with Sec. 5.3.3 of EN 1993-1-1.

The dialog box contains the **Section** and **Properties of Bracing** tabs.

Select one of two basic wind bracing arrangements using the buttons of the **Bracing type** group in the **Section** tab (Fig. 5.9.3-1). These are either the vertical bracing used in a multi-storey frame, or the horizontal bracing over the roof of a building. Each of these structures can be described by one of the following design models:

- a compression-tension model, in which the bracing members should be able to resist both compression and tension;
- a tension model, in which a compressed flexible member is excluded from consideration after it loses its stability.

Select a model using radio buttons in the **Behavior of members** group. The cross-section can be selected from the rolled profile database or from a custom database.

The cross-section of bracing members can be from rolled or welded profiles and is assigned with the help of controls of the **Section** group.

This tab also contains the  button, which enables to select steel. The procedure of the steel selection is described in the Section 5.7.1. Moreover, the importance factor which will be further multiplied by the loads applied to the bracing has to be specified in the **Importance factor** field.

The **Properties of Bracing** tab (Fig. 5.9.3-2) is used to enter the information on the properties of a structural scheme and loads transferred to the bracing.

For the horizontal bracing (Fig. 5.9.3-2), the axial force *N* is determined as the force in the top chord of the braced trusses of the roof. If the roof beams are restrained against buckling, then in compliance with Sec. 5.3.3 (3) of EN 1993-1-1 the force *N* is determined by the value of the maximal bending moment. The force *N* is assumed to be the same for all girders.

For vertical bracing (Fig. 5.9.3-3), the force *N* is a load transferred to the column within the storey. It is assumed to be the same for all columns and all storeys.

Once you click the **Calculate** button, the respective field will display a value of the K_{max} factor and the name of a check in which it took place.

5.9.4 Trusses

This mode enables to perform all the necessary strength and stability checks of bar elements of steel trusses in accordance with the requirements of EN 1993-1-1:2005 and EN 1993-1-5:2005. The work begins with calculating the design values of the forces caused by the given external loads in structural designs most frequently used in practice. There is an option of finding a section from a pre-composed assortment of cross-sections.

List of checks of bar elements of trusses according to EN 1993-1-1:2005 and EN 1993-1-5:2005 which are implemented in this mode are given in the Table 5.9.4-1.

Table 5.9.4-1. List of checks of bar elements of trusses according to EN 1993-1-1 and EN 1993-1-5

Factor	EN 1993-1-1	EN 1993-1-5
Strength under action of longitudinal tensile force *N*	6.2.3 (6.5), (6.6)	
Strength under action of longitudinal compressive force *N*	6.2.4 (6.9)…(6.11)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Stability at flexural buckling about the y-y axis under action of longitudinal compressive force *N*	6.3.1 (6.46)…(6.51)	
Stability at flexural buckling about the z-z axis under action of longitudinal compressive force *N*	6.3.1 (6.46)…(6.51)	
Stability at torsional-flexural buckling under action of longitudinal compressive force *N*	6.3.1 (6.46)…(6.49), 6.3.1 (6.52), (6.53)	

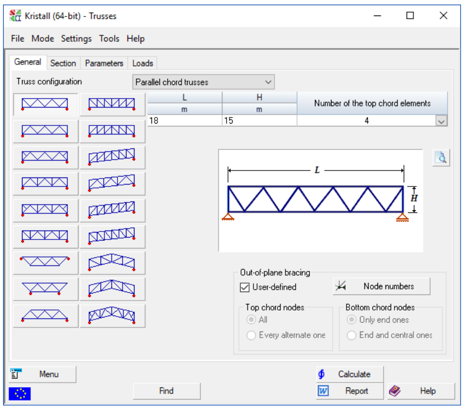


Figure 5.9.4-1. *The* **General** *tab of the* **Trusses** *dialog box*

The **Trusses** dialog box contains four tabs: **General**, **Section**, **Properties,** and **Loads**.

The **General** tab (Fig. 5.9.4-1) contains the **Truss configuration** drop-down list where the user has to select a truss type depending on the shape of its chords. The following types of trusses can be analyzed: parallel chord, triangular, trapezoid, with a polygonal top chord, one slope roof trusses and double slope roof trusses. All trusses are assumed to be statically determinate.

Specify the span of the truss, its height on the support and the number of of elements of the panel for the selected configuration. Additional geometric parameters have to be specified in case of a trapezoid truss or a truss with a polygonal top chord.

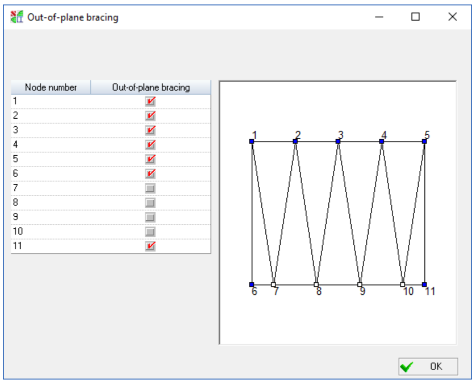


Figure 5.9.4-2. *The* **Out-of-plane bracing** *dialog box*

The respective radio buttons and checkboxes in the **Out-of-plane bracing** group are used to specify the method of out-of-plane bracing of the top chord nodes (all or every alternate) and the bottom chord nodes (only the end nodes or the end nodes and the midspan ones). The bracing in the truss plane is assumed to be statically determinate. When the **User-defined** checkbox is checked, the **Node numbers** button becomes accessible. Clicking this button invokes the **Out-of-plane bracing** dialog box (Fig. 5.6.4-2). The dialog provides a design model of the truss with numbered nodes and a table where each truss node is assigned a checkbox. A checked checkbox means that there is a bracing (linear and out of the truss plane) in the given node. The braced nodes are highlighted in blue in the model. Nodes in grey are braced by default and their condition cannot be modified.

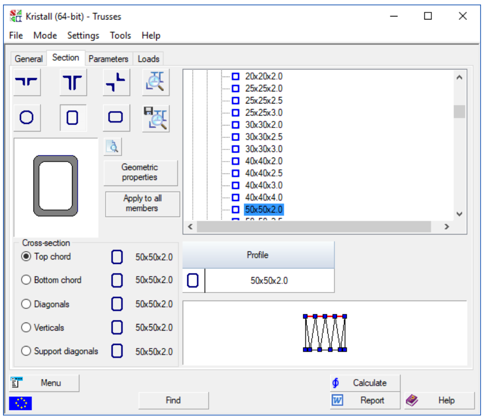


Figure 5.9.4-3. *The* **Section** *tab of the* **Trusses** *dialog box*

The **Section** tab (Fig. 5.9.4-3) is used to assign cross-sections to truss members. It is assumed that the sections of the chords do not vary along the truss length. The sections are made from double equal or unequal angles arranged as a tee (the latter come in two variations) or a cross of equal angles; and from round and rectangular pipes.

The sections are selected from a database of rolled profiles. The gap between the angles is specified in a table above the truss model when selecting a profile for each section type. Members of the same type (Top chord, Bottom chord, Diagonals, Verticals, Support diagonals) selected with the respective radio buttons are highlighted in red in the model, and the selection field displays their section.

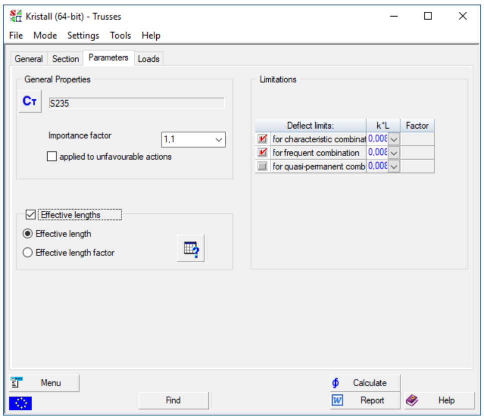
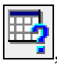


Figure 5.9.4-4. *The* **Properties** *tab of the* **Trusses** *dialog box*

The **Properties** tab (Fig. 5.9.4-4) enables to specify the class of steel for bar elements of a truss by clicking the button . The importance factor which will be automatically multiplied by the design values of all loads defined in the **Loads** tab has to be specified in the **Importance factor** field (see Fig. 5.9.4-6).

The **Limitations** group of interface elements in this tab (see Fig. 5.9.4-4) enables to specify a deflection limit of the truss in fractions of its span length for the *combinations of the serviceability limit states*: characteristic combination, for an almost permanent combination and for a frequent combination. You can specify the deflection limit for the truss in fractions of the span (value 1/*A*), where *A* is one of the most frequently used values (120, 150, 200, 250, 300 etc.).

This tab also contains the **Effective lengths** group of interface elements which enables to specify the information on the effective lengths of the bar elements of a truss. In order to do this check the **Effective lengths** checkbox, select a method of specifying the effective lengths (Effective length or the Effective length factor) and click the button , which invokes the **Effective Lengths** dialog box (see Fig. 5.9.4-5).

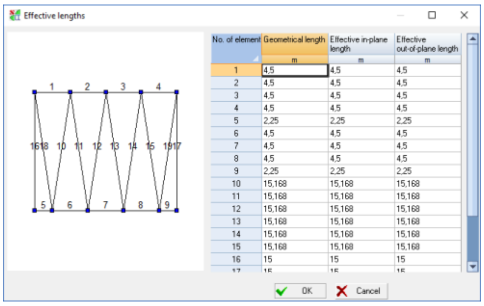
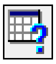
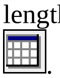


Figure 5.9.4-5. *The* **Effective Lengths** *dialog box*

The data on the effective lengths in the truss plane and out of the truss plane are specified in the table of the **Effective Lengths** dialog box for each bar element of the truss (the numbers of the elements are given in the model). Moreover each line of this table contains information of the geometric length of the respective element (see Fig. 5.9.4-5). The effective lengths of the bar elements of steel trusses are assumed to be equal to their geometric lengths by default (the effective length factor is taken as one by default).

Once you have entered the information on the effective lengths of bar elements of the truss the button  changes into: .

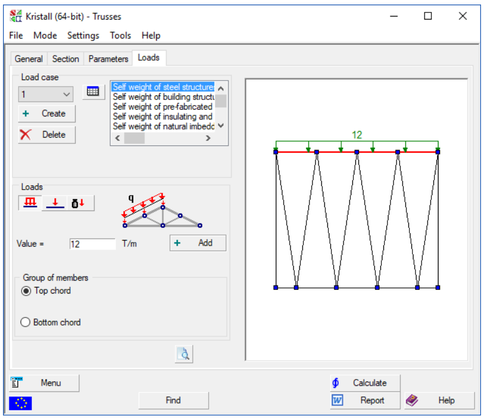


Figure 5.9.4-6. *The* **Loads** *tab of the* **Trusses** *dialog box*

The **Loads** tab (Fig. 5.9.4-6) is much similar to that described in the **Envelopes** mode (see Section 4.8.1). However, there are some differences. The user can specify either a uniformly distributed load or a concentrated load upon nodes. The distributed load is applied to either the whole chord or to the half of it. Number of the load application node is selected from the **Node number** drop-down list. The load application area is defined using the respective radio buttons in the **Group of elements** group: Top chord, Bottom chord, Left half of the top chord, or Right half of the top chord. Once you click the button to select the chord the uniformly distributed load is applied to, the respective chord will be highlighted in red in this window, and when you select the number of the node the concentrated load is applied to, the respective node will be highlighted in red. After clicking the **Add** button, a schematic of the respective load case with all the specified loads will appear in the window.

To edit the values of particular loads, you can use a table invoked by the button (see Section 4.8.1).

Clicking the **Forces in the truss members for the current load case** button invokes an information window displaying a design model of the truss with a diagram of internal forces (Fig. 5.9.4-7).

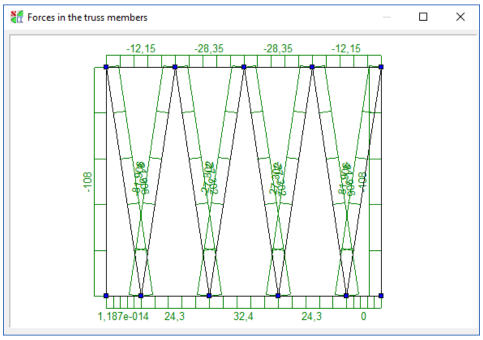


Figure 5.9.4-7. *The* **Forces in truss members** *information window*

Kristall (unlike SCAD) assumes that a truss is subjected to a nodal load. Thus the specified distributed load is not transferred onto the truss members; instead, it is assumed to be applied to some enclosing roof structure which performs the function of transferring the load onto the truss nodes.

As a rule, the top truss chord is subjected to a distributed load, and this fact is taken into account in the truss analysis performed by SCAD. The longitudinal force changes over the length in inclined members. SCAD calculates and displays maximum value of the force in the member. When converting the distributed load into the nodal loads, Kristall will display a value corresponding to the force in the middle of a SCAD finite element. Therefore the results generated by the two applications may differ.

It is possible to indicate the presence or absence of the dynamic loads on the truss. If all loads are static, the slenderness check of tension members is performed only in the vertical plane.

Clicking the **Calculate** button will display the value of K_{max} and indicate the type of check (strength, stability, slenderness) in which this maximum took place. You can also browse all the other utilization factors of restrictions by clicking the **Factors** button.

The **Find** button enables you to switch to a mode that performs the purposeful search of cross-sections for the truss members and replaces the numbers of profiles selected by the user (the cross-section type and bracing conditions are not modified). The application switches to the next number of profile greater in area in the catalogue the cross-sections were initially selected from if during the checks of the considered member (e.g., a top chord) the value of the utilization factor of restrictions $K > 1$ has been detected, or it switches to the next number of profile smaller in area if $K < 1$. Such switches are performed until all checks give $K < 1$ and the replacement of the profile with its adjacent smaller one gives $K > 1$, i.e. until the smallest profile that satisfies the requirements of the design codes is found.

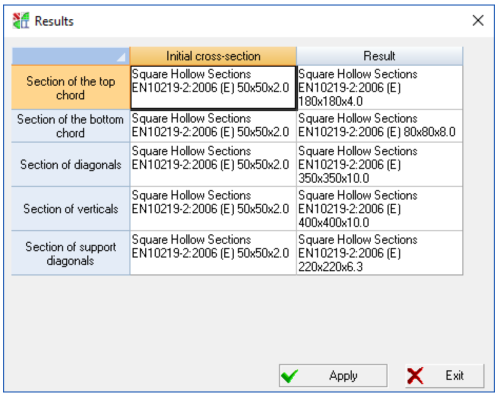


Figure 5.9.4-8. *The* **Results** *dialog box*

The searches through different groups of members (the top and bottom chords, diagonals, verticals) are mutually independent.

When the search is completed, a dialog with the recommendations on the selection of cross-sections appears on the screen (Fig. 5.9.4-8).

The user can reject these recommendations (using the **Exit** button) or accept them. In the latter case the **Apply** button is clicked, and all the recommended sections are transferred to perform the check analysis of the new structure.

If the maximum profile of the assortment was used in the search process and the value was still $K > 1$, the dialog will display the respective message and the **Apply** button will become inactive.

Note that the limitation of the deflection does not affect the results of the search.

Moreover, the report document will contain a table with support reactions and total weight of the truss.

5.10 Assumptions and Voluntary Decisions in Treatment of Eurocode 3

This section describes some assumptions taken by the developers of the software application in order to obtain solutions for cases not defined directly in EN 1993-1-1 Eurocode 3. The assumptions of this kind are, generally, a part of the treatment of a design code, and they are usually made by a design engineer during the practical work. However, seeing that the internal world of a software product is often closed from external observers, the developers decided that it was necessary to explicitly describe the assumptions here.

5.10.1 Calculation of a Critical Moment

The calculation of the critical moment is the focus of a special Annex of the French NAD of EN 1993-1-1. For beams with solid cross-sections, it defines a formula for calculating the elastic critical moment causing a torsional-flexural buckling:

$$\mathbf{M}_{\mathrm{cr}} = C_1 \frac{\pi^2 EI_z}{(kL)^2} \left\{ \left[\left(\frac{k}{k_w} \right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 GI_t}{\pi^2 EI_z} + \left(C_2 z_g - C_3 z_j \right)^2 \right]^{1/2} - \left(C_2 z_g - C_3 z_j \right) \right\},$$

where C_1, C_2, and C_3 are coefficients which depend on the type of loading and boundary conditions of the beam. They are given in Tables F.1.1 and F.1.2 of EN 1993-1-1 for six basic cases defined by the type of the moment diagram.

When the free length factor is k = 1.0 and the moment diagram is a straight line with the end values **M** and ψ**M**, the said Annex provides the following formula in addition to the table values of C_1:

$$C_1 = 1{,}88 - 1{,}40\,\psi + 0{,}52\,\psi^2 .$$

For other values of k and the other coefficients, similar formulas were obtained by us as an approximation that minimizes the root-mean-square deviation from the table values in the class of third-order polynomials. The calculations were performed in MS Excel and gave the following result:

k	C_1	C_3
1,0	$0{,}6674\psi^3 - 0{,}0279\psi^2 - 1{,}5486\psi + 1{,}9639$	$0{,}2354\psi^3 - 0{,}4454\psi^2 + 0{,}2697\psi + 0{,}9415$
0,7	$0{,}6569\psi^3 - 0{,}0856\psi^2 - 1{,}6902\psi + 2{,}1959$	$0{,}0247\psi^3 - 0{,}9187\psi^2 + 0{,}556\psi + 1{,}5173$
0,5	$0{,}6535\psi^3 - 0{,}1015\psi^2 - 1{,}7289\psi + 2{,}2596$	$-0{,}325\psi^3 - 1{,}5745\psi^2 + 0{,}9492\psi + 2{,}2586$

The quality of the approximation can be seen in diagrams given in Figs. 5.10.1-1 – 5.10.1-4.

It should be noted that this is not yet a solution to the problem because it is still unclear how to proceed in cases when the k factor has a different value, or in cases when the moment diagram is different from that shown in EN 1993-1-1.

The former problem is solved by interpolating between the solutions for the case k = 0,5 and for the case k = 1,0.

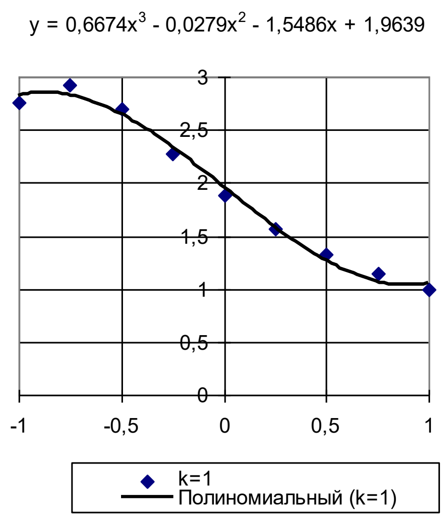


Figure 5.10.1-1

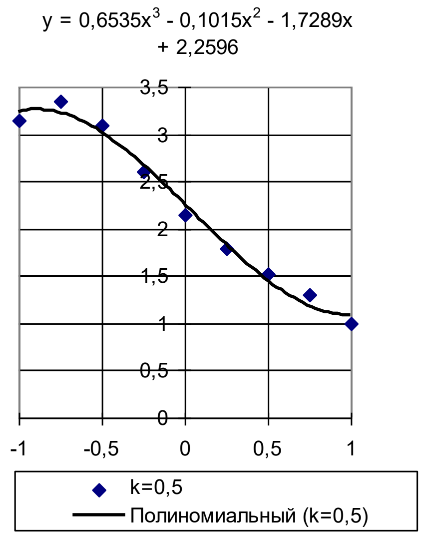


Figure 5.10.1-2

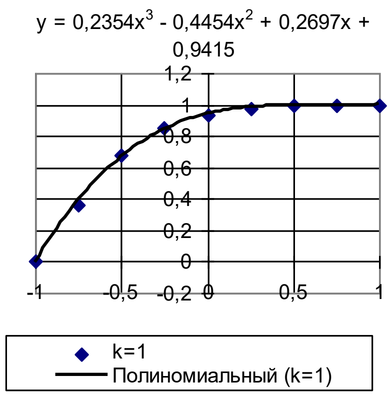


Figure 5.10.1-3

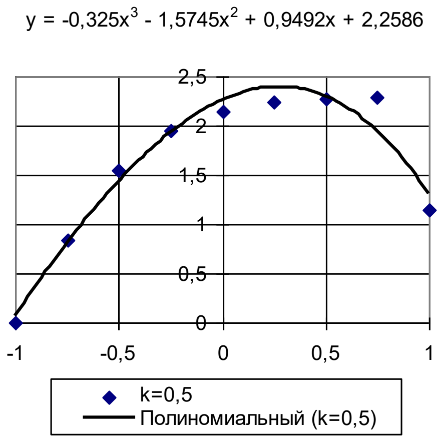
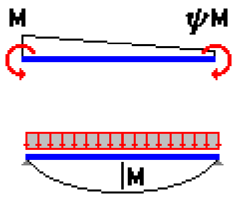


Figure 5.10.1-4

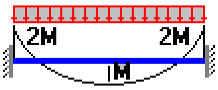
The latter problem is solved by expanding the actual moment diagram over a system of basis diagrams the solutions for which are provided by Annex F.



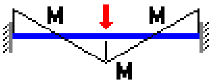
$\mathbf{M}_1(x) = \mathbf{M}_1\, f_1(x) = \mathbf{M}_1(1 - x / L)$

$\mathbf{M}_2(x) = \mathbf{M}_2\, f_2(x) = \mathbf{M}_2\, x / L$

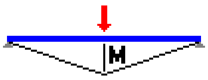
$\mathbf{M}_3(x) = \mathbf{M}_3\, f_3(x) = \mathbf{M}_3\, 4x(L - x) / L^2$



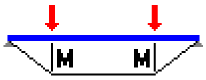
$\mathbf{M}_4(x) = \mathbf{M}_4\, f_4(x) = \mathbf{M}_4\,[-2 + 12x\,(L - x)\,/\,L^2]$



$\mathbf{M}_5(x) = \mathbf{M}_5\, f_5(x) = \mathbf{M}_5(-1 + 4x\,/\,L)$ at $(x < L\,/\,2)$
$= \mathbf{M}_5\, f_5(x) = \mathbf{M}_5(3 - 4x\,/\,L)$ at $(x > L\,/\,2)$



$\mathbf{M}_6(x) = \mathbf{M}_6\, f_6(x) = \mathbf{M}_6\, 2x\,/\,L$ at $(x < L\,/\,2)$
$= \mathbf{M}_6\, f_6(x) = \mathbf{M}_6\, 2\,(1 - x\,/\,L)$ at $(x > L\,/\,2)$



$\mathbf{M}_7(x) = \mathbf{M}_7\, f_7(x) = \mathbf{M}_7\, 4x\,/\,L$ at $(x < L\,/\,4)$
$= \mathbf{M}_7\, f_7(x) = \mathbf{M}_7$ at $(L\,/\,4 < x < 3L\,/\,4)$
$= \mathbf{M}_7\, f_7(x) = \mathbf{M}_7\,(4 - 4x\,/\,L)$ at $(x > 3L\,/\,4)$

The expansion over this system of functions is performed by finding the factors X_i with which the functional of a root-mean-square deviation of the given diagram $\mathbf{M}(x)$ from the weighted sum of the basis diagrams is minimized,

$$D = \int_0^L \left[\mathrm{M}(x) - \sum_{i=1}^{7} X_i\, f_i(x) \right]^2 dx\,.$$

The weighted average values of the coefficients are calculated in the following way:

$\psi = X_1\,/\,X_2;$

$$\begin{cases} C_1 = X_1\left(1{,}9639 - 1{,}548\psi - 0{,}0279\psi^2 + 0{,}6674\psi^3\right); \\ C_2 = 0; \\ C_3 = X_1\left(0{,}9415 + 0{,}2697\psi - 0{,}4454\psi^2 + 0{,}2354\psi^3\right); \end{cases}$$

$$\begin{cases} C_1 = C_1 + 1{,}132\,X_3; \\ C_2 = C_2 + 0{,}459X_3; \\ C_3 = C_3 + 0{,}525X_3; \end{cases}$$

$$\begin{cases} C_1 = C_1 + 1{,}285\,X_4; \\ C_2 = C_2 + 1{,}562\,X_4; \\ C_3 = C_3 + 0{,}753\,X_4; \end{cases}$$

$$\begin{cases} C_1 = C_1 + 1{,}365\,X_5; \\ C_2 = C_2 + 0{,}553\,X_5; \\ C_3 = C_3 + 1{,}730\,X_5; \end{cases}$$

$$\begin{cases} C_1 = C_1 + 1{,}565\,X_6; \\ C_2 = C_2 + 1{,}267\,X_6; \\ C_3 = C_3 + 2{,}640\,X_6; \end{cases}$$

$$\begin{cases} C_1 = C_1 + 1{,}046\,X_7; \\ C_2 = C_2 + 0{,}430\,X_7; \\ C_3 = C_3 + 1{,}120\,X_7. \end{cases}$$

5.11 References

[1] L.Gardner, D.Nethercot, Designers Guide to Eurocode 3: Design of Steel Buildings, EN 1993-1-1, EN 1993-1-3, EN 1993-1-8, ICE Publishing, 2011.

6. Magnum

Magnum enables to perform structural analysis and various checks of load-bearing cold-formed steel members for compliance with requirements of SP 260.1325800. A particular **Magnum** submodule enables to perform checks of members and joints for compliance with the requirements of EN 1993. It can be useful for experts who develop design documentation ordered by European companies. *EN 1993-1-3:2006. Eurocode 3 - Design of steel structures - Part 1-3: General rules - Supplementary rules for cold-formed members and sheeting* is used for the checks of load-bearing cold-formed members.

Magnum enables to perform various checks of load-bearing cold-formed bar members the sizes of which are predefined by the designer, i.e. the application works in the examination mode. If the user has selected a section of a load-bearing member from an assortment of cold-formed profiles, the application also enables to find a cold-formed profile from the assortment.

Magnum provides reference information on steel grades and assortments of rolled profiles, as well as recommendations of codes for selecting service factors and determining deflection limit values. The application has special reference modes to implement these features (see below).

6.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 6.1-1), with a set of buttons for selecting a working mode. These modes can be subdivided into the following groups:

- reference modes;
- auxiliary modes for designing steel structures;
- modes for checking the sections of load-bearing cold-formed members for compliance with the codes;
- design modes for performing the analysis of structures and their members.

A detailed description of each mode is given below. Their brief characteristic is given here.

The reference modes include:

- **Steel –** provides reference data on the mechanical properties (characteristic and design strength) of steels used for the load-bearing cold-formed members according to GOST R 52246 and GOST 14919 (when the analysis is performed according to SP 260.1325800), and according to the requirements of EN 10346, EN 10025-2, EN 10025-3 and EN 10025-4 when the analysis is performed according to EN 1993;
- **Assortment of Rolled Profiles** – enables to browse through assortments of cold-formed profiles;
- **Service Factors** – is used to browse and select values of service factors (γ_c) for structures and load-bearing cold-formed members in accordance with the requirements of Table 5.1 from SP 260.1325800 and Table 1 from SP 16.13330 (this mode is not provided for the analysis according to EN 1993);
- **Deflection Limits —** provides tables from SP 20.13330 “Loads and Actions” with limitations on deflections of load-bearing structural members (this mode is not provided for the analysis according to EN 1993).

The *auxiliary modes* include:

- **Envelopes** – enables to determine unfavorable combinations of multiple loads acting on flexural members, and to plot the envelope diagrams of moments and shear forces;
- **Influence Lines** – enables to plot influence lines for multi-span continuous beams of constant cross-section;
- **Geometric Properties** – enables to calculate the geometric properties of cold-formed steel sections;
- **Effective Lengths** – enables to calculate the effective lengths of the bar structural members depending on their boundary conditions.

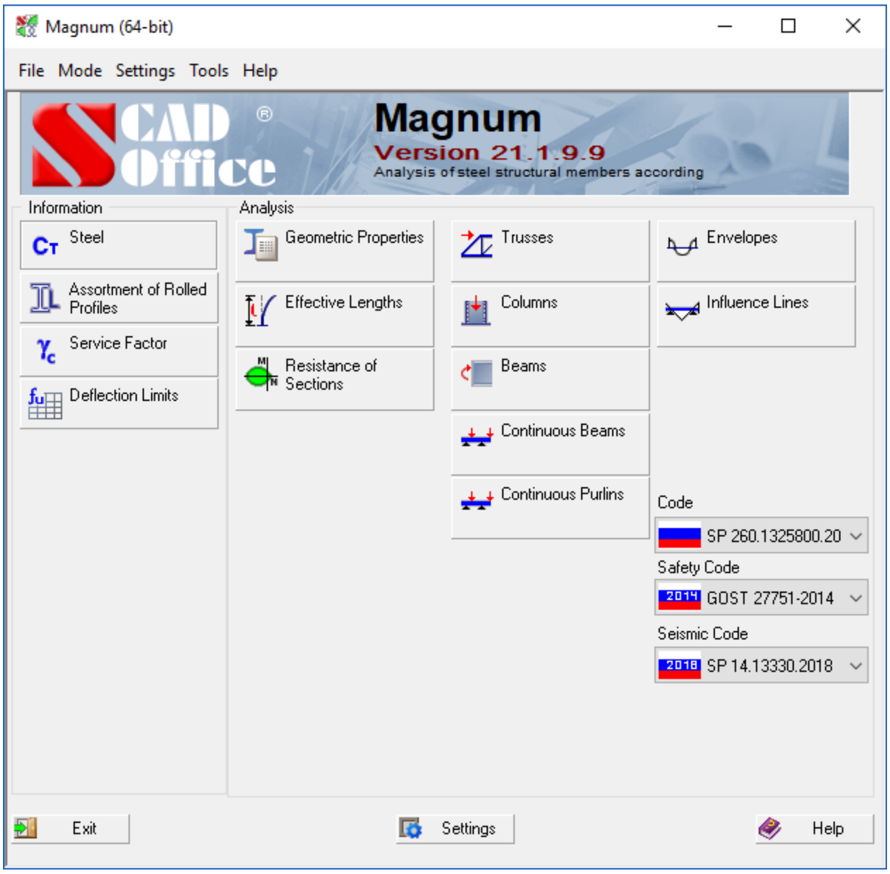


Figure 6.1-1. *The main window*

The modes described below are functional, used to perform checks of structural designs of cold-formed steel structures for compliance with strength, stability, and deformability requirements of design codes. The modes also enable to perform the investigation of the load-bearing capacity of cold-formed bar members by plotting the interaction curves. They include:

- **Resistance of Sections** – enables to determine the utilization factors of restrictions for any cross-section type available in the application database, under the action of any combination of internal forces;
- **Trusses** — this mode enables to perform all the necessary strength and stability checks of the cold-formed truss members for the most common truss designs used in self-framing metal buildings. The design values of internal forces and moments caused by the given vertical external loads are calculated automatically. The available cross-sections include compound or lattice doubly symmetric cold-formed sections, as well as hat channels and compound sections from double cold-formed angles;
- **Beams** — this mode is similar to the previous one, but it deals with single-span beams with doubly symmetric cold-formed steel sections and various boundary conditions;
- **Continuous Beams** — this mode implements the same functions as the **Beam** mode, but in application to a multi-span structure (up to five spans are allowed) which can have cantilevers at its ends. The available cross-sections include compound doubly symmetric cold-formed steel sections;
- **Continuous Purlins** – this mode is similar to the previous one, but it deals with multi-span cold-formed steel beams serving as roof purlins (including members of horizontal wind trusses) subjected to biaxial bending and an axial force. The available cross-sections include C- and Z-shaped profiles with or without edge fold stiffeners (single or double);

- **Columns** – this mode enables to perform all the necessary strength and stability checks of columns. The available cross-sections include compound doubly symmetric cold-formed steel sections.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

The main window also contains a number of buttons which are common controls for all working modes. These include the **Exit**, **Settings**, and **Help** buttons. The **Help** and **Exit** buttons perform functions common for a Windows application: providing reference information and finishing the current session respectively. The **Settings** button invokes the **Application Settings** dialog box where you can customize the program.

Design codes can be selected from the respective list. Information on the selected code is displayed in the bottom left corner of the active mode window.

The **Menu** button switches from any mode to the main window.

6.2 Settings

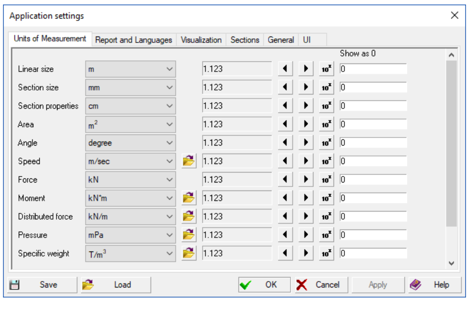


Figure 6.2-1. *The* **Units of Measurement** *tab of the* **Application Settings** *dialog box*

The **Application Settings** dialog box is invoked by the **Settings** button of the main window or from the respective menu. It can be invoked at any moment when working with the program. It is used to customize general parameters of the program.

The dialog contains the following tabs: **Units of Measurement, Report and Languages**, **Visualization**, **Sections**, **General**, **EN 1993 Settings**, **EN 1990 Settings**, and **Interface**.

The **Application Settings** dialog box is described in detail in Section **Settings** (2.2.) can be saved to an external file using the **Save** button, which can be subsequently loaded (the **Load** button).

6.3 Creating Cold-Formed Steel Sections

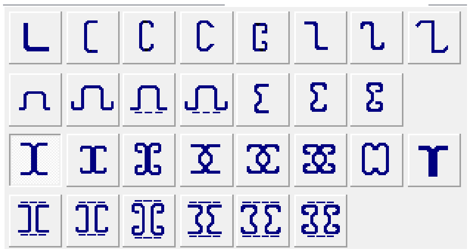


Figure 6.3-1. Types of cross-sections

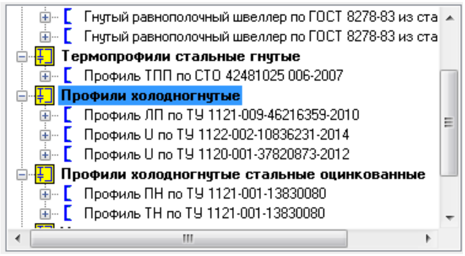


Figure 6.3-2. First level of the tree (selection of a catalogue)

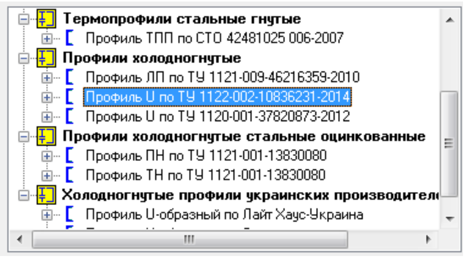


Figure 6.3-3. Second level of the tree (selection of a group)

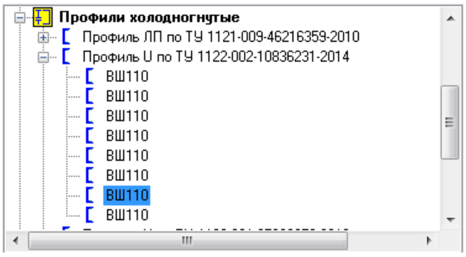


Figure 6.3-4. Third level of the tree (selection of a profile)

The action of selection of a cross-section for a structural member is common for most working modes of the application. To avoid unnecessary repetitions, these operations are described here separately.

Magnum works with various types of cold-formed sections. They include cold-formed steel angles, C-, Z-, sigma-shaped sections and hat channels with or without edge fold stiffeners (single or double), various compound and lattice cold-formed sections (see Fig. 6.3-1).

It should be noted that compound cold-formed thin-walled sections are assembled by bolting individual profiles with a certain spacing along the length of the bar, which does not require any additional structural members.

Cold-formed steel profiles for each type of section provided in the application can be selected from a tree-like database (Fig. 6.3-2).. It should be noted that it contains only the catalogues included in the In **Use list** in the Sections tab of the **Application Settings** dialog box.

First level of the tree is used to choose a catalogue of assortments of cold-formed steel profiles (Fig. 6.3-2). Second level of the tree (Fig. 6.3-3) enables you to select an assortment of cold-formed steel profiles, which will later be used to select the required profile. The list of accessible catalogues and assortments of cold-formed steel profiles is defined by the selected cross-section type. For example, if you click the button , only the C-shaped cold-formed steel profiles with single edge fold stiffeners will be accessible.

Finally, the third level of the tree enables to select a particular cold-formed steel profile (Fig. 6.3-4) which will be used in the cross-section of the bar member.

When a cold-formed profile is selected from the database, all its sizes will be provided in the table under the displayed section. The user can modify all or some of the dimensions of the section! (Fig. 6.3-5).

It is assumed that the thickness specified by the user is nominal, i.e. without zinc coating layer. It should be noted that the thickness of the zinc coating layer is not enough to calculate the nominal steel core thickness used in the analysis (see, for example, Sec. 7.1.5 of SP 260.1325800.2016). You also have to know the tolerance for the original sheet thickness. Therefore, it was decided that the user will immediately specify only the nominal steel core thickness (without specifying the thickness of the zinc coating layer and the tolerance for the original sheet thickness).

If a lattice cold-formed steel section with chords interconnected through the battens is selected, the Section tab will contain text fields for specifying the parameters of the battens (Fig. 6.3-6).

The limitations on the sizes of cold-formed steel sections and on the parameters of compound and lattice sections are listed in Table 6.3-1.

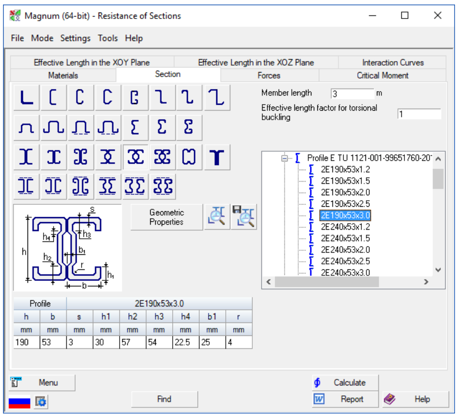


Figure 6.3-5. Specifying the cross-section sizes *in the* **Section** *tab*

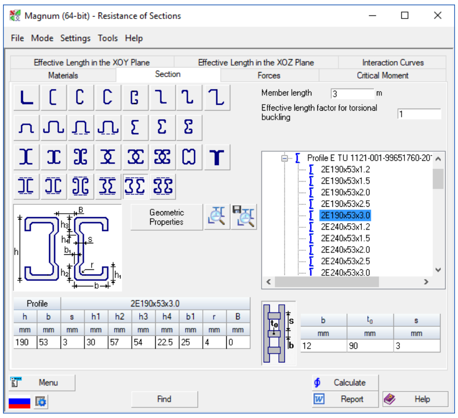


Figure 6.3-6. Specifying the parameters of battens for lattice sections *in the* **Section** *tab*

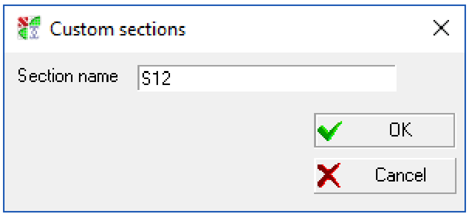


Figure 6.3-7. *The* **Custom sections** *dialog box* (*specifying a section name*)

Compound or lattice cold-formed steel sections defined in the application can be saved in a custom user database. This can be done by clicking the **Save cross-section in the user database** button — .

The **Custom sections** dialog box will appear where you can specify a name for the section you want to save (Fig. 6.3-7). Since the application does not verify the uniqueness of the names used, it is the user who has to take care of it.

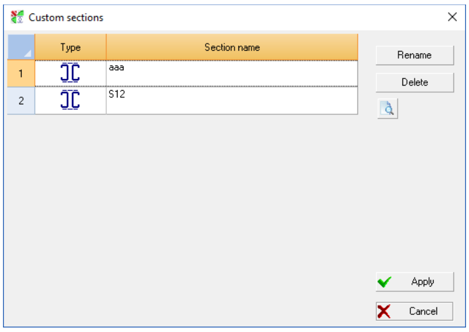


Figure 6.3-8. *The* **Custom sections** *dialog box*

To access the sections from the user database click the **Load cross-section from the user database** button — . The list of sections is displayed in the **Custom sections** dialog box (Fig. 6.3-8).

If duplicate names are detected in the database, use the **Rename** button to change the name of the respective profile**.**

Any of the sections listed in the **Custom sections** dialog box can be selected for further work. To do this, highlight the respective row in the table and exit the dialog by clicking the **Apply** button.

It should be noted that the program does not allow to select a previously saved section, if this section type is *not applicable* in a certain mode.

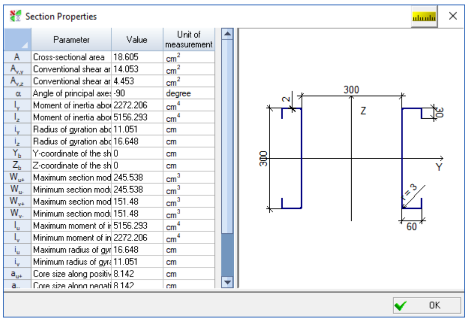


Figure 6.3-9. *The* **Section properties** *dialog box*

Geometric properties of the selected or specified cold-formed steel section can be browsed in the **Section properties** dialog box (Fig. 6.3-9), which is invoked by the **Geometric Properties** button. The section is displayed to scale, principal centroidal axes of inertia and main sizes are indicated.

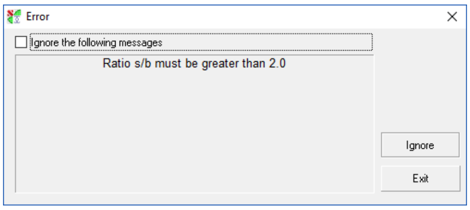


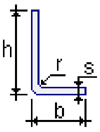
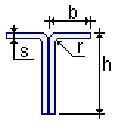
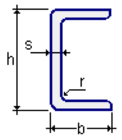
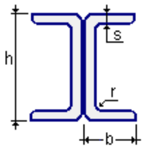
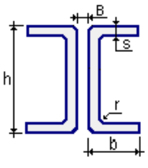
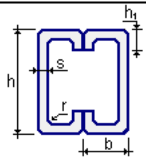
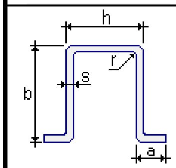
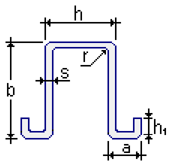
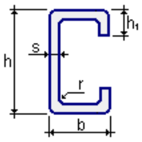
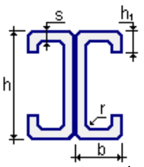
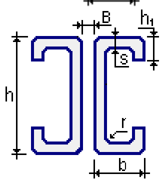
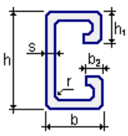
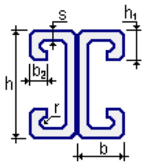
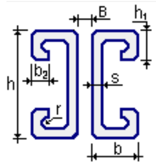
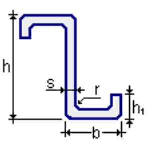
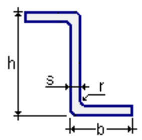
Figure 6.3-10. *An* **Error** *message*

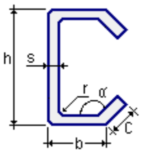
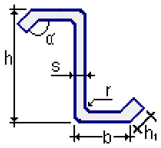
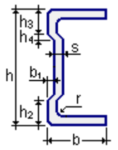
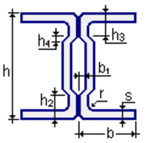
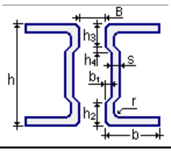
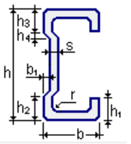
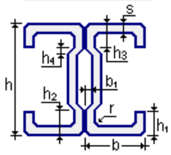
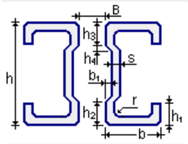
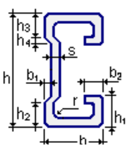
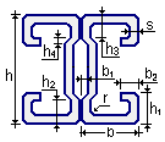
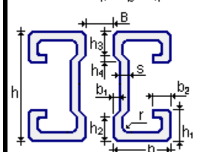
The application performs an automatic check of the section design. The list of limitations is given below in Table 6.3-1. When some of the limitations are violated, an error message with the description of the error will appear (see an example in Fig. 6.3-10).

In some cases the application allows you to ignore the warning that some limitations are violated (*Ignore* button). However, the negative consequences of this decision will not be analyzed.

In cases when an invalid numerical value is specified for a parameter, the error message will say **Invalid data**. Such a check is performed in all working modes.

Table 6.3-1. **Limitations on the sizes of cold-formed steel sections**

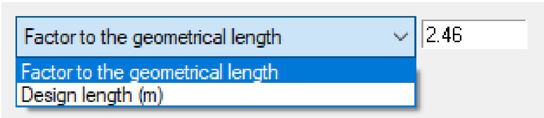
Section	Limitations	Section	Limitations
	$h \geq h_{min}$; $b \geq b_{min}$; where: $h_{min} = b_{min} = r + \ell_{min} + s$;		$h \geq h_{min}$; $b \geq b_{min}$; where: $h_{min} = b_{min} = r + \ell_{min} + s$;
  	$h \geq h_{min}$; $b \geq b_{min}$; where: $h_{min} = 2r + \ell_{min} + 2s$; $b_{min} = r + \ell_{min} + s$;		$h \geq h_{min}$; $b \geq b_{min}$; $h_1 \geq h_{1min}$; $h_1 \leq h/2$; where: $h_{min} = b_{min} = 2r + \ell_{min} + 2s$; $h_{1min} = r + \ell_{min} + s$;
			$h \geq h_{min}$; $b \geq b_{min}$; $a \geq a_{min}$; where: $h_{min} = b_{min} = 2r + \ell_{min} + 2s$; $a_{min} = r + \ell_{min} + s$;
			$h \geq h_{min}$; $b \geq b_{min}$; $a \geq a_{min}$; $h_1 \geq h_{1min}$; $h_1 \leq b$; where: $h_{min} = b_{min} = a_{min} = 2r + \ell_{min} + 2s$; $h_{1min} = r + \ell_{min} + s$;
  	$h \geq h_{min}$; $b \geq b_{min}$; $h_1 \geq h_{1min}$; $h_1 \leq h/2$; where: $h_{min} = b_{min} = 2r + \ell_{min} + 2s$; $h_{1min} = r + \ell_{min} + s$;	  	$h \geq h_{min}$; $b \geq b_{min}$; $h_1 \geq h_{1min}$; $b_2 \geq b_{2min}$; $h_1 \leq h/2$; $b_2 \leq b - s$; where: $h_{min} = b_{min} = h_{1min} = 2r + \ell_{min} + 2s$; $b_{2min} = r + \ell_{min} + s$;
	$h \geq h_{min}$; $b \geq b_{min}$; $h_1 \geq h_{1min}$; $h1 \leq h/2$; where: $h_{min} = b_{min} = 2r + \ell_{min} + 2s$; $h_{1min} = r + \ell_{min} + s$;		$h \geq h_{min}$; $b \geq b_{min}$; where: $h_{min} = 2r + \ell_{min} + 2s$; $b_{min} = r + \ell_{min} + s$;

Section	Limitations	Section	Limitations
	$179^o >= \alpha > 0^o$; $c \le h/2/\sin(180^o - \alpha)$; $h \ge h_{min}$; $b \ge b_{min}$; $c \ge c_{min}$; where: $h_{min} = 2r + \ell_{min} + 2s$; $b_{min} = r\,(1 + tg(90^o - \alpha/2)) + 2s + \ell_{min}$; $c_{min} = r * tg(90^o - \alpha/2) + \ell_{min} + s$;		$179^o \ge \alpha > 0^o$; $h_1 \le h/2/\sin(180^o - \alpha)$; $h \ge h_{min}$; $b \ge b_{min}$; $h_1 \ge h_{1min}$; where: $h_{min} = 2r + \ell_{min} + 2s$; $b_{min} = r\,(1 + tg(90^o - \alpha/2)) + 2s + \ell_{min}$; $h_{1min} = r\,tg(90^o - \alpha/2) + \ell_{min} + s$;
  	$b \ge b_{min}$; $h_3 \ge h_{3min}$; $h_2 \ge h_{2min}$; $h_4 \ge h_{4min}$; $b_1 \ge b_{1min}$; $h \ge h_{min}$; $b_1 \le b_{1max}$; $h_2 \le h_{2max}$; $h_3 \le h_{3max}$; $h_4 \le h_{4max}$; where: $b_{min} = r + \ell_{min} + s$; $\alpha = arctg(b_1/h_4)$; $h_{3min} = h_{2min} = r + s + \ell_{min} + (r + s)/tg(90^o - \alpha/2)$; $h_{4min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \cos(\alpha)$; $b_{1min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \sin(\alpha)$; $h_{min} = h_{3min} + h_{2min} + 2h_{4min} + 2(r + s)/tg(90^o - \alpha/2) + \ell_{min}$; $b_{1max} = b - s$; $h_{2max} = h_{3max} = (h - 2(r + s)/tg(90^o - \alpha/2) - \ell_{min} - 2h_{4min})/2$; $h_{4max} = (h - h_{2min} - h_{3min} - 2(r + s)/tg(90^o - \alpha/2) - \ell_{min})/2$;		
  	$b \ge b_{min}$; $h_3 \ge h_{3min}$; $h_2 \ge h_{2min}$; $h_4 \ge h_{4min}$; $b_1 \ge b_{1min}$; $h \ge h_{min}$; $b_1 \le b_{1max}$; $h_2 \le h_{2max}$; $h_3 \le h_{3max}$; $h_4 \le h_{4max}$; $h_1 \le h/2$; where: $b_{min} = 2r + \ell_{min} + 2s$; $\alpha = arctg(b_1/h_4)$; $h_{3min} = h_{2min} = r + s + \ell_{min} + (r + s)/tg(90^o - \alpha/2)$; $h_{4min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \cos(\alpha)$; $b_{1min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \sin(\alpha)$; $h_{min} = h_{3min} + h_{2min} + 2h_{4min} + 2(r + s)/tg(90^o - \alpha/2) + \ell_{min}$; $b_{1max} = b - s$; $h_{2max} = h_{3max} = (h - 2(r + s)/tg(90^o - \alpha/2) - \ell_{min} - 2h_{4min})/2$; $h_{4max} = (h - h_{2min} - h_{3min} - 2\,(r + s)/tg(90^o - \alpha/2) - \ell_{min})/2$;		
  	$b \ge b_{min}$; $h_3 >= h_{3min}$; $h_2 >= h_{2min}$; $h_4 \ge h_{4min}$; $b_1 \ge b_{1min}$; $h \ge h_{min}$; $b_1 \le b_{1max}$; $h_2 \le h_{2max}$; $h_3 \le h_{3max}$; $h_4 \le h_{4max}$; $h_1 \le h/2$; $b_2 \le b_{2min}$; where: $b_{min} = 2r + \ell_{min} + 2s$; $\alpha = arctg(b_1/h_4)$; $h_{3min} = h_{2min} = r + s + \ell_{min} + (r + s)/tg(90^o - \alpha/2)$; $h_{4min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \cos(\alpha)$; $b_{1min} = (2(r + s)/tg(90^o - \alpha/2) + \ell_{min}) \sin(\alpha)$; $h_{min} = h_{3min} + h_{2min} + 2h_{4min} + 2(r + s)/tg(90^o - \alpha/2) + \ell_{min}$; $b_{1max} = b - s$; $h_{2max} = h_{3max} = (h - 2\,(r + s)/tg(90^o - \alpha/2) - \ell_{min} - 2h_{4min})/2$; $h_{4max} = (h - h_{2min} - h_{3min} - 2(r + s)/tg(90^o - \alpha/2) - \ell_{min})/2$; $b_{2min} = b - b_1 - s$ – when $h_1 \ge h_2 + h_4$; otherwise: $b_{2min} = b - s$ – when $h_1 \le h_2$; $b_{2min} = b - s/\cos(\alpha) - (h_1 - h_2) tg(\alpha)$ – when $h_1 > h_2$;		

Notes: ℓ_{min} is the minimum linear section size, ℓ_{min} = 1 mm.

6.4 Specifying the Effective Lengths

It is necessary to specify the data on the effective lengths of structural members in some modes. In many cases you can specify it either in the form of the effective length factor or the effective length, which can be selected from the drop-down list next to the text field:



6.5 Advanced Settings

In some modes there is a button  next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional restrictions for selecting steel sections (Fig. 6.5-1).

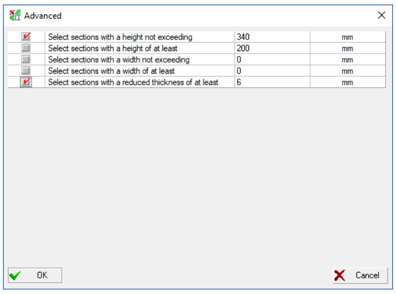


Figure 6.5-1. *The* **Additional Settings** *dialog box*

In particular, you can specify the restrictions on the height and width of a section.

When calculating for the action of accidental loads, by default characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017). Since this section was excluded by Change No. 2 to SP 296.1325800.2017, the user can (using the appropriate checkbox) choose which material properties (characteristic or design) to use.

6.6 Reference Modes

6.6.1 Steel

The **Steel** mode provides reference data on the mechanical properties (characteristic and design strength) of steels used for the load-bearing cold-formed members according to GOST R 52246 and GOST 14919 (when the analysis is performed according to SP 260.1325800), and according to the requirements of EN 10346, EN 10025-2, EN 10025-3 and EN 10025-4 (when the analysis is performed according to EN 1993).

The **Cold-Rolled Steel Sheets** tab (Fig. 6.6.1-1, Fig. 6.6.1-2) provides reference data on the design strength based on the yield strength and on the ultimate strength of the cold-rolled steel sheets used in the cold-formed structures.

When the **Steel** mode is invoked from the design modes, the **Apply** button appears in the **Steel** dialog box (Fig. 6.6.1-3) and enables to transfer the properties of the selected steel to the active design mode.

If you need to perform an analysis for steel with mechanical properties different from the standard ones, you can use the **User Defined Steel** tab (Fig. 6.6.1-3) of the **Steel** dialog box. It enables to specify the values of the yield and ultimate strength, which will later be used in the analysis.

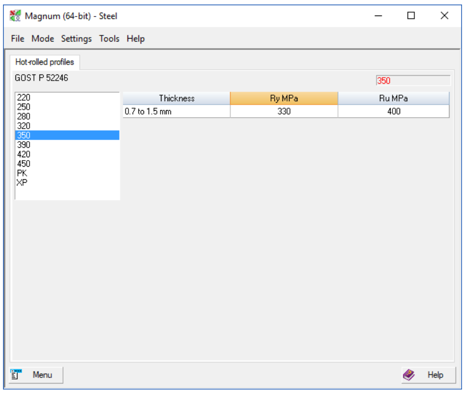


Figure 6.6.1-1. *The* **Cold-Rolled Steel Sheets** *tab of the* **Steel** *dialog box* (when the analysis is performed according to SP 260.1325800)

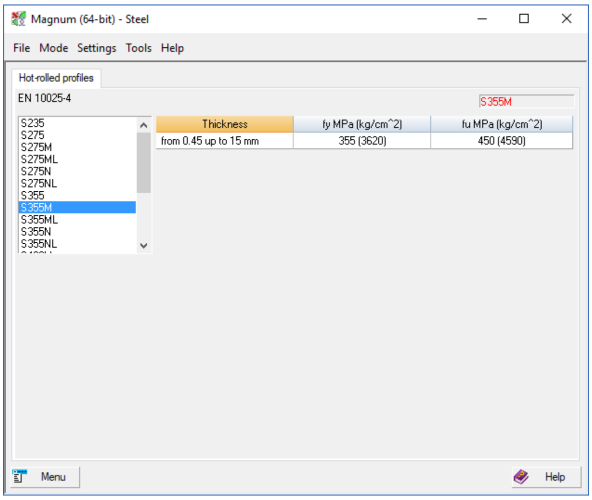


Figure 6.6.1-2. *The* **Cold-Rolled Steel Sheets** *tab of the* **Steel** *dialog box* (when the analysis is performed according to EN 1993)

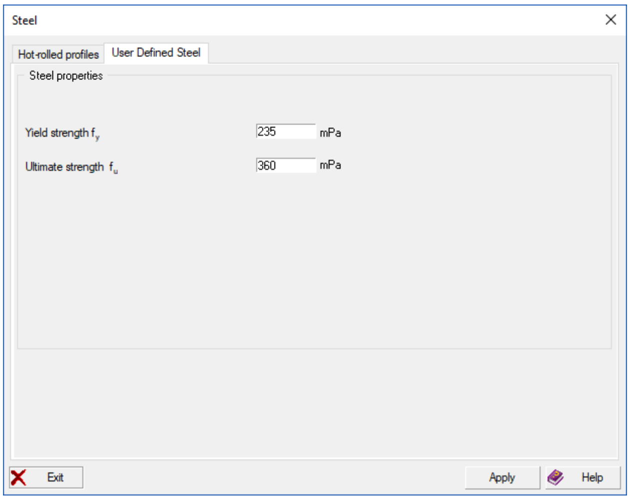


Figure 6.6.1-3. *The* **User Defined Steel** *tab of the* **Steel** *dialog box* (invoked from a design mode)

6.6.2 Assortment of Rolled Profiles

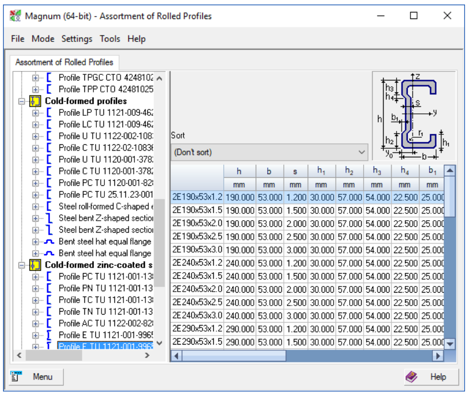


Figure 6.6.2-1. *The* **Assortment of Rolled Profiles** *dialog box* (an assortment has been selected)

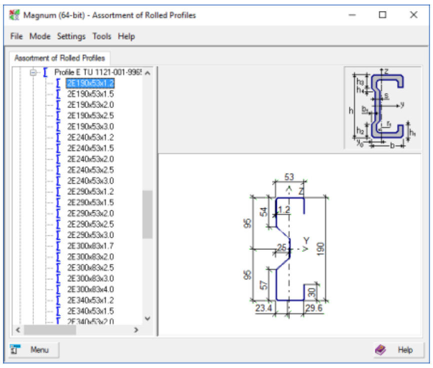


Figure 6.6.2-2. *The* **Assortment of Rolled Profiles** *dialog box* (a particular profile has been selected)

The **Assortment of Rolled Profiles** mode (Fig. 6.6.2-1) enables you to browse through cold-formed steel profile assortments available in the database of **Magnum**.

The dialog box contains a list of assortments represented by a tree-like structure on the left, and a table with data on the respective cold-formed profiles on the right. A drop-down list of profile properties which can be used to sort the table is placed above the table.

To view the section with its dimensions, open the profile group list and choose the respective profile (Fig. 6.6.2-2).

6.6.3 Service Factor

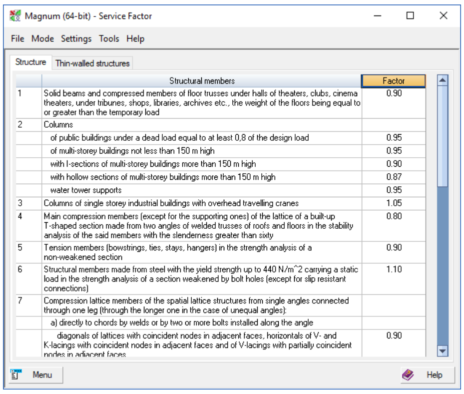


Figure 6.6.3-1. *The* **Structure** *tab of the* **Service Factor** *dialog box*

The **Service Factor** mode is used to browse and select values of service factors (γ_c) for structures and load-bearing cold-formed members. This mode is not provided for the analysis according to EN 1993.

The **Service Factor** dialog box contains two tabs: **Structure** (Fig. 6.6.3-1) and **Thin-walled structures** (Fig. 6.6.3-2), which provide data on the values of service factors given in Table 5.1 of SP 260.1325800 and Table 1 of SP 16.13330 respectively.

If the **Service Factor** mode has been invoked from a design mode, then after clicking the **Apply** button the selected value of the factor will be exported into this mode and used in the analysis of load-bearing cold-formed members (Fig. 6.6.3-2).

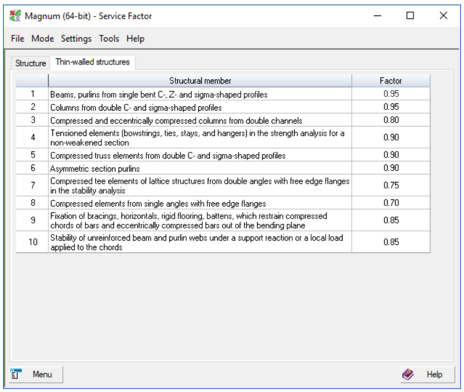


Figure 6.6.3-2. *The* **Thin-walled Structures** *tab of the* **Service Factor** *dialog box*

6.6.4 Deflection Limits

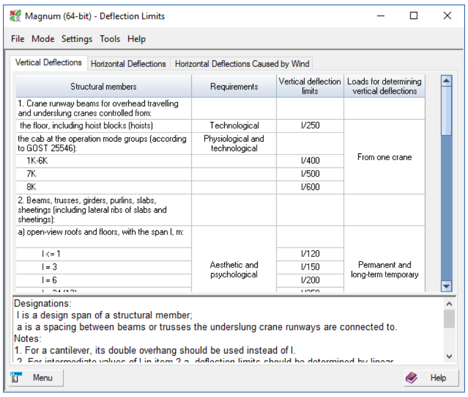


Figure 6.6.4-1. *The* **Deflection Limits** *dialog box*

The **Deflection Limits** mode provides tables from SP 20.13330 “Loads and Actions” with limitations on deflections and displacements of load-bearing structural members. This mode is not provided for the analysis according to EN 1993.

The **Deflection Limits** dialog box contains three tabs (Fig. 6.6.4 -1): **Vertical Deflections**, **Horizontal Deflections** and **Horizontal Deflections Caused by Wind**, which provide data from the tables of SP 20.13330.

6.7 Auxiliary Modes

6.7.1 Envelopes

This mode is used to solve a partial problem of determination of the basic unfavorable combinations of multiple loads acting on flexural members.

It should be noted that the temporary loads implicitly include a zero load (it enables to describe the case of absence of all temporary loads). Therefore, when calculating maximum values (e.g., bending moments) we take the greatest of the positive moments and zero, and when calculating minimum values we take the least of the negative moments and zero.

Specify the beam span in the **Envelopes** dialog box (Fig. 6.7.1-1) and click the **Apply** button. Select one of the following boundary conditions using buttons on the top right: clamped at both ends; simply supported; hinged at one end and clamped at the other; or a cantilever beam.

The application is capable of analyzing several (up to ten) patterns of loading and each load case can in its turn consist of multiple loads.

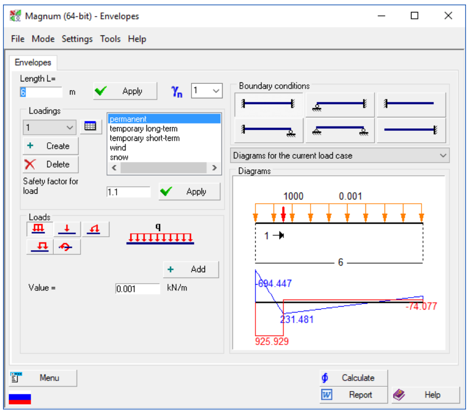


Figure 6.7.1-1. *The* **Envelopes** *dialog box*

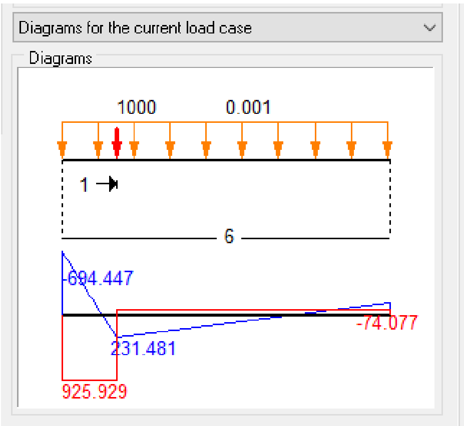


Figure 6.7.1-2. *Diagrams of moments and shear forces*

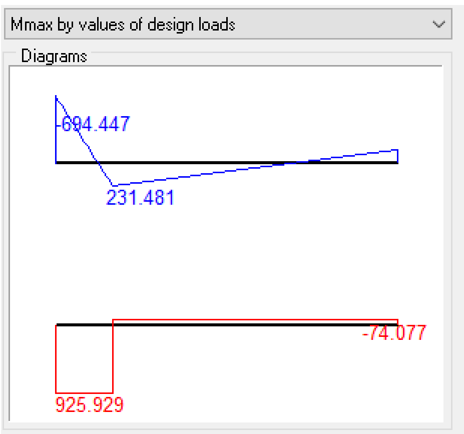


Figure 6.7.1-3. *Envelope diagrams*

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Loadings** group**;**
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors to be used with the loads of this load case in a combination of loads;
- modify, if necessary, the value of the safety factor for load;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;
- click the **Add** button.

A few load components can be specified for each load case. It is assumed that the design values of loads are entered.

Depending on the load type, its parameters may include:

- for distributed loads — the load intensity;
- for a distributed load on a part of the span — the intensity of the load, its position and width of application;
- for a concentrated force — the value of the force and its position in the span;
- for a concentrated moment — the value of the moment and its position in the span.

The **Delete** button is used to delete a load case (not a separate load included in it)**.**

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Loadings** list.

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** field with the superimposed diagrams of the bending moments and shear forces underneath (Fig. 6.7.1-2). Once you have entered all the loadings, you can view the values of the extreme moments and their corresponding shear forces, as well as the extreme shear forces and their corresponding bending moments. This can be done by selecting the respective item from the drop-down list above the **Diagrams** field (Fig. 6.7.1-3). The envelope diagrams of maximal and minimal forces are displayed separately.

If you place the mouse pointer in the diagram field, values of the moment and shear force in a particular cross-section corresponding to the position of the pointer will be displayed (Fig. 6.7.1-4).

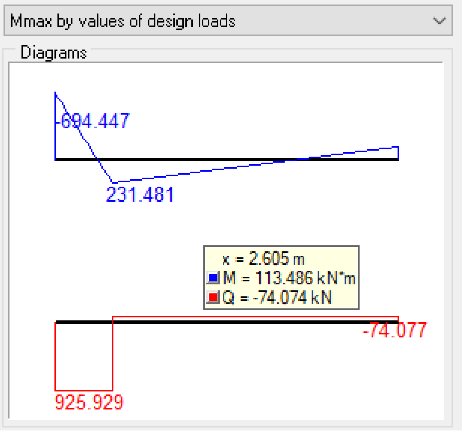


Figure 6.7.1-4. *Indication on the diagram of the moment and shear force values in a particular cross-section*

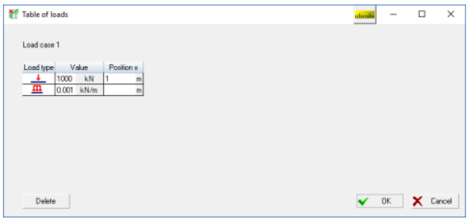
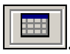


Figure 6.7.1-5. *The* **Table of loads** *dialog box*

To edit (and to delete, if necessary) particular loads from one load case, you can use a table of loads which is displayed in the respective dialog box (Fig. 6.7.1-5) after you click the button . In this table, you can modify the value of a load or its application point, and delete one or more loads. In the latter case select the icon of the load you want to delete and click the **Delete** button.

6.7.2 Influence Lines

The mode is used to plot influence lines for multi-span continuous beams of constant section. Only the influence lines of bending moments and shear forces are considered.

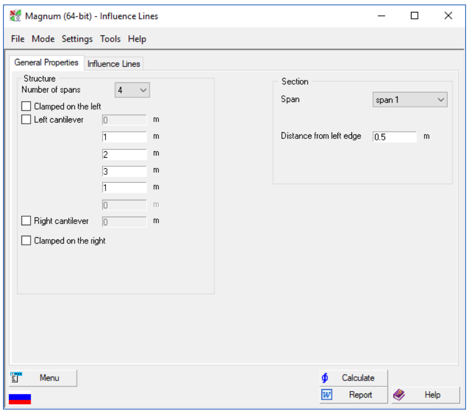


Figure 6.7.2-1. *The* **General Properties** *tab of the* **Influence Lines** *dialog box*

The design of a multi-span beam is specified in the **Structure** group of the **General Properties** tab (Fig. 6.7.2-1). It is defined by the number of spans, their length and the presence or absence of cantilevers. If you specify stiff clamping (on the right and/or on the left), the respective cantilever can no longer be defined. The **Section** group is used to select a span and specify the distance from its left edge to a section for which you want to plot the influence lines. The influence lines can be obtained by clicking the **Calculate** button or opening the **Influence Lines** tab.

The **Influence Lines** tab (see Fig. 6.7.2-2) provides the structural scheme of the beam and two influence lines (of bending moments and shear forces). If you place the mouse pointer over any point along the beam, the values of the moment and the shear force in the pointed section will be displayed.

Tables on the left from the influence lines contain information on the areas of the influence lines in segments between zero points. These values can be used to find the moment and/or the shear force under an evenly distributed temporary load.

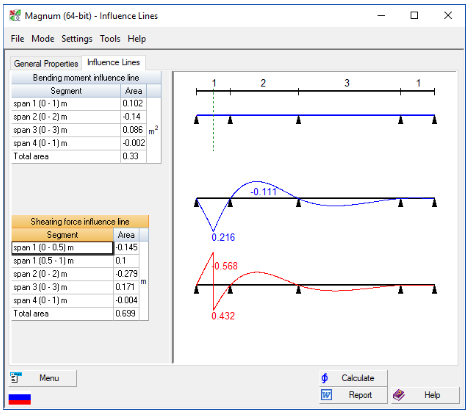


Figure 6.7.2-2. *The* **Influence Lines** *tab of the* **Influence Lines** *dialog box*

6.7.3 Geometric Properties

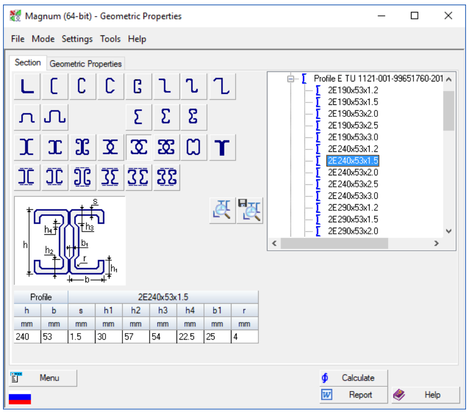


Figure 6.7.3-1. *The* **Section** *tab of the* **Geometric Properties** *dialog box*

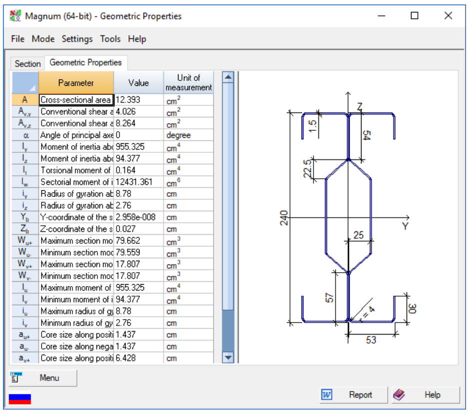


Figure 6.7.3-2. *The* **Geometric Properties** *tab of the* **Geometric Properties** *dialog box*

The **Geometric Properties** mode enables to calculate the geometric properties of cold-formed steel sections.

The initial data are entered in the **Section** tab (Fig. 6.7.3-1). The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

The calculated geometric properties of the selected or created cold-formed steel section will be displayed in the **Geometric Properties** dialog box (Fig. 6.7.3-1).

Geometric properties of sections of cold-formed bar members are calculated in this mode taking into account their rounded corners (between flanges and webs).

6.7.4 Effective Lengths

The **Effective Lengths** mode enables to calculate the effective lengths of cold-formed bar members depending on their boundary conditions in accordance with the requirements of SP 16.13330 and EN 1993-1-1. The result is the value of the effective length factor.

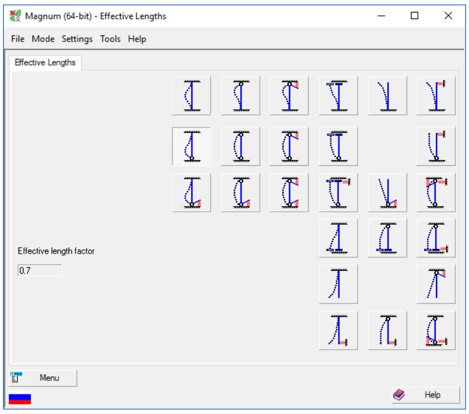


Figure 6.7.4-1. *The* **Effective Lengths** *tab of the* **Effective Lengths** *dialog box* (idealized boundary conditions are selected)

The **Effective Lengths** dialog box provides a set of possible boundary conditions of bar structural members. They are selected by clicking the respective button (Fig. 6.7.4-1), the **Effective length factor** field will display a value of this factor that corresponds to the instructions given in design codes. Only the case when the axial force is applied to the end of the bar is considered.

If the elastic supports with respect to displacements or rotations are selected (Fig. 6.7.4-2), you have to specify the numerical data on the stiffness of these supports in the respective text fields. Once you click the **Calculate** button, the **Effective length factor** field will display the calculated value of this factor in accordance with the requirements of SP 16.13330.

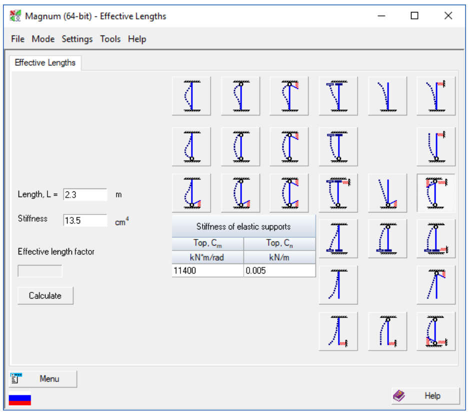


Figure 6.7.4-2. *The* **Effective Lengths** *tab of the* **Effective Lengths** *dialog box* (elastic supports are selected)

6.8 Design Modes

The modes described below are functional, used to perform checks of structural designs of steel structures from cold-formed profiles for compliance with strength, stability, and deformability requirements of design codes. The modes also enable to perform the investigation of the load-bearing capacity of cold-formed bar members by plotting the interaction curves.

The set of checks of the load-bearing capacity of cold-formed bar members depends on the type of the cross-section and the set of internal forces acting in it. The list of checks performed in **Magnum** is given in Table 6.8-1.

Table 6.8-1. The list of the load-bearing capacity checks of cold-formed bar members performed in Magnum

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
Tensile strength of the member subject to axial force N	Sec. 6.2.3 (6.5)	Sec. 6.1.2 (6.1)	Sec. 7.7.2.1 (7.68)
Compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9)	Sec. 6.1.3 (6.3)	Sec. 7.7.2.2 (7.69)
Post-buckling compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9)	Sec. 6.1.3 (6.2)	Sec. 7.7.2.2 (7.69)
Elasto-plastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.5)	not calculated
Elastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Post-buckling bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Elasto-plastic bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.5)	not calculated
Elastic bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Post-buckling bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Elasto-plastic bi-axial bending strength of the member		Sec. 6.1.4 (6.5), (6.7)	not calculated
Elastic bi-axial bending strength of the member		Sec. 6.1.4 (6.6), (6.7)	Sec. 7.7.3 (7.76)
Post-buckling bi-axial bending strength of the member		Sec. 6.1.4 (6.4), (6.7)	Sec. 7.7.3 (7.76)
Shear strength of the member subject to shear force Qz	Sec. 6.2.6 (6.17)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Shear strength of the member subject to shear force Qy	Sec. 6.2.6 (6.17)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)
Post-buckling bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Bending strength of the member subject to bending moment Mz taken into account shear force Qy	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)
Post-buckling bending strength of the member subject to bending moment Mz taken into account shear force Qy	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Strength of the member subject to combined axial force N and bending moments My, Mz		Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
Post-buckling strength of the member subject to combined axial force N and bending moments My, Mz		Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Strength of the member subject to combined axial force N, bending moments My, Mz and shear forces Qz, Qy		Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Post-buckling strength of the member subject to combined axial force N, bending moments My, Mz and shear forces Qz, Qy		Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Post-buckling strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Strength by the shear stresses in the member due to combined action of shear forces Qz, Qy, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11b)	Sec. 7.7.7 (7.83)
Strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Post-buckling strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw		Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Buckling strength of the member subject to flexural buckling relating to axes y-y due to axial force N	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes z-z due to axial force N	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes z-z due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to torsional and torsional-flexural buckling due to axial force N	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Buckling strength of the member subject to torsional and torsional-flexural buckling due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
Elasto-plastic buckling strength of the member subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	—
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Elasto-plastic buckling strength of the member subject to lateral-torsional buckling due to bending moment Mz	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	—
Buckling strength of the member subject to lateral-torsional buckling due to bending moment Mz	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subjected to axial force N and bending moments My, Mz	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz taken into account lateral-torsional buckling	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz taken into account lateral-torsional buckling (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Chord's tensile strength of the member subject to axial force N	Sec. 6.2.3 (6.5), 6.4.1 (6.69)	Sec. 6.1.2 (6.1)	Sec. 7.7.2.1 (7.68)
Chord's compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9), 6.4.1 (6.69)	Sec. 6.1.3 (6.3)	Sec. 7.7.2.2 (7.68)
Chord's post-buckling compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9), 6.4.1 (6.69)	Sec. 6.1.3 (6.2)	Sec. 7.7.2.1 (7.69)
Chord's elasto-plastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.5)	not calculated
Chord's elastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Chord's post-buckling bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Chord's shear strength of the member subject to shear force Qz	Sec. 6.2.6 (6.17), 6.4.1 (6.69)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Chord's shear strength of the member subject to shear force Qy	Sec. 6.2.6 (6.17), 6.4.1 (6.69)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Chord's bending strength of the member subject to bending moment My taken into account shear	Sec. 6.2.8 (6.29), 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
force Qz			
Chord's post-buckling bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29), 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Chord's strength of the member subject to combined axial force N and bending moment My	Sec. 6.4.1 (6.69)	Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Chord's post-buckling strength of the member subject to combined axial force N and bending moment My	Sec. 6.4.1 (6.69)	Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Chord's strength of the member subject to combined axial force N, bending moment My and shear forces Qz, Qy	Sec. 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Chord's post-buckling strength of the member subject to combined axial force N, bending moment My and shear forces Qz, Qy	Sec. 6.4.1 (6.69)	Sec. 6.1.10 Sec. (6.27)	Sec. 7.7.4 (7.77), Sec. 7.7.5 (7.78)
Chord's strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Chord's post-buckling strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Chord's strength by the shear stresses in the member due to combined action of shear forces Qz, Qy, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11b)	Sec. 7.7.7 (7.83)
Chord's strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Chord's post-buckling strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw		Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Buckling strength of the chord subject to flexural buckling relating to axes y-y due to axial force N	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes z-z due to axial force N	6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes z-z due to axial force N	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
(post-buckling behavior)			
Buckling strength of the chord subject to torsional and torsional-flexural buckling due to axial force N	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53), 6.4.1 (6.69)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Buckling strength of the chord subject to torsional and torsional-flexural buckling due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53), 6.4.1 (6.69)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Elasto-plastic buckling strength of the chord member subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	not calculated
Buckling strength of the chord subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57), 6.4.1 (6.69)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the chord subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57), 6.4.1 (6.69)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the chord subjected to axial force N and bending moment My	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My taken into account lateral-torsional buckling	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My taken into account lateral-torsional buckling (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Strength of lattice	Sec. 6.2.3 (6.5), 6.2.4 (6.9), 6.4.1 (6.70), 6.4.2.1 (6.72)	–	Sec. 7.7.11.7 (7.106), 7.7.11.8
Stability of lattic	Sec. 6.3.1 (6.46) – (6.53), 6.4.1 (6.70), 6.4.2.1 (6.71), (6.72)	–	Sec. 7.7.11.7 (7.106), 7.7.11.8
Load-carrying capacity of a batten	Sec. 6.2.5 (6.12), 6.2.6 (6.17), 6.2.8 (6.29), 6.4.1 (6.70), 6.4.3.1 (6.73), (6.74)	–	Sec. 7.7.11.7 (7.106), 7.7.11.8
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes y-y due to axial force N	–	–	Sec. 7.7.11.7 (7.87), (7.88), (7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	–	–	Sec. 7.7.8 (7.87), (7.88), (7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes z-z	–	–	Sec. 7.7.8 (7.87), Sec. (7.88),

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
due to axial force N			(7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes z-z due to axial force N (post-buckling behavior)	–	–	Sec. 7.7.8 (7.87), (7.88), (7.102) – (7.105)
Slenderness of the chord	–	–	Sec. 7.7.11.5, 7.7.11.6

The following checks are not performed:

- of the weakened sections with holes for bolts;
- of the bolted connection between the battens and the cold-formed lattice chords;
- of the local stress in the cross-section web/webs from the concentrated force or the support reaction.

Peculiarities of the implementation

1. When determining the relative eccentricity of members under compression and bending SP 260.1325800 recommends taking the design moment as the moment in the section, which is located in a particular area of the bar. This area is determined depending on the boundary conditions of the bar, on which the program has no information. Therefore, the value of the bending moment maximal along the length of the element is conservatively used.

2. The codes do not consider the problem of stability for a bar under tension and bending, but it would be unwise not to check it for stability at all, because even a relatively small tension can cause the buckling of the bar (in an elastic bar this would occur when some fibers were in compression, while in an elastoplastic bar this boundary would be harder to locate). Since design codes do not define a boundary for such a "relatively small tension", the check of stability of a bar under tension and bending is conservatively reduced to the check of the stability of in-plane bending.

3. When calculating the interaction factors k_{yy}, k_{yz}, the value of the internal bending moment is assumed to be uniformly distributed along the length of the rod. This approach provides the maximum margin of safety.

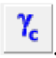
4. Interaction coefficients for checking the overall stability of structural bar elements under axial compressive force and bending moments are determined according to Annexes A (method 1) or B (method 2) of EN1993-1-1:2005. When calculating the k_{zy} coefficient, the standards distinguish between elements sensitive to torsional deformations and those not sensitive to such deformations.

When calculating the k_{zy} coefficient, the program automatically determines whether an element is sensitive to torsional deformations or to in-plane buckling based on the values of the stability coefficients χ_{LT} and χ_{TF}, which correspond to in-plane buckling and torsional-flexural buckling, respectively. If $\chi_{LT} < 1.0$ or $\chi_{TF} < 1.0$, the element is considered sensitive to torsional deformations. Additionally, the user can tick the **Not subject to torsion** checkbox to directly indicate that the element is not sensitive to torsional deformations (for example, in the case when the element is restrained by torsional constraints along its entire length).

6.8.1 Resistance of Sections

The **Resistance of Sections** mode enables to perform the analysis of the load-bearing capacity of cold-formed bar members with a given type of cross-section. In the general case, the analyzes are performed for a longitudinal force, bending moments and shear forces acting in the principal planes of inertia, as well as free torques, bimoments and constrained torques. The whole set of checks for strength and stability of load-bearing cold-formed members is implemented in compliance with the selected design code.

The **Resistance of Sections** dialog box contains the following tabs: **Materials** (Fig. 6.8.1-1, Fig. 6.8.1-2)**, Section** (Fig. 6.8.1-3), **Forces** (Fig. 6.8.1-4, Fig. 6.8.1-5), **Critical Moment** (Fig. 6.8.1-6, Fig. 6.8.1-7), **Effective Length in the XoY Plane** (Fig. 6.8.1-8), **Effective Length in the XoZ Plane**, **Interaction Curves** (Fig. 6.8.1-11). The last tab is used to analyze the results of the calculation, and the other ones are used to enter the initial data.

The **Materials** tab (Fig. 6.8.1-1) contains buttons for accessing the reference modes **Steel** –  and **Service Factor** – . Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.8.1-2). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **Materials** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.8.1-1).

You can also use the **Not subject to torsion** and **Not subject to warping** checkboxes in the **Materials** tab. The first option assumes that the element is reliably restrained by torsional constraints along its entire length and the torque will not be taken into account in the analysis (including the torsion caused by the shear force at the eccentricity of the bending center with respect to the center of mass). When the second option is selected, free warping of the support sections of the element is assumed, so the bimoment and the constrained torque will not be taken into account in the analysis.

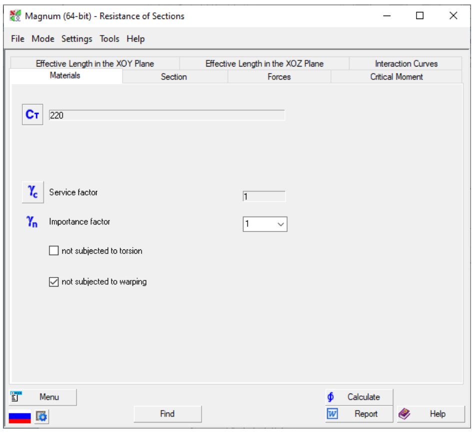


Figure 6.8.1-1. *The* **Materials** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to SP 260.1325800)

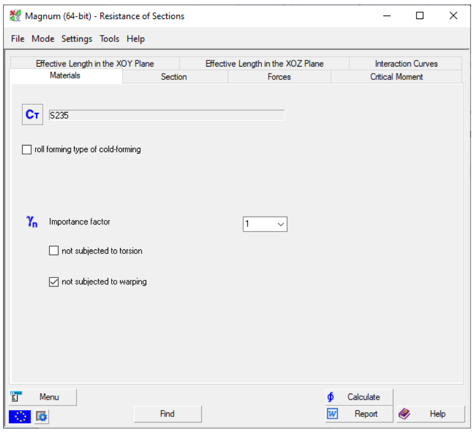


Figure 6.8.1-2. *The* **Materials** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to EN 1993)

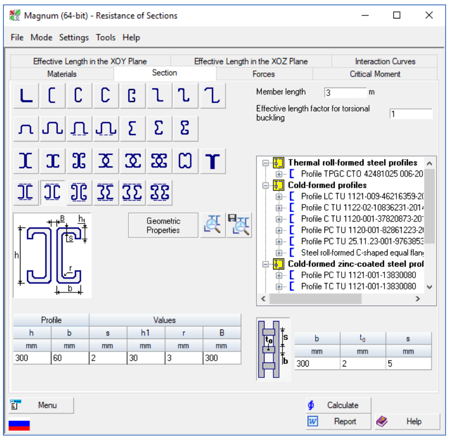


Figure 6.8.1-3. *The* **Section** *tab of the* **Resistance of Sections** *dialog box*

The **Section** tab (Fig. 6.8.1-3) contains a set of buttons, clicking which enables you to set the desired cross-section type. The user has to either select a specific cold-formed profile from the database or enter the dimensions of the section into the respective table. The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

The selected section can be saved in the **Custom Sections** catalogue the access to which is provided by the button .

The **Forces** tab (Fig. 6.8.1-4, Fig. 6.8.1-5) is used to specify forces acting in the cross-section of the bar member. In the general case, the analyzes are performed for a longitudinal force, bending moments and shear forces acting in the principal planes of inertia, as well as free torques, bimoments and constrained torques.

This tab displays a cross-section with the principal axes of inertia and the positive directions of forces. The tab contains a table for specifying the forces acting in the section from one or more load cases. The number of rows in the table corresponds to the number of the load cases and can be increased by clicking the **Add** button. To delete the selected rows, click the **Delete** button.

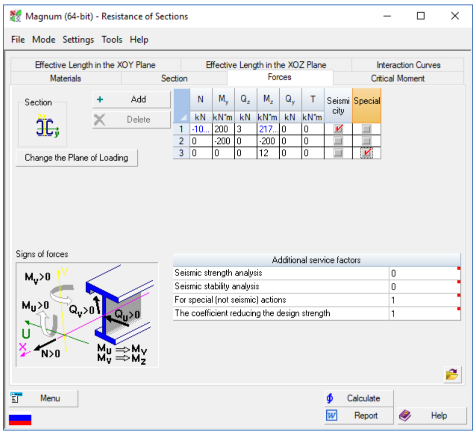


Figure 6.8.1-4. *The* **Forces** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to SP 260.1325800)

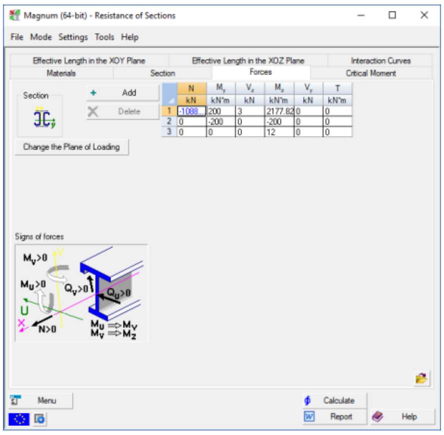


Figure 6.8.1-5. *The* **Forces** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to EN 1993)

When the analysis is performed according to SP 260.1325800, the **Seismic** checkbox can be checked for some loadings (Fig. 6.8.1-4). In this case, requirements of the respective code (selected in the main window) on the use of the additional service factor at the construction in seismic regions will be automatically taken into account. Moreover, the **Additional service factors** table will appear in this dialog box where you can specify the coefficients allowing for seismic action at the strength and stability analysis. If zero values are specified, the values are taken in accordance with the respective seismic codes by default. Having specified, for example, these two coefficients equal to 0.9, you can take into account the standard requirements for the calculation of steel structures operating in unheated rooms or in the open air at the design temperature below minus 40°C.

If the analysis is performed according to SP 260.1325800, the **Special** checkbox can be checked for some loadings (a special non-seismic loading) (Fig. 6.8.1-4). In this case you will be able to specify a service factor in accordance with Annex B of SP 296.1325800 in the **Additional service factors** table (moreover, the importance factor γ_n = 1.0 in accordance with Sec. 5.5 of SP 296.1325800 will be used for such loadings). Furthermore, an additional service factor which reduces the design strength values and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017 can be taken into account.

The table of internal forces can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of **SCAD** and can then be imported into **Magnum** by clicking the button  above the table.

To change the load plane, use the respective button. This will transfer the values of M_y and Q_z to the respective columns of the table for M_z and Q_y, and vice versa.

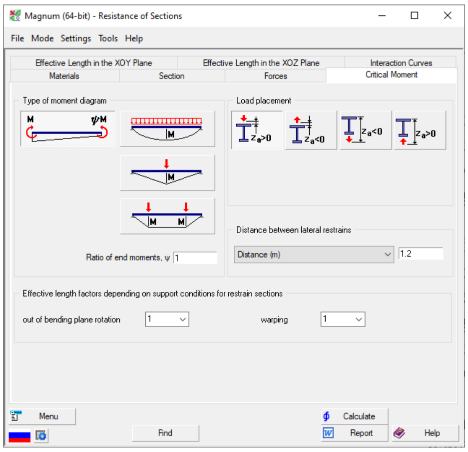


Figure 6.8.1-6. *The* **Critical Moment** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to SP 260.1325800)

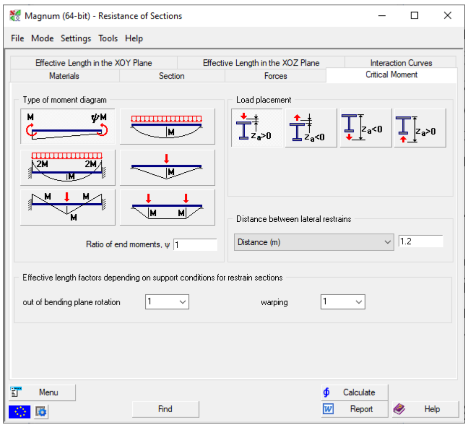


Figure 6.8.1-7. *The* **Critical Moment** *tab of the* **Resistance of Sections** *dialog box* (when the analysis is performed according to EN 1993)

The **Critical Moment** tab (Fig. 6.8.1-6, Fig. 6.8.1-7) enables to enter the initial data for the calculation of the critical bending moment which is further used in the check of the stability of in-plane bending. The calculation of the critical moment is performed using the analytical solution given in ENV 1993-1-1, depending on the character of the bending moment diagram, different boundary conditions of the bar element, restraints against warping, load application with respect to the shear center of the section, and also depending on the asymmetry of the section with respect to the axis of greater stiffness. Moreover, the calculation of the critical moment takes into account the pre-buckling curvature in accordance with [1].

The calculations are performed only for the cross-sections symmetric with respect to the axis of the lower stiffness at their bending with respect to the axis of the greater stiffness. Moreover, the codes limit the use of this check method for the sections which are characterized by a significant rotation of the principal axes of inertia of the “effective” section with respect to the principal axes of inertia of the gross cross-section.

The respective limitations are introduced in the software as well. Thus, if the user has selected a section which is not symmetric with respect to the axis of the lower stiffness at its bending with respect to the axis of the greater stiffness, it is assumed that the stability of in-plane bending of the considered bar member is ensured by the appropriate restraints.

In addition, to calculate the critical moment, two effective length factors should be specified. The factor k refers to end rotation on plan and the factor k_w refers to end warping. Recommendations for assigning the coefficients k, k_w for some special cases can be found in the books

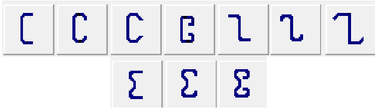
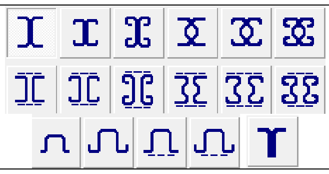
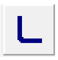
- L. Gardner, *Stability of Steel Beams and Columns: In Accordance with Eurocodes and the UK National Annexes*, Steel Construction Institute, 2011.
- N.Boissonnade, R.Greiner, J.P.Jaspart, J.Lindner, *Rules for member stability in EN 1993-1-1 : Background documentation and design guidelines*, ECCS European Convention for Constructional Steelwork, 2006.

The buttons of the **Type of moment diagram** group enable to specify a bending moment diagram type for a bar member. Since SP 260.1325800 provides guidelines for determining the critical moment for only four types of bending moment diagrams, when the analysis is performed according to SP 260.1325800, this group contains only the respective four buttons.

If the first option is selected, you should also specify the parameter ψ, which characterizes the ratio of the end moments. The effective length factor for the lateral-torsional buckling has to be specified in the **l/L for a compressed chord** field. Moreover, the distance from the load application point to the top chord level, z_a, should be specified. The sign of z_a is assigned according to a schematic given in this tab.

The lateral-torsional buckling analysis depends on the cross-section type of the bar member (see Table 6.8.1-2).

Table 6.8.1-2. The lateral-torsional buckling analysis for bar members with various cross-section types

Cross-section type of the bar member	A response of the **Critical Moment** mode to the section type
	It is assumed that the stability of in-plane bending is provided by the respective restraints of the compressed chord of the flexural member
	The lateral-torsional buckling analysis is performed
	The codes do not provide any recommendations on the determination of the critical moment and on the lateral-torsional buckling analysis
	It is assumed that lateral-torsional buckling does not occur

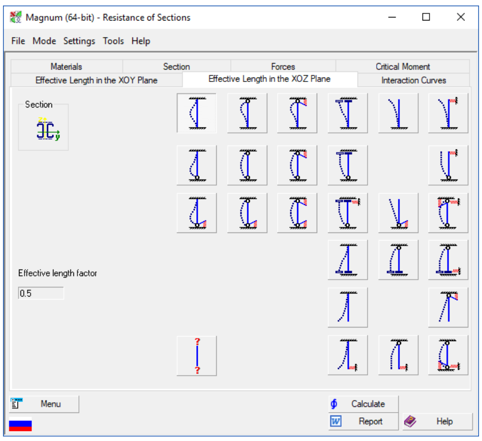
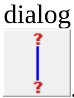


Figure 6.8.1-8. *The* **Effective Length in the XoY Plane** *tab of the* **Resistance of Sections** *dialog box*

The **Effective Length in the XoY (XoZ) Plane** tabs (Fig. 6.8.1-8) are the exact replicas of the **Effective Lengths** tab from the **Effective Lengths** mode, and they suggest a set of possible conditions of end support in the respective planes of loading for a compressed bar member, which differ from one another in combinations of the boundary conditions (free end, hinge, elastic support, elastic clamping, clamped).

If the boundary conditions include an elastic clamping or an elastic support, a table for entering the data on the stiffness of the respective restraint will appear.

Factors $k = l / L$ describe the ratio of the effective length, l, to the geometric length of the bar, L**,** in different planes. They are displayed in the respective **Effective length factor** fields.

This dialog box is described in detail in Section 6.7.4 Unlike the **Effective Lengths** mode, this dialog has the **Effective length factor specified by user** button – . Clicking this button will enable you to enter any desired values for the effective length factor and confirm your choice by clicking the **Apply** button. The specified value will then be displayed in the **Effective length factor** field. In all other cases this field is inaccessible.

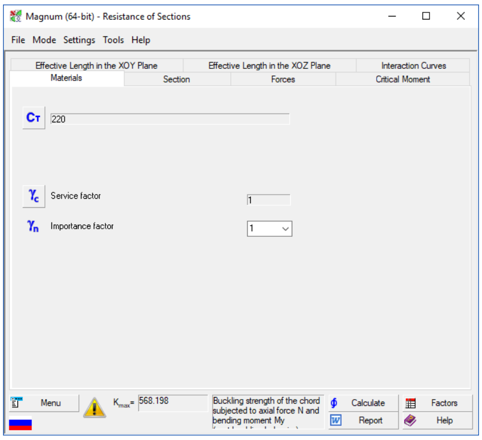


Figure 6.8.1-9. *The main window of the* **Resistance of Sections** *mode* (when the analysis is performed according to SP 260.1325800)

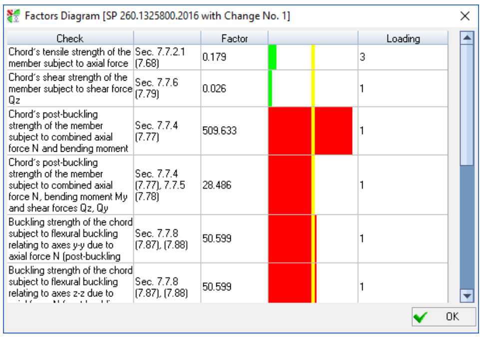


Figure 6.8.1-10. *The* **Factors Diagram** *dialog box*

Once you have entered the initial data, you can click the **Calculate** button, and the $\mathbf{K_{max}}$ field located at the bottom of the dialog will display the maximum value of utilization factor of restrictions and the type of the standard check (strength, stability, etc.) for which this maximum value was obtained (Fig. 6.8.1-9). You can also browse interactively the values of all the other utilization factors of restrictions. To do it, use the **Factors** button which becomes available once the analysis is completed. The **Factors Diagram** dialog box (Fig. 6.8.1-10) displays the respective factors numerically and graphically.

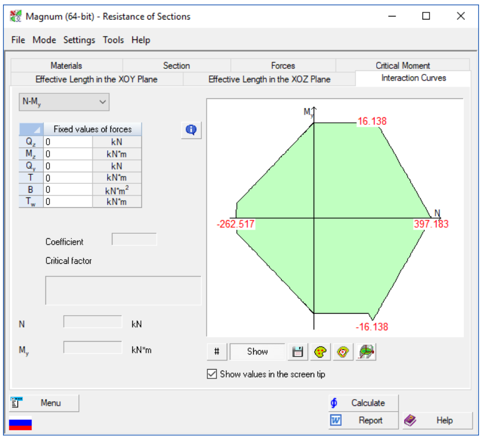


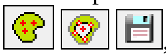
Figure 6.8.1-11. *The* **Interaction Curves** *tab of the* **Resistance of Sections** *dialog box*

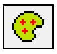
The curves enclosing an area of the section load-bearing capacity under various pairs of forces which can arise in the considered section are plotted in the **Interaction Curves** tab (Fig. 6.8.1-11). Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of forces, and clicking the button displays a grid in the display field. The curves surround the coordinate origin by a closed line inside which there are points with conditionally acceptable pairs of the considered forces. A pair of forces is deemed acceptable when $K_{max} \leq 1$. All other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the forces variation shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the acting forces; their values are displayed in the respective fields.

The dialog also displays the maximum value of the utilization factor of restrictions that corresponds to these forces and the type of check in which it takes place. When the pointer is placed over a point where $K_{max} > 1$, a warning sign is displayed .

Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the position of the pointer in the plot area of the interaction curve.

The dialog box also contains three buttons: , which enable to perform the following operations:

 — if the forces are specified, clicking this button will draw the points the coordinates of which in the area of the load-bearing capacity correspond to these forces;

 — drawing a convex hull of the points specified above, i.e. an entire set of points which may result from a linear combination of specified forces, including their incomplete values;

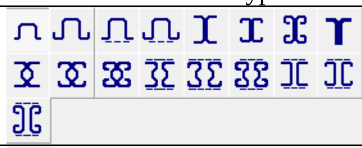
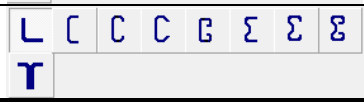
 — saving the forces that can lead to K_{max}=1 in a text file (this file can be imported into other programs for further analysis).

6.9 Design of Cold-Formed Steel Members

6.9.1 Trusses

This mode enables to perform all the necessary strength and stability checks of cold-formed truss members. The **Trusses** mode enables to perform the analysis of hinged bar roof structures commonly used in self-framing metal buildings using compound or lattice sections from U-, C- and Σ-shaped cold-formed profiles, as well as hat channels and compound sections from double cold-formed angles (see Table 6.9.1-1). The design values of internal forces and moments in the truss members are calculated automatically depending on the given vertical external loads. If the user has selected a section of a load-bearing member from an assortment of cold-formed profiles, this mode also enables to find a cold-formed profile from the assortment.

Table 6.9.1-1. Types of cross-sections of the cold-formed truss members available in the Trusses mode

Truss member	Cross-section type
Top chord Bottom chord	
Support vertical / Vertical Support diagonal / Diagonal	

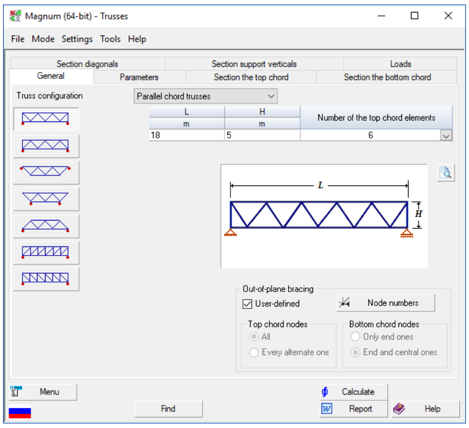


Figure 6.9.1-1. *The* **General** *tab of the* **Trusses** *dialog box*

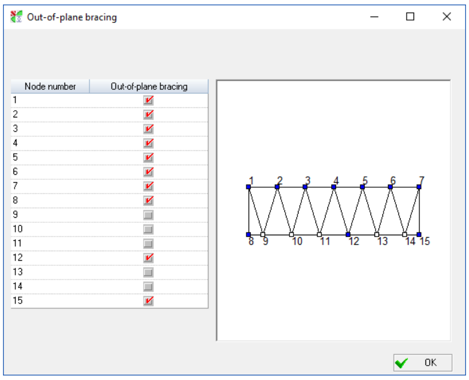


Figure 6.9.1-2. *The* **Out-of-plane bracing** *dialog box*

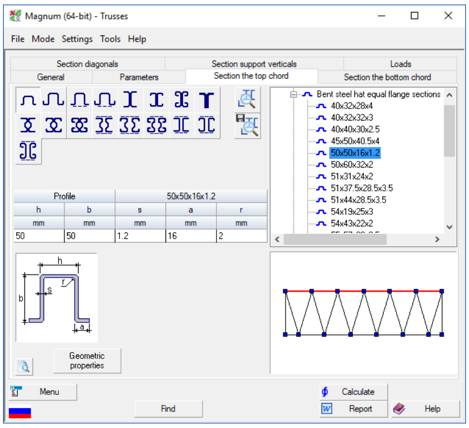


Figure 6.9.1-3. *The* **Top Chord Section** *tab of the* **Trusses** *dialog box*

The **Trusses** dialog box contains the following tabs: **General** (Fig. 6.9.1-1), **Top Chord Section** (Fig. 6.9.1-3), **Bottom Chord Section** (Fig. 6.9.1-4), **Vertical Section, Support Vertical Section** (Fig. 6.9.1-5), **Diagonal Section** (Fig. 6.9.1-6), **Support Diagonal Section**, **Parameters** (Fig. 6.9.1-7, Fig. 6.9.1-8) and **Loads** (Fig. 6.9.1-9).

The **General** tab (Fig. 6.9.1-1) contains a drop-down list for selecting the truss type by its chord shape and a group of buttons for selecting the truss configuration. The following types of trusses can be analyzed: parallel chord, triangular, trapezoid, with a polygonal top chord, mono-pitched and dual-pitched. All trusses considered in this mode are statically determinate and are assumed to be fixed in the support nodes in a statically determinate way according to a beam scheme.

You also have to specify the geometric parameters for the selected truss configuration in the **General** tab (the span of the truss, its height on the support, its height in the middle of the span, etc.). Specify the span of the truss and its height on the support for the selected configuration. Additional geometric parameters have to be specified in case of a trapezoid truss.

The respective radio buttons and checkboxes are used to specify the method of out-of-plane bracing of the top and bottom chord nodes (the bracing in the truss plane is assumed to be statically determinate: a hinge support for the left support node and a roller support for the right one). When the **User-defined** checkbox is checked, the **Node numbers** button becomes accessible. Clicking this button invokes the **Out-of-plane bracing** dialog box (Fig. 6.9.1-2). The dialog provides a design model of the truss with numbered nodes and a table where each truss node is assigned a checkbox. A checked checkbox means that there is a bracing in the given node. The braced nodes are highlighted in blue in the model. Nodes in grey are braced by default and their condition cannot be modified.

The **Top Chord Section** (Fig. 6.9.1-3) and **Bottom Chord Section** tabs (Fig. 6.9.1-4) are used to assign cross-sections to truss members of the top and bottom chords respectively. It is assumed that the chord sections do not vary along the truss length. A cold-formed hat channel (with or without battens and single or double edge fold stiffeners) can be selected as a cross-section for the top and bottom chords of the truss. The available cross-sections also include compound cold-formed sections made from double equal or unequal angles arranged as a tee, as well as compound and lattice cold-formed sections from U-, C- and Σ- shaped sections with or without edge fold stiffeners (single or double).

When a cold-formed profile is selected from the database, all its sizes will be provided in the table above the displayed section (Fig. 6.9.1-3). The user can edit these sizes when necessary.

Text fields for specifying the parameters of the battens are provided for the lattice cold-formed steel sections with chords

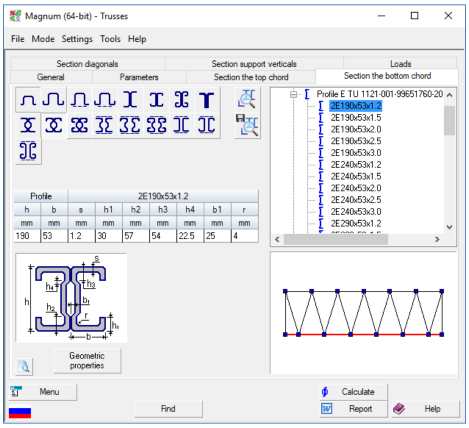


Figure 6.9.1-4. *The* **Bottom Chord Section** *tab of the* **Trusses** *dialog box*

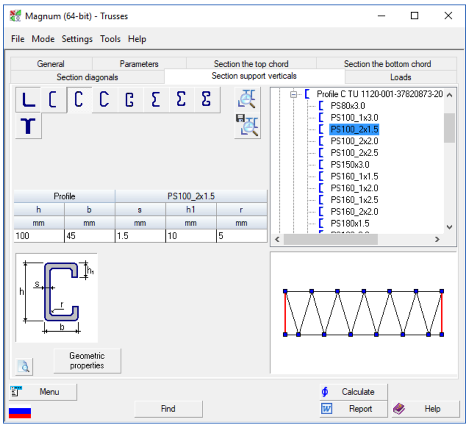


Figure 6.9.1-5. *The* **Support Vertical Section** *tab of the* **Trusses** *dialog box*

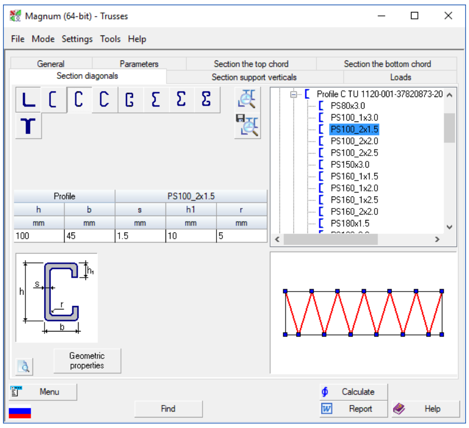


Figure 6.9.1-6. *The* **Diagonal Section** *tab of the* **Trusses** *dialog box*

interconnected through the battens (Fig. 6.9.1-4).

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

The **Diagonal Section** (Fig. 6.9.1-6) and **Support Vertical Section** tabs (Fig. 6.9.1-5) are used to assign cross-sections to diagonals and support verticals of the truss respectively. The available cross-sections include U-, C- and Σ- shaped cold-formed sections with or without edge fold stiffeners (single or double). Sections of the lattice members of the truss can be also made from single cold-formed equal or unequal angles, and from double cold-formed equal or unequal angles arranged as a tee.

When a cold-formed profile is selected from the database, all its sizes will be provided in the table above the displayed section (Fig. 6.9.1-5, Fig. 6.9.1-6). The user can edit these sizes when necessary.

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

The **Parameters** tab (Fig. 6.9.1-7, Fig. 6.9.1-8) enables to specify the steel for the load-bearing truss members (clicking the button C$_T$ invokes the **Steel** mode) and the service factor (clicking the button γ_c invokes the **Service Factors** mode). Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.9.1-8). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **Parameters** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.9.1-7).

The **Steel** and **Service Factors** modes are described in detail in Sections 6.6.1 and 6.6.3 respectively.

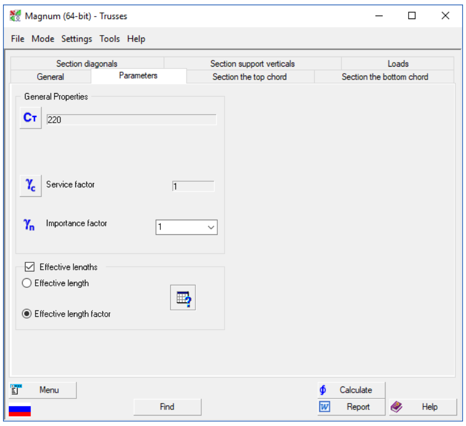


Figure 6.9.1-7. *The* **Parameters** *tab of the* **Trusses** *dialog box* (when the analysis is performed according to SP 260.1325800)

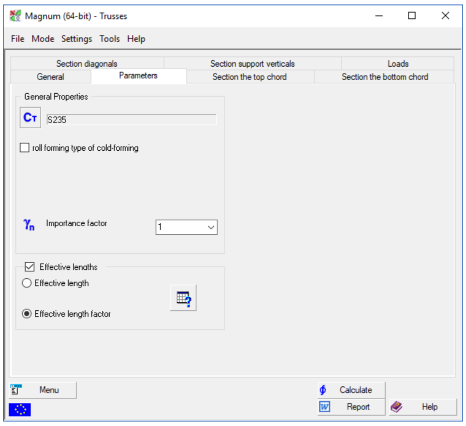


Figure 6.9.1-8. *The* **Parameters** *tab of the* **Trusses** *dialog box* (when the analysis is performed according to EN 1993)

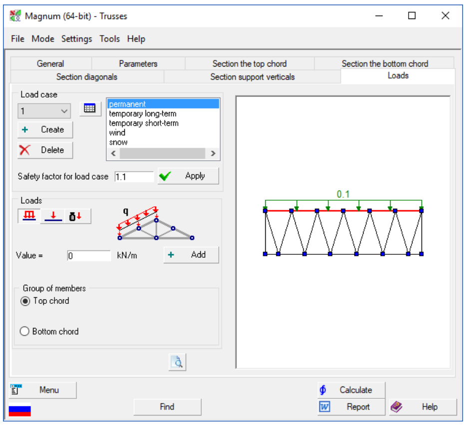


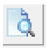
Figure 6.9.1-9. *The* **Loads** *tab of the* **Trusses** *dialog box*

The **Loads** tab (Fig. 6.9.1-9) is much similar to that described in the **Envelopes** mode (see Section 4.8.1). However, there are some differences. The user can specify either a uniformly distributed load or a concentrated load upon nodes of the top or bottom truss chord. The distributed load is applied to either the whole chord (top or bottom) or to the half of it. The load application area is defined using the respective radio buttons in the **Group of elements** group. Number of the load application node is selected from the **Node number** drop-down list.

The graphical control of the input data can be performed in the special window located in the **Loads** tab on the right and displaying a truss model. Once you click the button to select the chord the uniformly distributed load is applied to, the respective chord will be highlighted in red in this window, and when you select the number of the node the concentrated load is applied to, the respective node will be highlighted in red.

After clicking the **Add** button, a schematic of the respective load case with all the specified loads will appear in the window.

To edit the values of particular loads, you can use a table invoked by the button  (see Section 4.8.1).

Clicking the **Forces in the truss members for the current load case** button  invokes an information window displaying a design model of the truss with a diagram of internal forces for the current load case (Fig. 6.9.1-10).

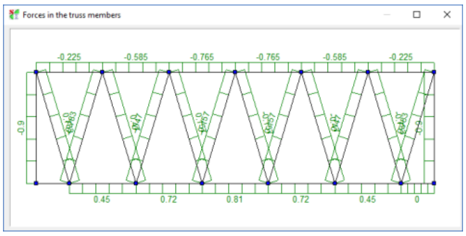


Figure 6.9.1-10. *The* **Forces in truss members** *information window*

Magnum (unlike SCAD) assumes that a truss is subjected to a nodal load. Thus the specified distributed load is not transferred onto the truss members; instead, it is assumed to be applied to some enclosing roof structure which performs the function of transferring the load onto the truss nodes.

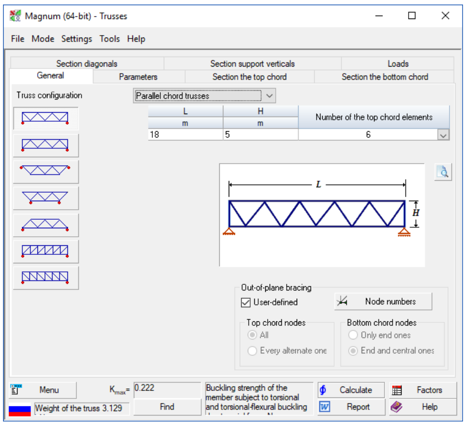


Figure 6.9.1-11. *The main window of the* **Trusses** *mode*

Clicking the **Calculate** button in the main window of the **Trusses** mode (Fig. 6.9.1-11) will display the value of $\mathbf{K_{max}}$ and indicate the type of check (strength, stability…), in which the maximum takes place. You can also browse all the other utilization factors of restrictions by clicking the **Factors** button in the **Factors Diagram** dialog box (Fig. 6.9.1-12).

The **Find** button in the main window of the **Trusses** mode enables you to switch to a mode that performs the purposeful search of cross-sections for the truss members and selects an optimal (by the criterion of minimum cross-sectional area) cold-formed profile that meets all the requirements of the codes.

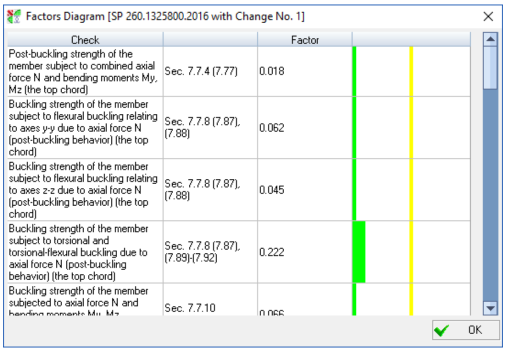


Figure 6.9.1-12. *The* **Factors Diagram** *dialog box*

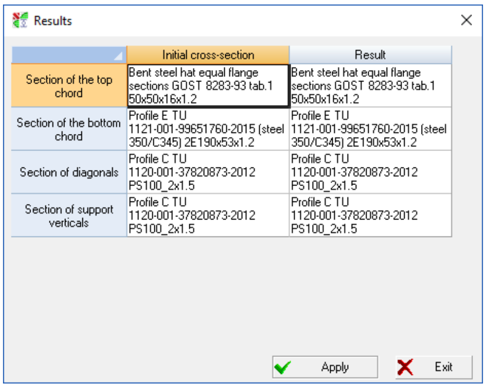


Figure 6.9.1-13. *The* **Results** *dialog box*

The searches through different groups of members (the top and bottom chords, diagonals, verticals) are mutually independent.

When the search is completed, a dialog with the recommendations on the selection of cross-sections for various truss members appears on the screen (Fig. 6.9.1-13). Clicking the **Apply** button will export all the recommended sections as the initial data to perform the check analysis of the new truss structure with the new sections.

If the maximum section of the assortment was used in the search process and the value was still $K > 1$, the dialog will display the respective message and the **Apply** button will become inactive.

Clicking the **Report** button will generate a document with all the initial data for the analysis (a design model of the truss, cross-sections of its members, load types and values), as well as the results of the static analysis of the truss, the determined unfavorable load combinations and the results of the load-bearing capacity checks of the truss members in accordance with the design codes selected by the user.

6.9.2 Beams

This mode enables to perform all the necessary strength and stability checks of cold-formed beams. The **Beams** mode enables to perform the analysis of beams commonly used in self-framing metal buildings using compound or lattice sections from U-, C- and Σ-shaped cold-formed profiles. The design values of internal forces and moments (bending moment and shear force) in the beam sections are calculated automatically depending on the given external loads. If the user has selected a section of a load-bearing member from an assortment of cold-formed profiles, this mode also enables to find a cold-formed profile from the assortment.

The **Beams** dialog box contains the following tabs: **General Properties** (Fig. 6.9.2-1, Fig. 6.9.2-2), **Section** (Fig. 6.9.2-3), **Restraints** (Fig. 6.9.2-4) and **Loads** (Fig. 6.9.2-5).

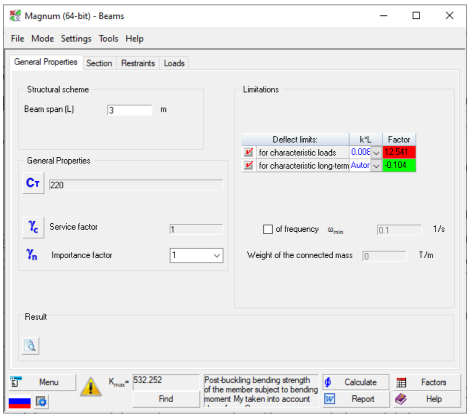


Figure 6.9.2-1. *The* **General Properties** *tab of the* **Beams** *dialog box* (when the analysis is performed according to SP 260.1325800)

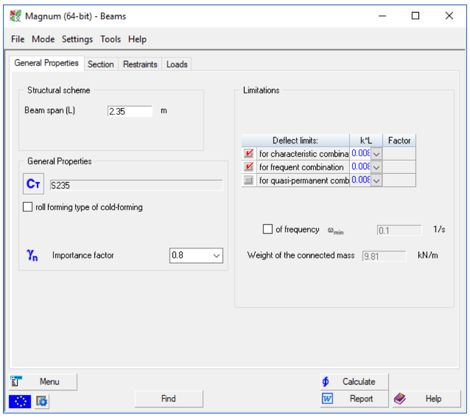


Figure 6.9.2-2. *The* **General Properties** *tab of the* **Beams** *dialog box* (when the analysis is performed according to EN 1993)

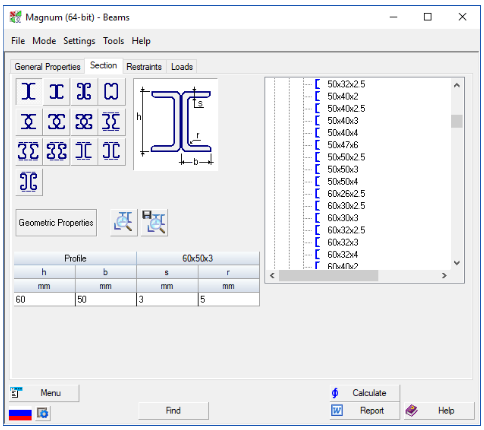


Figure 6.9.2-3. *The* **Section** *tab of the* **Beams** *dialog box*

The **General Properties** tab (Fig. 6.9.2-1, Fig. 6.9.2-2) enables to specify the span of the beam in the **Structural scheme** group, as well as the steel for the beam (clicking the button invokes the **Steel** mode) and the service factor (clicking the button invokes the **Service Factors** mode) in the **General Properties** group. Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.9.2-2). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **General Properties** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.9.2-1).

The **Steel** and **Service Factors** modes are described in detail in Sections 6.6.1 and 6.6.3 respectively.

You can specify limitations of the absolute deflection value or of the natural oscillation frequency in the **Limitations** group. In the latter case you can specify the weight of the connected mass which will be added to the own mass of the structure. The deflection limit for the beam is specified in fractions of its span length (it will be compared with the relative deflection under the loads with design values which correspond to the serviceability limit state).

Note that no special factor is introduced for the deflection check in **Magnum** — the application only calculates and outputs the maximum deflection value.

The **Section** tab (Fig. 6.9.2-3) is used to assign a cross-section to the beam.

It is assumed that the sections do not vary along the beam length. The available cross-sections include compound and lattice cold-formed sections from U-, C- and Σ- shaped sections with or without edge fold stiffeners (single or double).

When a cold-formed profile is selected from the database, all its sizes will be provided in the table under the displayed section (Fig. 6.9.2-3). The user can edit these sizes when necessary.

Text fields for specifying the parameters of the battens are provided for the lattice cold-formed steel sections with chords interconnected through the battens (Fig. 6.9.2-3).

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

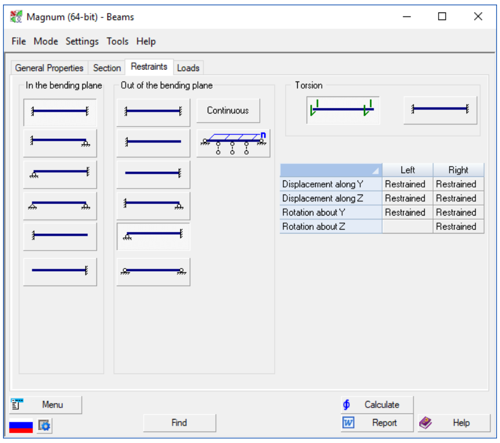


Figure 6.9.2-4. *The* **Restraints** *tab of the* **Beams** *dialog box*

The **Restraints** tab (Fig. 6.9.2-4) contains three groups of buttons for specifying a system of beam restraints in and out of the bending plane and torsional restraints. The selection within each group is performed independently by clicking a respective button. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

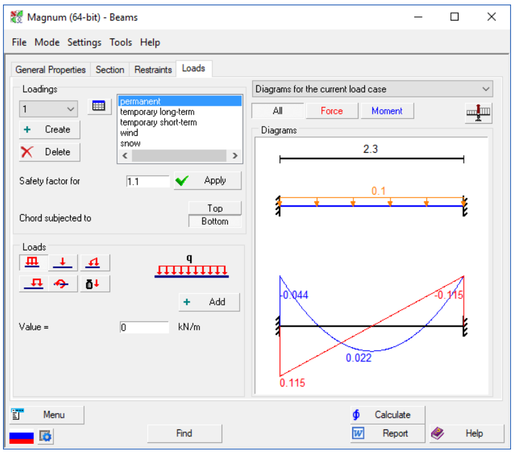


Figure 6.9.2-5. *The* **Loads** *tab of the* **Beams** *dialog box*

The **Loads** tab (Fig. 6.9.2-5) is used to specify the loads acting on the beam. This tab is nearly identical to that from Section 4.8.1. The difference is that an application area (top or bottom chord) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case.

In Beam mode verification of the deflection has been implemented for the beams subjected to one load case combination only. It should be noted that the problem of determining the stress-strain state of a thin-walled bar made from cold-formed profile is nonlinear, since it considers the postbuckling behavior, and therefore the superposition principle is inapplicable in this case, i.e. the deflections caused by various static actions cannot be summed up when combining these actions.

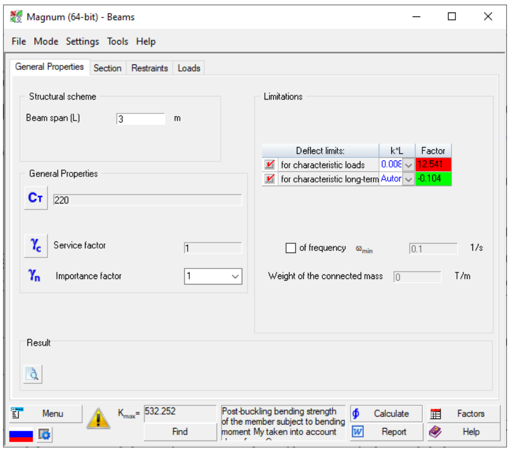


Figure 6.9.2-6. *The main window of the* **Beams** *mode*

The **Factors** and **Find** buttons (the latter is available only for beams from cold-formed steel profiles from an assortment) enable you to analyze the calculation results or to perform the search of cross-sections for the beam and select an optimal (by the criterion of minimum cross-sectional area) cold-formed profile that meets all the requirements of the codes. These modes are described in Section 4.10.1 (Fig. 6.9.2-6).

It should be noted that the search of sections will be based only on the strength and stability conditions.

Clicking the **Report** button (Fig. 6.9.2-6) will generate a document with all the initial data for the analysis (a design model of the beam, its cross-section with geometric properties, load types and values), as well as the results of the static analysis of the beam including envelope diagrams and the results of the load-bearing capacity checks of the beam in accordance with the design codes selected by the user.

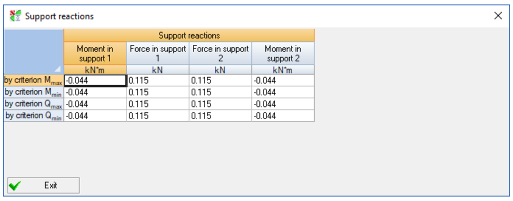
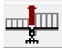


Figure 6.9.2-7. *The* **Support reactions** *dialog box*

Moreover, the report document will contain a table with design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 6.9.2-7), which can be invoked by clicking the respective button  in the **Loads** tab.

6.9.3 Continuous Beams

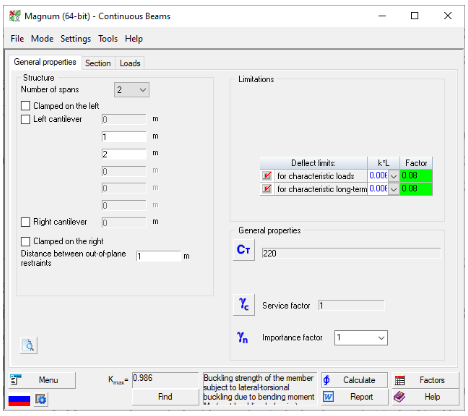


Figure 6.9.3-1. *The* **General Properties** *tab of the* **Continuous Beams** *dialog box* (when the analysis is performed according to SP 260.1325800)

This mode enables to perform all the necessary strength and stability checks of cold-formed continuous beams. The **Continuous Beams** mode enables to perform the analysis of continuous beams commonly used in self-framing metal buildings using compound or lattice sections from U-, C- and Σ-shaped cold-formed profiles. The design values of internal forces and moments (bending moment and shear force) in the sections of the continuous beams are calculated automatically depending on the given external loads. If the user has selected a section of a load-bearing member from an assortment of cold-formed profiles, this mode also enables to find a cold-formed profile from the assortment.

The **Continuous Beams** dialog box contains the following tabs: **General Properties** (Fig. 6.9.3-1, Fig. 6.9.3-2), **Section** (Fig. 6.9.3-3) and **Loads** (Fig. 6.9.3-4).

The design of a multi-span beam is specified in the **Structure** group of the **General Properties** tab (Fig. 6.9.3-1, Fig. 6.9.3-2). It is defined by the number of spans, their length and the presence or absence of cantilevers.

The **Clamped on the left** and **Clamped on the right** checkboxes are used to model the stiff clamping on the ends of the continuous beam. If you specify stiff clamping (on the right and/or on the left), the respective cantilever can no longer be defined.

This group also contains a field for specifying the spacing

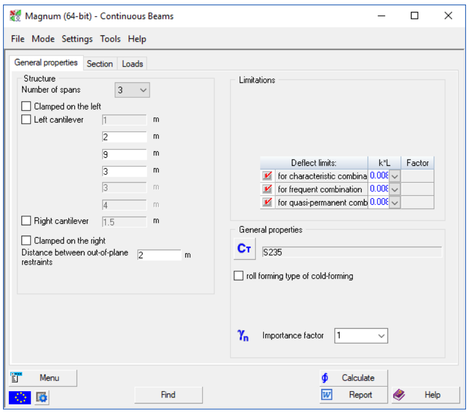


Figure 6.9.3-2. *The* **General Properties** *tab of the* **Continuous Beams** *dialog box* (when the analysis is performed according to EN 1993)

between out-of-plane restraints of the beam compressed chord. It is assumed that this spacing is uniform over the whole beam length and that there are such restraints on all supports.

The **General Properties** group in this tab enables to specify the steel for the continuous beam (clicking the button invokes the **Steel** mode) and the service factor (clicking the button invokes the **Service Factors** mode). Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.9.3-2). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **General Properties** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.9.3-1).

The **Steel** and **Service Factors** modes are described in detail in Sections 6.6.1 and 6.6.3 respectively.

You can specify limitations of the relative deflection value in the **Limitations** group in the **General Properties** tab (Fig. 6.9.3-1, Fig. 6.9.3-2). The deflection limit for the continuous beam is specified in fractions of its span length. It will be compared with the actual deflection of the beam under the loads with design values which correspond to the serviceability limit state.

Note that no special factor is introduced for the deflection check in **Magnum** — the application only calculates and outputs the maximum deflection value.

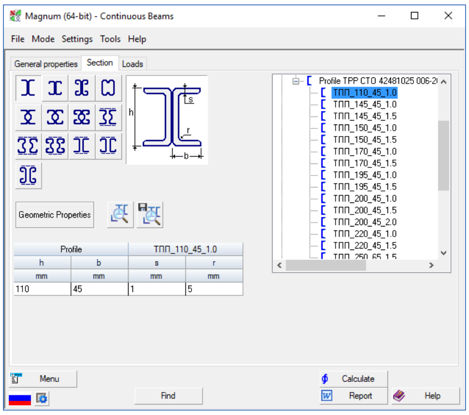


Figure 6.9.3-3. *The* **Section** *tab of the* **Continuous Beams** *dialog box*

The **Section** tab (Fig. 6.9.3-3) is used to assign a cross-section to the continuous beam.

It is assumed that the sections do not vary along the beam length. The available cross-sections include compound and lattice cold-formed sections from U-, C- and Σ- shaped sections with or without edge fold stiffeners (single or double).

When a cold-formed profile is selected from the database, all its sizes will be provided in the table under the displayed section (Fig. 6.9.3-3). The user can edit these sizes when necessary.

Text fields for specifying the parameters of the battens are provided for the lattice cold-formed steel sections with chords interconnected through the battens (Fig. 6.9.3-3).

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

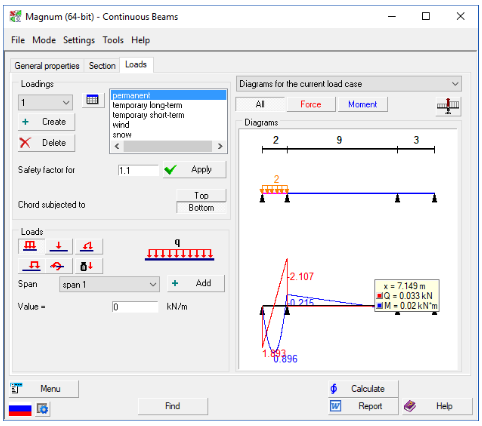


Figure 6.9.3-4. *The* **Loads** *tab of the* **Continuous Beams** *dialog box*

The **Loads** tab (Fig. 6.9.3-4) is used to specify the loads acting on the considered continuous beam.

This tab is nearly identical to that from Sections 4.8.1 and 4.10.3. The difference is that an application area (top or bottom chord) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case. Moreover, the loads are specified for each span. The number of a span (or a cantilever) to which a load is applied is selected from the **Span** drop-down list. The position of the concentrated force is measured from the left edge of the span.

In Continuous Beam mode verification of the deflection has been implemented for the continuous beams subjected to one load case combination only. It should be noted that the problem of determining the stress-strain state of a thin-walled bar made from cold-formed profile is nonlinear, since it considers the postbuckling behavior, and therefore the superposition principle is inapplicable in this case, i.e. the deflections caused by various static actions cannot be summed up when combining these actions.

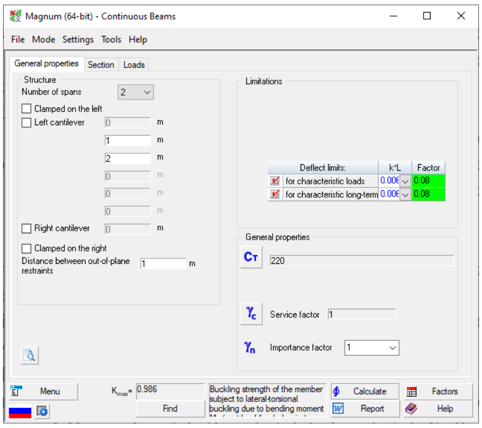


Figure 6.9.3-5. *The main window of the* **Continuous Beams** *mode*

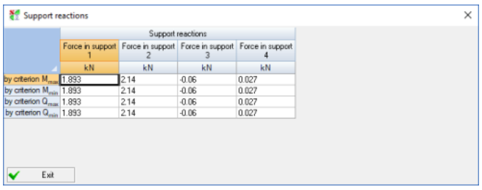
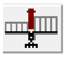


Figure 6.9.3-6. *The* **Support reactions** *dialog box*

When the analysis is completed, the **Factors** and **Find** buttons appear (the latter is available only for continuous beams from cold-formed steel profiles from an assortment) and enable you to analyze the calculation results or to perform the search of cross-sections for the continuous beam and select an optimal (by the criterion of minimum cross-sectional area) cold-formed profile that meets all the requirements of the codes. These modes are described in Section 4.10.1 (Fig. 6.9.3-5).

It should be noted that the search of sections for a continuous beam will be based only on the strength and stability conditions.

Clicking the **Report** button (Fig. 6.9.3-5) will generate a document with all the initial data for the analysis (a design model of the continuous beam, its cross-section with geometric properties, load types and values), as well as the results of the static analysis of the continuous beam including envelope diagrams and the results of the load-bearing capacity checks of the beam in accordance with the design codes selected by the user.

Moreover, the report document will contain a table with design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 6.9.3-6), which can be invoked by clicking the respective button  in the **Loads** tab.

6.9.4 Columns

The **Columns** mode enables to perform the load-bearing capacity checks of columns and posts of compound and lattice sections from U-, C- and Σ-shaped cold-formed profiles. The whole set of checks for strength and stability is implemented in compliance with the codes. A planar loading is assumed with a longitudinal force, bending moment and shear force acting in the sections of the column.

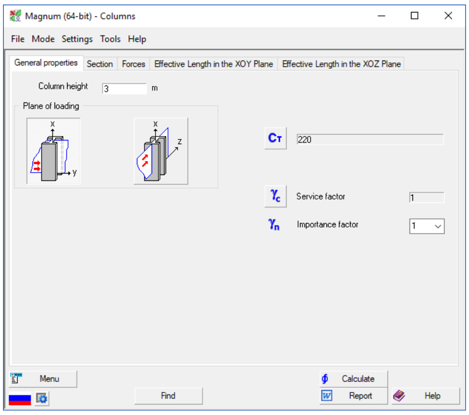


Figure 6.9.4-1. *The* **General Properties** *tab of the* **Columns** *dialog box* (when the analysis is performed according to SP 260.1325800)

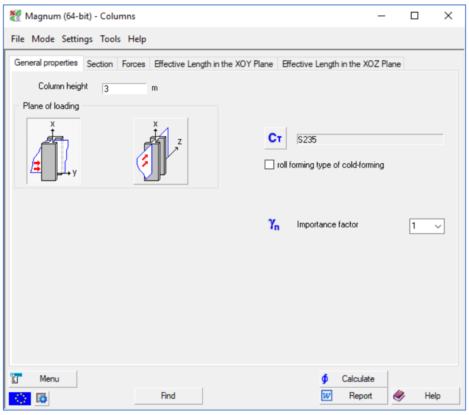


Figure 6.9.4-2. *The* **General Properties** *tab of the* **Columns** *dialog box* (when the analysis is performed according to EN 1993)

The **Columns** dialog box contains the following tabs: **General Properties** (Fig. 6.9.4-1, 6.9.4-2), **Section** (Fig. 6.9.4-3), **Forces** (Fig. 6.9.4-4), **Effective Length in the XOY Plane** (Fig. 6.9.4-6), **Effective Length in the XOZ Plane**.

The **General Properties** tab (Fig. 6.9.4-1, 6.9.4-2) contains a text field for entering the column height and two buttons for selecting the load plane (an orientation of the deformation plane). For frame structures, the load plane is defined by the way a column is incorporated into a planar frame.

This tab also enables to specify the steel for the column (clicking the button Cт invokes the **Steel** mode) and the service factor (clicking the button γ_c invokes the **Service Factors** mode). Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.9.4-2). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **General Properties** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.9.4-1).

The **Steel** and **Service Factors** modes are described in detail in Sections 6.6.1 and 6.6.3 respectively.

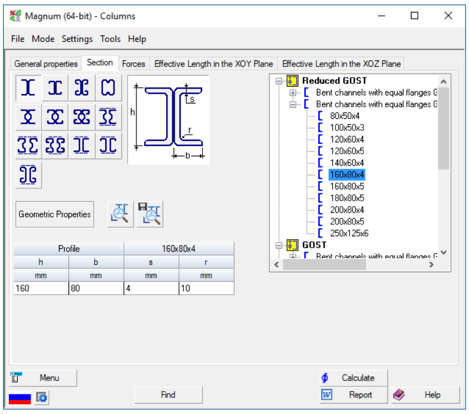


Figure 6.9.4-3. *The* **Section** *tab of the* **Columns** *dialog box*

The **Section** tab (Fig. 6.9.4-3) is used to assign a cross-section to the column. The available cross-sections include compound and lattice cold-formed sections from U-, C- and Σ- shaped sections with or without edge fold stiffeners (single or double).

When a cold-formed profile is selected from the database, all its sizes will be provided in the table under the displayed section (Fig. 6.9.4-3). The user can edit these sizes when necessary.

Text fields for specifying the parameters of the battens are provided for the lattice cold-formed steel sections with chords interconnected through the battens.

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

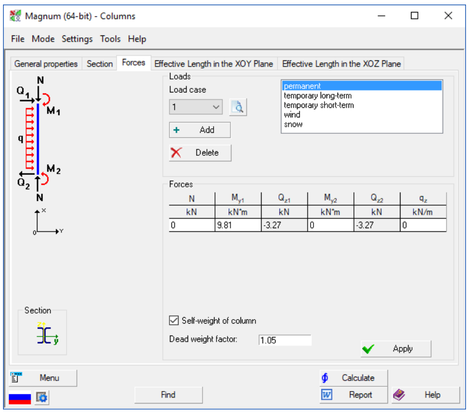


Figure 6.9.4-4. *The* **Forces** *tab of the* **Columns** *dialog box*

The **Forces** tab (Fig. 6.9.4-4) is used to specify forces and moments (longitudinal force, bending moment and shear force) at the ends on the column, and a uniformly distributed lateral load. The drawing in the top left corner of the tab defines the positive directions of internal forces. It should be noted that a positive longitudinal force corresponds to compression in this mode.

All the forces and lateral load are specified simultaneously for each load case. The general conditions of equilibrium are satisfied for these forces and moments. In particular, shear forces Q_1 and Q_2, as well as the nodal moments M_1 and M_2 should be taken from the results of the analysis of the system as a whole. The conditions of equilibrium are as follows:

$$Q_1 - Q_2 + qL = 0,$$

$$M_2 - M_1 - Q_1L - qL^2/2 = 0.$$

Clicking the **Apply** button will perform the calculation of the lacking force factors on the basis of the conditions of equilibrium.

The **Self-weight of column** checkbox enables to automatically represent the self-weight of the column as a uniformly distributed load directed and applied along the column axis. You also have to specify the safety factor for load in the **Self-weight factor** field.

Clicking the button  will open the **Preview** dialog box (Fig. 6.9.4-5) with diagrams of N, M_y and Q_z.

It should be noted that all the loads act in the XOY plane or in the XOZ plane (the X axis is oriented along the bar). The plane is selected in the **General Properties** tab.

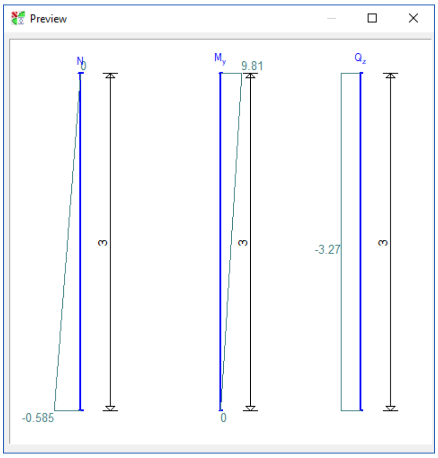


Figure 6.9.4-5. *The* **Preview** *dialog box*

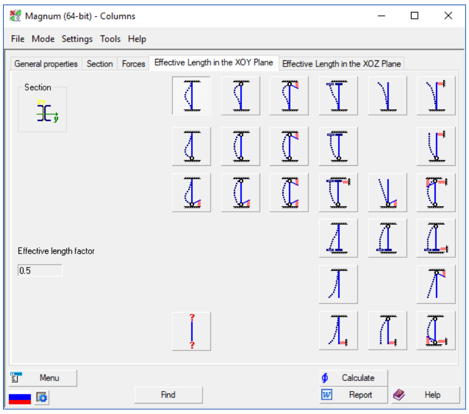
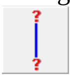


Figure 6.9.4-6. *The* **Effective Length in the XOY Plane** *tab of the* **Columns** *dialog box*

The **Effective Length in the XOY (XOZ) Plane** tabs (Fig. 6.9.4-6) are the exact replicas of the **Effective Lengths** tab from the **Effective Lengths** mode, and they suggest a set of possible conditions of end support in the respective planes of loading for a compressed bar member, which differ from one another in combinations of the boundary conditions (free end, hinge, elastic support, elastic clamping, clamped).

If the boundary conditions include an elastic clamping or an elastic support, a table for entering the data on the stiffness of the respective restraint will appear.

Factors $k = l\,/\,L$ describe the ratio of the effective length, l, to the geometric length of the bar, L**,** in different planes. They are displayed in the respective **Effective length factor** fields.

This dialog box is described in detail in Section 6.7.4. Unlike the **Effective Lengths** mode, this dialog has the **Effective length factor specified by user** button – . Clicking this button will enable you to enter any desired values for the effective length factor and confirm your choice by clicking the **Apply** button. The specified value will then be displayed in the **Effective length factor** field. In all other cases this field is inaccessible.

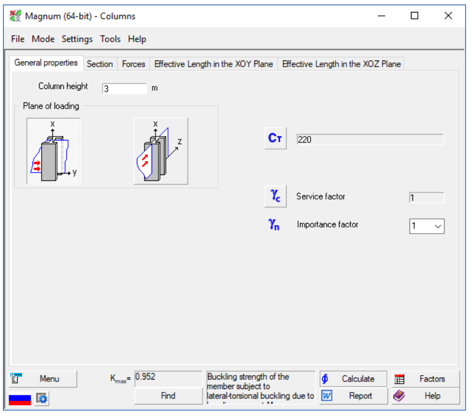


Figure 6.9.4-7. *The main window of the* **Columns** *mode*

The **Factors** and **Find** buttons (the latter is available only for columns from cold-formed steel profiles from an assortment) in the main window of the **Columns** mode (Fig. 6.9.4-7) enable you to analyze the calculation results or to perform the purposeful search of cross-sections for the column and select an optimal (by the criterion of minimum cross-sectional area) cold-formed profile that meets all the requirements of the codes. These modes are described in Section 4.10.1.

Clicking the **Report** button (Fig. 6.9.4-7) will generate a document with all the initial data for the analysis (a design model of the column, its cross-section with geometric properties, load types and values), as well as the results of the static analysis of the column including envelope diagrams and the results of the load-bearing capacity checks of the column in accordance with the design codes selected by the user.

6.9.5 Continuous Purlins

This mode enables to perform all the necessary strength, stability and deformability checks of cold-formed purlins in compliance with the codes. It deals with multi-span cold-formed steel beams serving as roof purlins (including members of horizontal wind trusses) subjected to biaxial bending and an axial force.

The **Continuous Purlins** mode enables to perform the analysis of continuous purlins commonly used in self-framing metal buildings using sections from C- and Z-shaped cold-formed profiles with or without edge fold stiffeners (single or double). The design values of internal forces and moments (longitudinal force N, bending moments M_y, M_z and shear forces Q_z and Q_y) in the sections of the purlin are calculated automatically depending on the given external loads. If the user has selected a section of a load-bearing member from an assortment of cold-formed profiles, this mode also enables to find a cold-formed profile from the assortment.

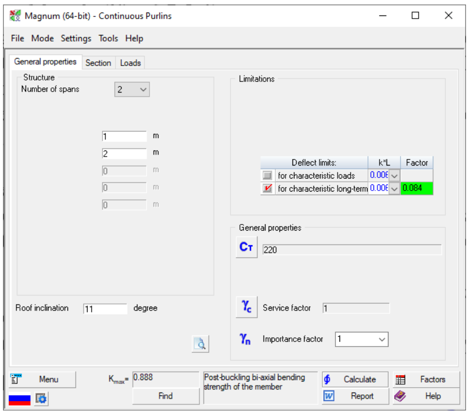


Figure 6.9.5-1. *The* **General Properties** *tab of the* **Continuous Purlins** *dialog box* (when the analysis is performed according to SP 260.1325800)

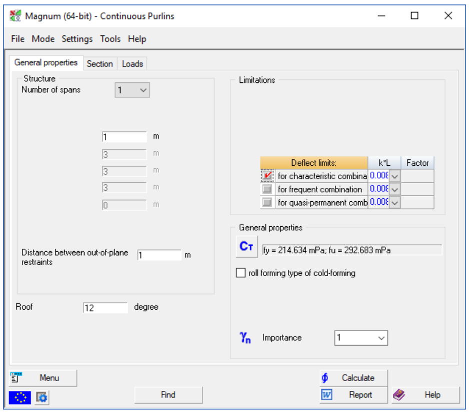


Figure 6.9.5-2. *The* **General Properties** *tab of the* **Continuous Purlins** *dialog box* (when the analysis is performed according to EN 1993)

The **Continuous Purlins** dialog box contains the following tabs: **General Properties** (Fig. 6.9.5-1, Fig. 6.9.5-2), **Section** (Fig. 6.9.5-3) and **Loads** (Fig. 6.9.5-4).

The design of a continuous purlin is specified in the **Structure** group of the **General Properties** tab (Fig. 6.9.5-1, Fig. 6.9.5-2). It is defined by the number of spans and their length.

When developing this mode, it was assumed that purlins are restrained against flexural buckling by the roof sheeting, which is "continuously supported by the top chord of the purlin and is securely connected to it". Taking into account that rigid horizontals are usually not installed between the purlins, the design model of a purlin out of the bending plane is assumed to be the same as in the bending plane.

This tab also contains the **Slope** field where you have to specify the slope of the roof the continuous purlins are mounted on.

The **General Properties** group in this tab enables to specify the steel for the continuous purlin (clicking the button  invokes the **Steel** mode) and the service factor (clicking the button  invokes the **Service Factors** mode). Properties selected in the reference modes are transferred to the respective fields, and can be modified only by accessing the same modes again. The **Importance factor** drop-down list enables to select or enter the value of the importance factor depending on the purpose of the structure.

Service factor cannot be specified if the analysis is performed according to EN 1933 (Fig. 6.9.5-2). The **Cold-rolled forming** checkbox enables to specify the metal cold-forming process for creating a profile, which is subsequently taken into account in the analysis by an increased yield strength of steel due to the edge fold stiffeners. Since SP 260.1325800 does not take these stiffeners into account, this checkbox is not provided in the **General Properties** tab when the analysis is performed according to SP 260.1325800 (Fig. 6.9.5-1).

The **Steel** and **Service Factors** modes are described in detail in Sections 6.6.1 and 6.6.3 respectively.

You can specify limitations of the relative deflection value in the **Limitations** group in the **General Properties** tab (Fig. 6.9.5-1, Fig. 6.9.5-2). The deflection limit for the continuous purlin is specified in fractions of its span length. It will be compared with the actual vertical deflection of the continuous purlin under the loads with design values which correspond to the serviceability limit state.

Note that no special factor is introduced for the deflection check in **Magnum** — the application only calculates and outputs the maximum deflection value.

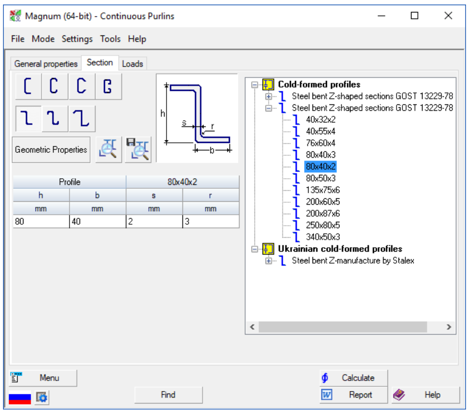


Figure 6.9.5-3. *The* **Section** *tab of the* **Continuous Purlins** *dialog box*

The **Section** tab (Fig. 6.9.5-3) is used to assign a cross-section to the continuous purlin.

It is assumed that the sections do not vary along the purlin length. The available cross-sections include cold-formed C- and Z-shaped sections with or without edge fold stiffeners (single or double).

When a cold-formed profile is selected from the database, all its sizes will be provided in the table under the displayed section (Fig. 6.9.5-3). The user can edit these sizes when necessary.

The procedures for selecting a cold-formed steel profile and for creating a cold-formed steel section are described in detail in Section 6.3.

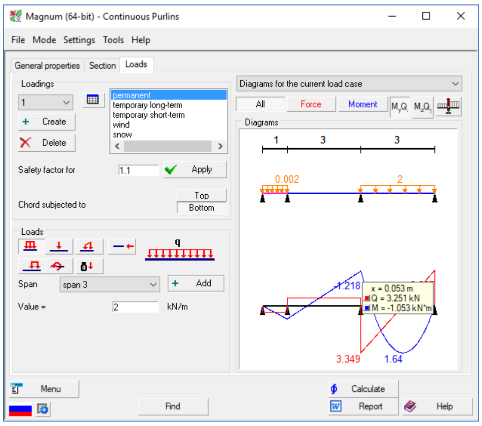


Figure 6.9.5-4. *The* **Loads** *tab of the* **Continuous Purlins** *dialog box*

The **Loads** tab (Fig. 6.9.5-4) is used to specify the loads acting on the considered continuous purlin.

This tab is nearly identical to that from Sections 4.8.1 and 4.10.3. The difference is that an application area (top or bottom chord) of the considered loads has to be specified, which is assumed to be the same for all components of the current load case. Moreover, the loads are specified for each span. The number of a span to which a load is applied is selected from the **Span** drop-down list. The position of the concentrated force is measured from the left edge of the considered span.

If a purlin is used as a rigid horizontal of a wind truss, a longitudinal force may occur in its section. In this case corresponding concentrated longitudinal forces are applied at the ends of the support sections of the purlin.

Bending moment and shear force diagrams for the current load case are displayed in the special window located in the **Loads** tab on the right. Clicking the button  will display the diagrams of moment M_y and shear force Q_z, and clicking the button  will display the diagrams of moment M_z and shear force Q_y.

In Continuous Purlins mode verification of the deflection has been implemented for the continuous purlins subjected to one load case combination only. It should be noted that the problem of determining the stress-strain state of a thin-walled bar made from cold-formed profile is nonlinear, since it considers the postbuckling behavior, and therefore the superposition principle is inapplicable in this case, i.e. the deflections caused by various static actions cannot be summed up when combining these actions.

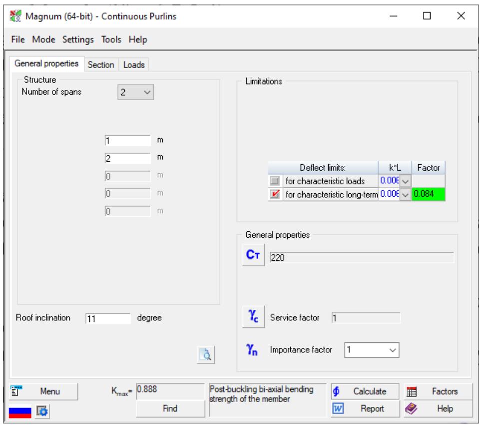


Figure 6.9.5-5. *The main window of the* **Continuous Purlins** *mode*

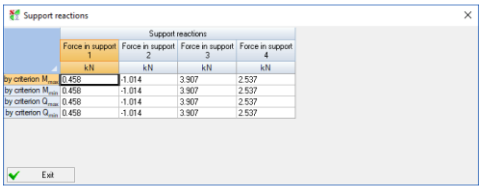
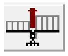


Figure 6.9.5-6. *The* **Support reactions** *dialog box*

The **Factors** and **Find** buttons (the latter is available only for continuous purlins from cold-formed steel profiles from an assortment) enable you to analyze the calculation results or to perform the search of cross-sections for the continuous purlin and select an optimal (by the criterion of minimum cross-sectional area) cold-formed profile that meets all the requirements of the codes. These modes are described in Section 4.10.1 (Fig. 6.9.5-5).

It should be noted that the search of sections for a continuous purlin will be based on the strength, stability and stiffness conditions.

Clicking the **Report** button (Fig. 6.9.5-5) will generate a document with all the initial data for the analysis (a design model of the continuous purlin, its cross-section with geometric properties, load types and values), as well as the results of the static analysis of the continuous purlin including envelope diagrams and the results of the load-bearing capacity checks of the purlin in accordance with the design codes selected by the user.

Moreover, the report document will contain a table with design combinations of support reactions. During the working session, values of the support reactions obtained for their most unfavorable combinations of loadings are displayed in the **Support reactions** dialog box (Fig. 6.9.5-6), which can be invoked by clicking the respective button  in the **Loads** tab.

6.10 References

[1] L.Gardner, D.Nethercot, Designers Guide to Eurocode 3: Design of Steel Buildings, EN 1993-1-1, EN 1993-1-3, EN 1993-1-8, ICE Publishing, 2011.

7. ARBAT

ARBAT enables you to check the load-bearing capacity and to select the reinforcement for concrete and reinforced concrete structural members, to calculate the deflections in reinforced concrete beams, to check the local strength of reinforced concrete structural members (including inserts) in compliance with requirements of one of the following codes:

- SNiP 2.03.01-84*;
- SNiP 2.03.01-84* with Ukrainian modifications;
- KMK 2.03.01-96;
- SNiP 52-01-2003 (SP 52-101-2003);
- SP 63.13330;
- DBN B.2.6-98:2009;
- STR 2.05.05:2005.

The analysis is performed taking into account the ultimate and serviceability limit states (strength and crack resistance) for the design combinations of forces (DCF) selected automatically depending on the specified design loads in compliance with SNiP 2.01.07-85* “Loads and Actions” and SNiP 2.03.01-84*, KMK 2.03.01-96 “Concrete and Reinforced Concrete Structures” (SNiP 52-01-2003). If the selected design code is SP 63.13330, then loads will be combined according to the requirements of SP 20.13330.

Design values of loads (forces) are specified in all modes of the application. Characteristic values of loads are used in the analysis of the serviceability limit state. User-specified safety factors for load are used for each load case to convert design values to the characteristic ones.

The selection of reinforcement and the check are performed for beams, columns, and slabs from heavy-weight, fine-grained and lightweight concrete with the reinforcement steel of the following classes A-I (A240), A-II (A300), A-III (A400), A-IV, A-V and A-VI, and the reinforcing wire of class Bp-I (B500) and reinforcement of classes A400C, A500C (A500), A600C. Moreover, the selection and check of the following types of fiber reinforced polymer (FRP) rebar are implemented in accordance with the requirements of SP 295.1325800: glass, basalt, carbon, aramid, and hybrid.

Moreover, **ARBAT** plays the role of a manual that provides data on the assortment and properties of reinforcement, characteristic 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 the results of the analysis, which are implemented in the application, are exactly the same as those in other design and analysis applications included in the **SCAD Office®** system. The program uses a common multi-tab technique. To switch to a mode, click on its tab or use an appropriate menu item.

7.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 7.1-1) with a set of buttons for selecting a working mode.

Design codes can be selected from the respective 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 according to SNiP 2.03.01-84*, KMK 2.03.01-96, SNiP 52-01-2003 (SP 52-101-03), SP 63.13330.2012, SP 63.13330.2018, DBN B.2.6-98:2009, STR 2.05.05:2005, and SNiP 2.03.01-84* with Ukrainian modifications.

All modes can be classified into four groups:

- reference modes united in the **Information** group;
- design modes for checking of sections and members with the given reinforcement united in the **Check** group;
- design modes united in the **Local Strength** group which implement the check of strength of structural members near the load application area;

- design modes for selecting the reinforcement united in the **Selection of Reinforcement** group.

The reference modes include:

- **Concrete Class** — enables to browse the characteristic and design values of strength of various classes of concrete for the ultimate and serviceability limit states according to SNiP 2.03.01-84*, KMK 2.03.01-96 or SP 52-101-2003, SP 63.13330;
- **Concrete Grade (SNiP II-21-75)** — provides information similar to that of the previous mode but related to the concrete grades according to SNiP II-21-75;
- **Reinforcement** — provides data on the assortment, characteristic and design strength of various classes of reinforcement;
- **Service Factors** — provides information from the tables of SNiP and SP;
- **Deflection Limits** — provides the deflection limit values from Table 19 of SNiP 2.01.07-85*;
- **Geometric Properties —** enables to determine the geometric properties (area, moments of inertia, parameters of a reduced section etc.) of a given concrete or reinforced concrete section;
- **Reinforcement Anchorage** — calculation of the anchorage (embedment) length of the reinforcement in concrete and the overlap length (lapped reinforcement) of rebars.

Figure 7.1-1. *The main window*

The **Check** group includes the following modes:

- **Strength of RC Sections** — enables to determine the load-bearing capacity of sections of reinforced concrete members with the given reinforcement;
- **Strength of Concrete Sections** — enables to determine the load-bearing capacity of sections of concrete members;
- **Beam Deflection** — enables to determine the deflections caused by a given load in a multi-span beam;
- **Single-Span Beam Deflection** — enables to determine the deflections caused by a given load in a single-span beam;
- **Beam Check** — enables to check the load-bearing capacity of a multi-span beam with the given reinforcement;

- **Single-Span Beam Check** — enables to check the load-bearing capacity of a single-span beam with the given reinforcement;
- **Column Check** — enables to check the load-bearing capacity of a column with the given reinforcement;
- **Slab Check** — enables to check the load-bearing capacity of a slab supported along its contour and having the given reinforcement;
- **Slab Check (Karpenko' theory)** — enables to select the reinforcement for slabs according to Karpenko' theory;
- **Ultimate Slab Load** —enables to determine the ultimate uniformly distributed load on the slab.

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

- **Local Compression —** enables to check the load-bearing capacity of structural members for local compression;
- **Punching —** enables to check the load-bearing capacity of slab structures for punching;
- **Tearing** — enables to check the load-bearing capacity of junctions between structures for tearing;
- **Inserts** — enables to check the load-bearing capacity of inserts;
- **Short Cantilevers** — enables to check short cantilevers for the action of a lateral force;
- **Keys** — enables to check concrete keys with the given parameters under the action of shear and longitudinal forces.

The **Selection of Reinforcement** group includes:

- **Selection of Beam Reinforcement** — enables to select the reinforcement for a multi-span beam;
- **Selection of Single-Span Beam Reinforcement** — enables to select the reinforcement for a single-span beam;
- **Selection of Column Reinforcement** — enables to select the reinforcement for columns;
- **Selection of Section Reinforcement** — enables to select the reinforcement for cross-section of the element;
- **Selection of Slab Reinforcement (Karpenko' theory)** — enables to check the load-bearing capacity of a slab according to Karpenko' theory.

7.2 Information Modes

The reference modes provide data on the materials given in SNiP and SP. All values in the tables are given in the same units of measurement as in the design codes, and do not depend on the settings of the application.

7.2.1 Concrete Class

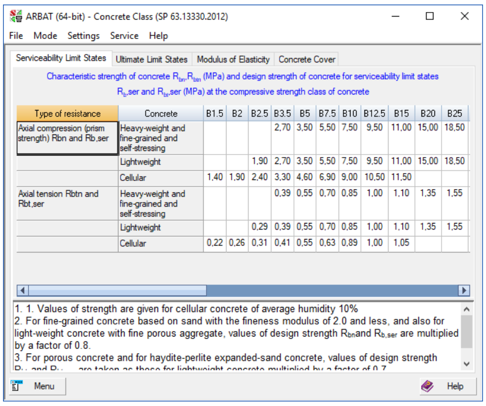


Figure 7.2.1-1. *The* **Concrete Class** *dialog box*

A multi-tab dialog box of this mode (Fig. 7.2.1-1) contains data given in Section 2 (Tables 12, 13 and 18) of SNiP 2.03.01-84* (Tables 5.1, 5.2 and 5.4 of SP 52-101-2003, Tables 6.7, 6.8 and 6.11 of SP 63.13330), as well as information on the minimum allowable concrete cover.

7.2.2 Concrete Grade

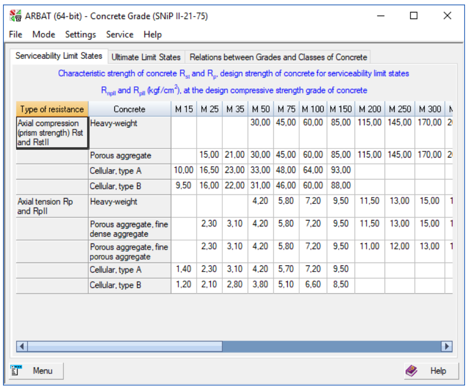


Figure 7.2.2-1. *The* **Concrete Grade** *dialog box*

This mode (Fig. 7.2.2-1) can be used to obtain information about the design and characteristic values of strength of concrete of various grades in compliance with SNiP II-21-75. This mode also provides data about the relation between the grades and the classes of concrete according to GOST 26633-91.

7.2.3 Reinforcement

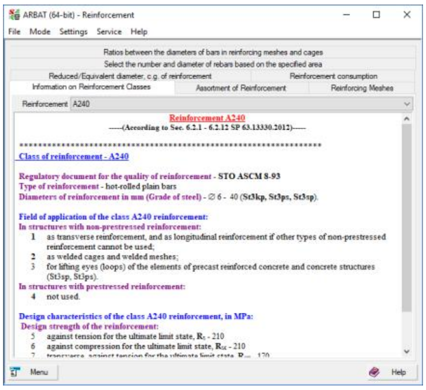


Figure 7.2.3-1. *The* **Reinforcement** *mode*

This mode (Fig. 7.2.3-1) provides data about classes of reinforcement and the assortment of reinforcement products, including an effective area of the cross-section for a particular number of rebars, a theoretical weight of one linear meter and diameters of reinforcement of various classes. Moreover, a separate tab provides information about reinforcing meshes according to GOST 23279-85 and the minimum distances between bars in welded cages and welded meshes. A separate tab provides information about the ratio between the diameters of welded and tied bars in reinforcing meshes and cages (according to the Manual [11]).

A separate tab enables to convert the cross-sectional area of reinforcement into bars, select the number of bars, and arrange the rebars in the section of the element.

The program enables to select both the minimum required amount of reinforcement of one diameter, and the combinations of bars. In this case, it is possible to obtain the dimensions of the bars and their arrangement in the section of the element.

The result can be obtained with a certain deviation specified by the user (i.e., the total area of the selected bars can be smaller/larger than the specified value by the specified deviation). If the deviation is 0%, the nearest larger value of the total area of the bars of a certain diameter is determined (the 0% deviation limitation is applicable only for single bars, a certain possible deviation has to be specified for a combination of bars).

The algorithm for selecting rebars is based on the following limitations and assumptions:

- the maximum possible number of rows of reinforcement along the height – 3 rows;
- number of rebars in the third row – 2;
- up to five different diameters can be used in a section of the element (it is recommended to use not more than two different diameters);
- bars of larger diameter are placed in the first row at the corners of the section. The bars of larger diameter in each row are always placed at the edges;
- the cross-sectional area of the reinforcement in the section of the smaller bar should not be less than 25% of the cross-sectional area of the larger bar;

- the bars of the 1st row are aligned along the border of the concrete cover (the centers of gravity of the bars are not on the same line, so that the stirrup could be installed);
- the centers of the bars of the second and third rows lie on the same straight line.

The program enables to obtain a report document. Only the variants of the calculation results selected by the user will be output in the report.

The **Reduced/Equivalent Diameter, C.o.G. of Reinforcement** tab enables to perform the following auxiliary operations:

Reduced diameter — enables to determine the reduced diameter of a rebar under the constrained placement of two rebars (pairwise without a gap between them, or with the distance between the rebars of a pair less than the distance required for individual rebars). When specifying the distances between the rebars and determining the anchorage length, this pair of rebars should be considered as a reduced rebar with a diameter of $d_{red} = \sqrt{d_1^2 + d_2^2 - c_1^2}$, where d_1 and d_2 – diameters of the considered rebars, d_1 – clear distance between these rebars taken as not greater than the diameter of a smaller rebar (see Sec. 5.40 and Fig. 102 Guide on designing of concrete and reinforced concrete structures made of heavy-weight or lightweight concrete (no prestressing) (to SNiP 2.03.01.-84)).

Equivalent diameter — enables to determine the equivalent diameter of a rebar when determining the crack opening width if rebars of different diameters are used in the section.

For a section with rebars n_1 of diameter d_1 and n_2 of diameter d_2, the equivalent diameter d_{eq} is determined according to the following formula:

$$d_{eq} = \frac{n_1 d_1^2 + n_2 d_2^2}{n_1 d_1 + n_2 d_2}$$

Center of gravity of the multi-row reinforcement — enables to determine the distance from the edge of concrete to the center of gravity of the multi-row reinforcement.

For a section with rebars of a certain diameter with the total area of $A_{s1}(A_{sn})$ located at a distance of $a_{s1}(a_{sn})$ from the edge of the element, the center of gravity of the multi-row reinforcement is determined according to the following formula:

$$a = \frac{A_{s1}a_{s1} + A_{s2}a_{s2} + ... + A_{sn}a_{sn}}{A_{s1} + A_{s2} + ... + A_{sn}}$$

7.2.4 Service Factors for Concrete

This mode (Fig. 7.2.4-1) provides information about service factors given in Tables 15, 16, and 17 of SNiP 2.03.01-84* (Sec. 5.1.10 of SP 52-101-2003, Sec. 6.1.12, 6.1.9 of SP 63.13330).

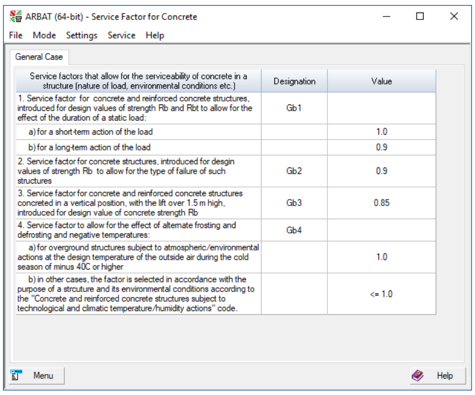


Figure 7.2.4-1. *The* **Service Factor** *mode*

7.2.5 Deflection Limits

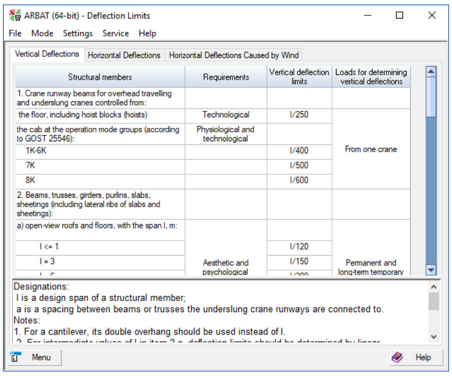


Figure 7.2.5-1. *The* **Deflection Limits** *mode*

This mode (Fig. 7.2.5-1) provides information about limit values of vertical deflections and also of horizontal deflections caused by cranes or wind given in Tables 19, 21 and 22 of SNiP 2.01.07-85* (Tables F.1, F.3 of SP 20.13330.2011) .

7.3 General Operations

Some operations or groups of controls in the application are of a general character and can be used in different modes both for analysis and check. These include operations of creating a cross-section, obtaining data about concrete and reinforcement, generating a report etc. Such operations and groups of controls are described below.

7.3.1 Creating Cross-Sections

Operations of selection of a cross-section for a structural member are common for most modes of the application.

The application works with six types of cross-sections of elements, which are shown in Fig. 7.3.1-1. A section shape is selected by clicking the respective button in the group, then the section sizes and the thickness of the concrete cover are specified (in the reinforcement selection modes you have to specify the distance to the center of gravity of reinforcement instead of the concrete cover).

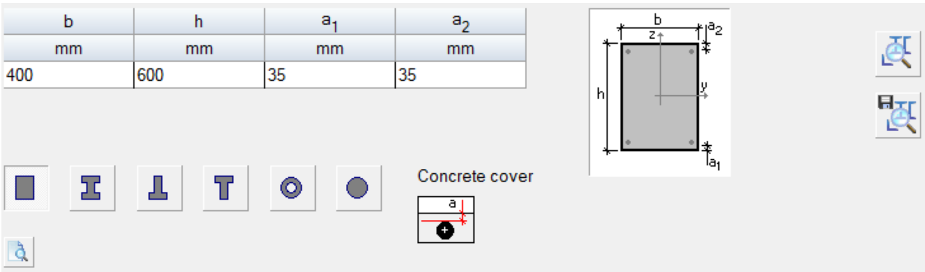


Figure 7.3.1-1. *Section types*

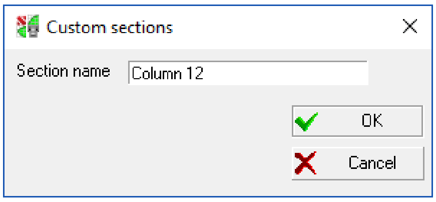


Figure 7.3.1-2. *The* **Custom sections** *dialog box*

As you work with the application, you can gradually create a custom sections database. To save a new section in the database, use the button  which invokes the **Custom sections** dialog box where you can specify a name for the section (Fig. 7.3.1-2).

Since the application does not verify the uniqueness of the names used, it is the user who has to take care of it.

To access the database and load a previously created section, use the button . The dialog box with the custom sections database (Fig. 7.3.1-3) contains a table with the section names and buttons for previewing —  or renaming (**Rename**) a section. To select a section, highlight the respective row in the list and click the **Apply** button.

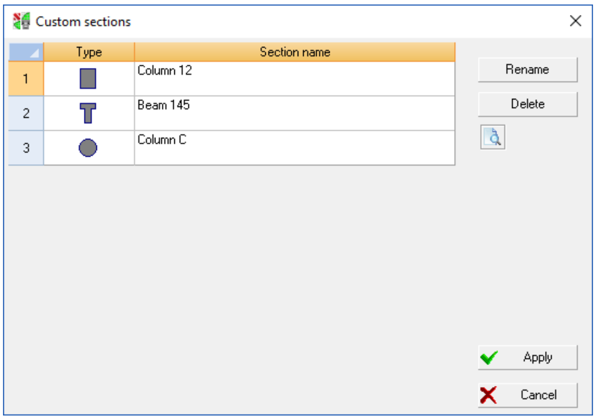


Figure 7.3.1-3. *A dialog box with the list of the custom sections*

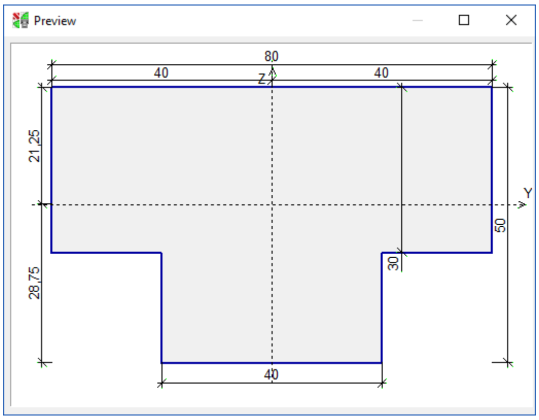


Figure 7.3.1-4. *The* **Preview** *dialog box*

Clicking the button  enables you to view a section in the **Preview** dialog box (Fig. 7.3.1-4).

Since **ARBAT** works with reinforced concrete sections, information about concrete and rebars is required as well. The rules for specifying these data are given below (Sections 7.3.1, 7.3.1).

7.3.2 Concrete Data

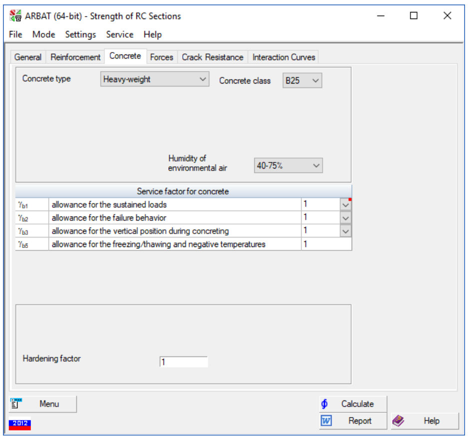


Figure 7.3.2-1. *The* **Concrete** *dialog box*

The following information should be specified in all design modes when preparing data in the **Concrete** tab: concrete type, concrete class, service factors for concrete, conditions of hardening, and the hardening factor (see Fig. 7.3.2-1).

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

If the analysis is performed according to SP 52-101-2003, only heavy-weight concrete will be available because this document does not provide rules for designing from other types of concrete. If the analysis is performed according to SP 63.13330, lightweight concrete can not be selected because the deformation characteristics should be taken according to the special instructions.

The service factor for concrete, γ_{b2} (or γ_{b1} if the analysis is performed according to SP 52-101-2003 and SP 63.13330), allows for the load duration.

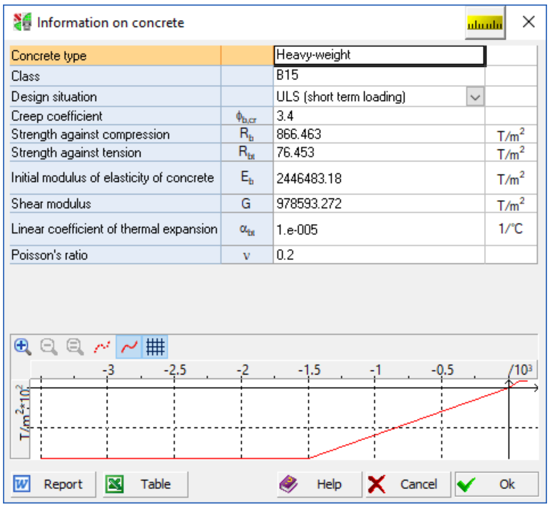
The value of the factor is set equal to either 1 or 0.9 (item 2 *a* in Table 15 of SNiP 2.03.01-84*, and Sec. 5.1.10 *a* of SP 52-101-2003 and Sec. 6.1.12 of SP 63.13330), the default value is 1. In cases when a different value has to be used for this factor according to conditions of the analysis, it can be specified directly in the list field.

If a combination of loadings includes short-term loads, the analysis will use γ_{b2} = 1.1 (regardless of the specified value) in compliance with regulations of item 2*b* in Table 15 of SNiP 2.03.01-84*. Similar actions are performed when the calculations are performed according to SP (where the designation γ_{b2} is replaced by γ_{b1}, and the value 1,1 is replaced by 1,0).

The service factor for concrete, γ_b, is a product of all service factors for concrete from Table 15 of SNiP 2.03.01-84* (Sec. 5.1.10 of SP 52-101-2003 and Sec. 6.1.12 of SP 63.13330), except for γ_{b2} (γ_{b1}). The default value of it is 1 (0.9 in the **Strength of Concrete Sections** mode).

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

The button  invokes the dialog box, which provides the design properties of the material.



The **Design situation** drop-down list enables to select the type of analysis (ultimate limit state, serviceability limit state,...) in order to display the respective parameters.

7.3.3 Reinforcement Data

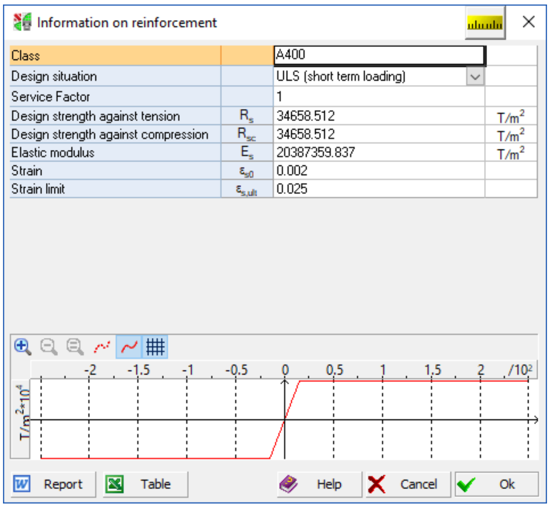
Data on the section reinforcement include information about the classes of longitudinal and transverse reinforcement, service factors for reinforcement, and data concerning the arrangement of the rebars.

The reinforcement class (or type of FRP rebar) is selected from a drop-down list, the content of which depends on the selected design code (see Fig. 7.3.3-1). The service factors for reinforcement should be entered by the user; their default value is 1. If one of the types of FRP rebar is selected, you then have to specify the operating conditions – indoors, outdoors, or in the ground.

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

Figure 7.3.3-1. *Reinforcement data*

The button  invokes the dialog box, which provides the design properties of the material.



The **Design situation** drop-down list enables to select the type of analysis (ultimate limit state, serviceability limit state,...) in order to display the respective parameters.

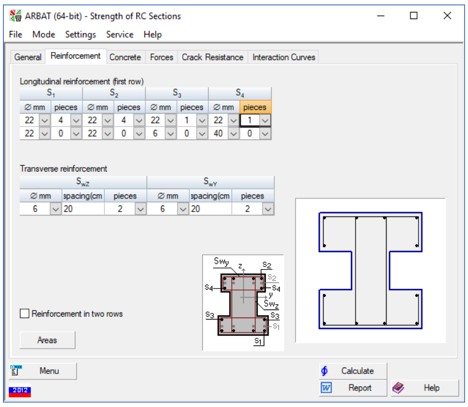


Figure 7.3.3-2. *Arrangement of rebars*

Information on the rebar arrangement is specified, as a rule, in a separate tab named **Reinforcement** (Fig. 7.3.3-2).

Three tables are used to specify the characteristics 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 contains information about the diameters and the number of rebars in the first row of the lower, upper, and side longitudinal reinforcement. The second line of the table is filled only when the first row of the lower (S_1), upper (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).

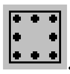
If the **Reinforcement in two rows** checkbox is checked, the **Longitudinal reinforcement (second row)** table appears in the window. It is used to specify the diameter and the number of rebars in the second row of the lower and upper longitudinal reinforcement, and the clear distance between the first and second rows of the lower (Δ_1) and upper (Δ_2) reinforcement. If the clear distance is set to zero, then the software takes it as the minimum design spacing between reinforcing bars which are placed vertically during concreting.

Data on the transverse reinforcement are entered in the **Transverse reinforcement** table (in the **pieces** column of this table you have to specify the number of cuts, i.e. the number of intersections of stirrups with the plane going through the longitudinal axis of the bar).

When you are filling in the tables, the entered data are interpreted according to the following 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 transverse reinforcement;
- the diameter and the number of rebars are selected from the respective drop-down lists.

The shape of the specified section and the arrangement of reinforcement in it are displayed in the window. The **Areas** button opens a table with the values of areas of the specified reinforcement (Fig. 7.3.3-3).

A “uniform” reinforcement can be specified for rectangular sections, when an equal number of rebars of the same diameter is placed along each face of the section and which is often used for columns. To specify the number and the diameter of rebars, use the **Rectangular section** dialog box (Fig. 7.3.3-4), which can be opened by clicking the button . Exiting this dialog will automatically fill the **Longitudinal reinforcement (first row)** table.

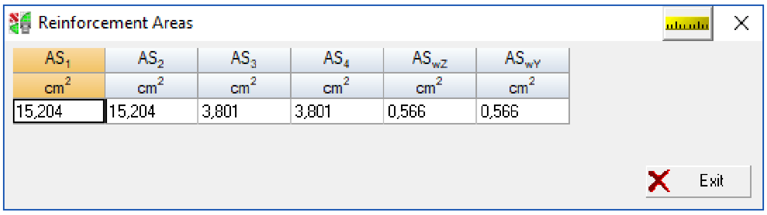


Figure 7.3.3-3. *The* **Reinforcement areas** *dialog box*

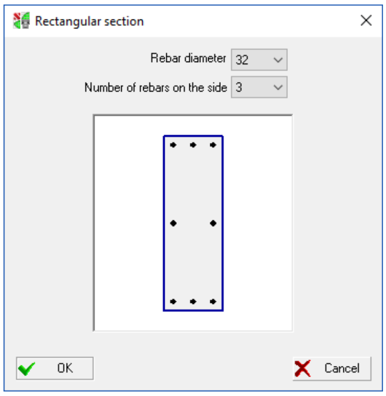


Figure 7.3.3-4. *The* **Rectangular section** *dialog box*

If the number of rebars is greater than two, both lines of the table are filled. The first line describes two corner rebars S_1 and S_2, and all rebars S_3. The second line contains information about intermediate rebars S_1 and S_2. This arrangement of data in the tables enables to change the diameter of the corner rebars easily.

The software checks whether the design requirements of Sec. 5.12 of SNiP 2.03.01-84*, KMK 2.03.01-96 (Sec. 5.3 of SP 52-101-2003, Sec. 10.3.5 of SP 63.13330) regarding the minimum distance between the rebars are satisfied, and if they are violated, a warning message is displayed. The user can ignore the warning about the violation of requirements by clicking the **Ignore** button, but the negative consequences will not be analyzed.

7.3.4 Crack Resistance

In cases when the analysis of the formation and opening of cracks is required, the user should provide data about a category of crack resistance, an allowable crack opening width etc. All these data are entered in the **Crack Resistance** tab (Fig. 7.3.4-1).

a *b*

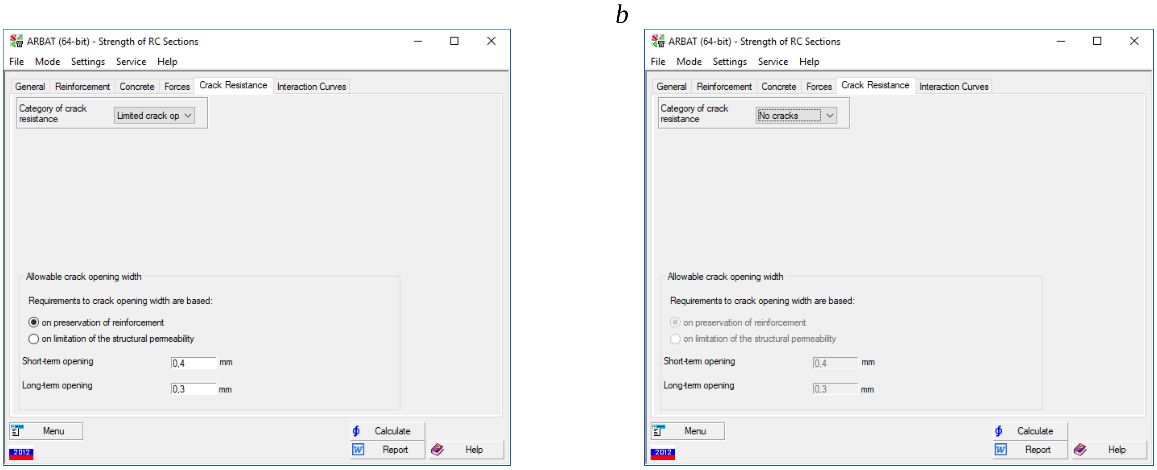


Figure 7.3.4-1. *Crack resistance data*

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

If the analysis is performed according to SNiP 2.03.01-84*, KMK 2.03.01-96 and the 3rd category of crack resistance is selected (Fig. 7.3.4-1, *a*), then you have to use the respective lists to select the conditions of operation, the mode of concrete humidity, and the humidity of environmental air. Next, specify an allowable width of short-term and long-term crack opening (the default values are those recommended by SNiP 2.03.01-84* on the basis of conditions of operation and humidity parameters).

If the analysis is performed according to SP 52-101-2003 and the *Limited crack opening width* item is selected in the drop-down list (Fig. 7.3.4-1, *b*), you have to specify the requirements to crack opening width (*based on preservation of reinforcement* or *based on limitation of the structural permeability*). This will automatically set the maximum allowable widths of long-term and short-term crack opening recommended by Sec. 7.2.3 of SP 52-101-2003. The user can edit these values when necessary.

7.3.5 Importance Factor

All design modes require data on the importance factor according to GOST 27751-88. The factor can be selected from a drop-down list in accordance with the characteristic of a structure (it can be of *special, essential* or *limited* importance) or entered by the user (in non-standard situations).

7.3.6 Fire Resistance

Fire resistance is the ability of structural elements to maintain strength under fire conditions. The fire resistance of a building depends on the fire resistance of its main structural parts. It is also necessary to take into account the variation of the thermal and mechanical properties of concrete and reinforcement under the fire heating of the structure along with the mechanical actions.

Evaluation of fire resistance includes:

- thermal analysis — analysis of structural heating;
- strength analysis — analysis of the mechanical behavior of the structure.

A structure is fire resistant when the necessary safety margin can be provided for a certain period of time.

The fire resistance analysis of individual unprotected elements of reinforced concrete structures is implemented in **Arbat**. These calculations can be performed according to the following codes: STO 36554501-006-2006, EN 1992 or DSTU-N B B.2.6-196:2014, DSTU-N B B.2.6-197:2014.

Thermal Analysis

The calculation is performed in order to determine the temperature distribution in a design cross-section, when the temperature field $T(y,z,t)$ is defined by the nonlinear transient heat equation

$$\frac{\partial}{\partial y}\left(\lambda\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\lambda\frac{\partial T}{\partial z}\right)=c\rho\frac{\partial T}{\partial t},$$

where $\lambda(y,z,T)$ is the thermal conductivity factor, $c(y,z,T)$ is the specific heat, ρ is the density of the material. The relationship between these parameters and the temperature is given in the design codes.

The solution of the above heat equation is found for the boundary conditions describing the heat transfer from the hot air

$$\frac{\partial T(t)}{\partial n}=\alpha\left[\theta(t)-T(t)\right].$$

And the air heating is defined by the fire temperature $\theta(t)$, which is described by a certain time function.

The initial conditions considered when solving the heat equation describe a temperature field $T(y,z,0)=20°C$ constant over the entire section.

Strength Analysis

The section strength check performed when evaluating the fire resistance differs from the usual case in that:

- a special (emergency) limit state is considered, and therefore only characteristic permanent and long-term loads are taken into account in addition to the fire action;
- the variation of physical and mechanical properties of materials (concrete and reinforcement) due to heating is taken into account, and since the temperature field is uneven, these characteristics are determined for each design point of the cross-section independently.

The program determines the value of the load-bearing capacity utilization factor K_{max} from the combination of loads which includes only the values of constant and long-term loads at t = 0. The characteristic values of the strength of the material are used in the calculation (see for example SP 296.1325800.2017).

The design strength of concrete R_{bt} and reinforcement R_{st}, as well as their elastic moduli E_{bt} and E_{st} are used taking into account the values R_b, R_s, E_b and E_s for the given classes of concrete and reinforcement, but they are adjusted for the heating temperature T,°C according to the formulas

$$R_{bt} = R_b \gamma_{bt}; \quad R_{st} = R_s \gamma_{st}; \quad E_{bt} = E_b \beta_b; \quad E_{st} = E_s \beta_s .$$

Moreover, the values of the critical strain of reinforcement ε_{s2} and concrete ε_{b2} depend on the temperature as well, because the “stress-strain” relationship changes with increasing temperature.

The following factors are obtained in the result of the analysis:

- Strain of concrete in a normal section;
- Strain of reinforcement in a normal section;
- Strength in a strip between oblique sections under the lateral force;
- Strength of an oblique section under the lateral force.

Calculation and Analysis of Results

The fire resistance analysis is implemented in the following modes: *Strength of RC Sections*, *Beam Check*, *Single-Span Beam Check* and *Column Check.* In order to perform this calculation, check the respective checkbox in the **General** tab, and the ***Fire Resistance*** tab will appear.

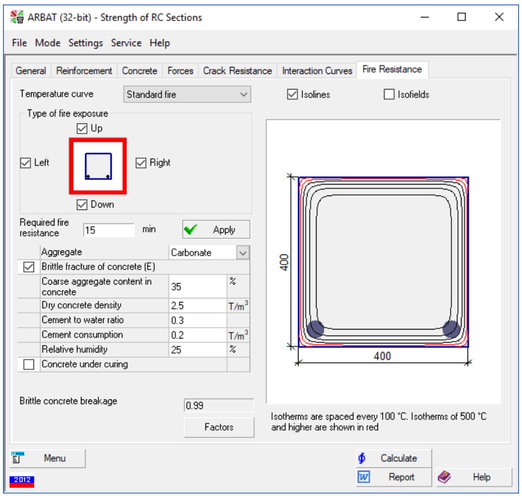


Figure 7.3.6-1. *The* **Fire Resistance** *tab*

This tab enables to select the **Temperature-time curve** from the respective drop-down list, define the **Type of fire** (using checkboxes **Up**, **Down**, **Left**, **Right**), select aggregate type, and to specify the **Required fire resistance**. Click the **Apply** button and the application will perform the thermal or strength analysis.

In the case of beams and columns, drop-down lists may additionally appear to select a span and segment.

If you want to perform the brittle fracture analysis of concrete, check the respective checkbox and specify the necessary data (cement to water ratio, cement consumption, coarse aggregate content in concrete, ...).

The results of the thermal analysis can be graphically displayed as isofields and/or isolines of the temperature distribution across the section at the given time (the **Required fire resistance** parameter). The type of display can be selected with the help of the respective checkboxes. The temperature field can be dynamically digitized, when the application displays the temperature in a certain section point indicated by the cursor (if the cursor is within a rebar, the temperature value in the center of this rebar is displayed).

The value of the utilization factor of restrictions K_{max} is obtained in the result of the analysis. Clicking the **Factors** button outputs the factors diagram.

Unfortunately, there are many errors and typos in STO 36554501-006-2006 (and the Manual to this STO). Here are some corrections that were made when implementing the fire resistance analysis.

- The formula for determining specific heat provided in Sec. 6.3 of STO 36554501-006-2006 has a minus sign, which is obviously a typo, because the respective formulas in MDS 21-2.2000 and other sources have a plus sign.
- The proportionality factor given in Sec. 9.3 of STO 36554501-006-2006 has incorrect units of measurement (W $m^{3/2}$/kg). The correct units are W $m^{5/2}$/kg (see *Recommendations for the Protection of Concrete and Reinforced Concrete Structures from Brittle Fracture*, M., Stroyizdat, 1979).
- According to Sec. 5.11 of STO 36554501-006-2006 the value ε_{s2}=0,0025 at t_s = 20 – 200 °C. This is obviously a typo, and the value ε_{s2}=0,025 should be used (see Sec. 6.2.14 of SP 63.13330), because there must be a correspondence between the normal calculation and that taking into account the fire action. For similar reasons, the value ε_{b2}=0,0035 (see Sec. 6.1.20 of SP 63.13330) is used instead of the value ε_{b2}=0,00035 given in Sec. 5.7 of STO 36554501-006-2006 (or ε_{b2}=0,000035 given in Sec. 8.2 of the Manual to STO 36554501-006-2006).

The above errors have been corrected in SP 468.1325800.2019, but this SP does not provide any recommendations for the fire resistance analysis based on a non-linear deformation model that allows for the general case. Therefore, the software performs the thermal design in accordance with SP 468.1325800.2019, and the strength analysis is performed in accordance with the recommendations of STO 36554501-006-2006.

7.3.7 Advanced Settings

In many modes there is a button  next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional calculation parameters.

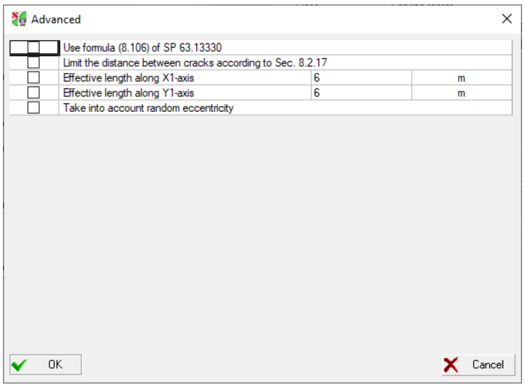


Figure 7.3.7-3. *The* **Advanced Settings** *tab*

- ***Limit the distance between cracks according to Sec. 8.2.17***

The program performs the analysis of plate elements based on the theory of N.I. Karpenko, since SP 63.13330 provides recommendations for calculating flat elements only for particular cases (slabs and deep beams). The codes do not provide any clear guidelines for the calculation of shell elements with simultaneously acting membrane and bending groups of forces. SCAD determines the crack opening width by the formulas recommended by N.I. Karpenko of the type $(\varepsilon_x+\varepsilon_y)l$, where l is the distance between cracks, $\varepsilon_x,\varepsilon_y$ are strains in the respective directions.

To determine the crack opening width in flat elements SP 63.13330 refers to Section 8.2 (requiring to neglect the torque), which considers the formation of cracks in bar elements and contains the following limitation on the distance between cracks: “not less than 10 d_s and 10 cm and not more than 40 d_s and 40 cm”. The authors of the program do not know whether this limitation is reasonable with respect to plate elements, therefore it is not used by default. However, you can activate it using this additional setting.

- ***Use the formula (8.106) from SP 63.13330***

The default algorithm for calculating reinforced concrete flat elements is based on two separate calculations — for the action of Q_x and Q_y. SP 63.13330 contains the following recommendation for the calculation for shear forces — formula (8.106)

$$\frac{Q_x}{Q_{x,ult}}+\frac{Q_y}{Q_{y,ult}}\le 1.$$

This formula sums the stresses at various sites and can hardly seem reasonable. Based on the vector addition rules, it would be more logical to consider the shear force $Q=\sqrt{Q_x^2+Q_y^2}$. The following paper S.N. Karpenko, *Modern design techniques of high buildings made of monolith reinforced concrete*, High Buildings, 2007, No. 3, 34-39, for example, contains a similar recommendation on the analysis of flat elements.

- ***Increase the area of longitudinal reinforcement when implementing Sec. 8.1.34 of SP 63.13330***

Section 8.1.34 which has appeared in SP 63.13330 regulates the consideration of a longitudinal force in the analysis of the behavior of a bar element under shear forces. This section is obviously taken from EN 1992-1-1 (see Sec. 6.3.2). However, unlike the Eurocode, where the coefficient φn is used only for prestressed structures, SP 63.13330 requires to use it even when there is no prestress.

When the longitudinal forces are relatively large, there are two methods for selecting transverse reinforcement. If the first (default) one is used, the transverse reinforcement is selected after the longitudinal one. The previously selected longitudinal reinforcement is taken into account when calculating the coefficient φ_n, but the area of longitudinal reinforcement does not change when selecting transverse reinforcement. In the second case having established that the value φ_n is too small, you can try (it is not always possible) to increase the area of longitudinal reinforcement and select the transverse reinforcement. The second method in most cases leads to an unreasonably high reinforcement ratio (it is much more effective to increase the formwork sizes of the section) and is not used by default. However, this additional setting enables to activate it.

- ***Ignore torsion***

This option can be used when the analysis is performed according to EN 1992-1-1. It has been added due to the fact that Sec. 6.3.1(2) EN 1992-1-1 mentions a rather wide range of cases when torsion can be ignored. Moreover, using this option enables to avoid one inconsistency of EN 1992-1-1. The thing is that if a member is subjected to shear, both the resistance of concrete and the resistance of reinforcement are taken into account in the analysis (see Sec. 6.2.3). If a member is simultaneously subjected to both torsion and shear, then Sec. 6.3.2(4) limits its resistance by the capacity of the concrete struts.

- ***Implementation of Sec. 8.1.34 of SP 63.13330***

The new Sec. 8.1.34 of SP 63.13330 regulates the shear analysis during eccentric compression/tension. It should be noted that the rules for calculating the coefficient φ_n in this section are almost literally taken from EN 1992-1-1, but in EN 1992-1-1 these rules apply only to prestressed elements. In addition, Sec. 8.1.34 of SP 63.13330 does not provide any rules for calculating the coefficient φ_n in the case when the average stress from the tensile force σ_t exceeds R_{bt}. The default value is φ_n=10^{-6}, but it is also possible (and quite logical) to set the following value φ_n=0.5 (which corresponds to $\sigma_t = R_{bt}$). In order to avoid the above problems, the software allows to set the lower limit of the coefficient φ_n for tension, as well as the upper and lower limits of φ_n for compression. In particular, by setting all limit values to 1.0, you can perform the analysis corresponding to the recommendations of EN 1992-1-1, and by setting the lower limit φ_n equal to 0.5, you can determine φ_n for $\sigma_t > R_{bt}$.

Change No. 2 to SP 63.13330.2018 changes the rules for calculating the coefficient φ_n. When choosing SP 63.13330.2018 as the design code, this change will be taken into account, but a special option allows you to refuse to use this change.

- ***Normalized Normal Force***

In the case of the seismic analysis of ductile reinforced concrete walls Sec. 5.4.3.4.1(2) of EN 1998-1 requires that the value of the normalized axial load should not exceed 0.4. You can use this checkbox to perform an additional calculation of the factor equal to $N/(0.4 \cdot A \cdot f_{cd})$ during the check in order to control the fulfillment of this requirement.

- ***Increase Design Shear Forces for Seismic Combinations***

You can use this checkbox to increase the values of shear forces by 50% for the seismic analysis and thereby satisfy the requirement of Sec. 5.4.2.4(7) of EN 1998-1.

- ***Trilinear stress-strain diagram of concrete in the strength analysis***

A bilinear stress-strain diagram of concrete is used in the ultimate limit state analyses by default according to the requirements of Sec. 6.1.16 of SP 311.1325800 and Sec. 8.26 of STO 36554501-006-2006. At the same time, SP 63.13330 allows to use both bilinear and trilinear diagrams. This parameter enables to switch to the trilinear diagram mode.

- ***Taking into account the behaviour of tensile concrete in the nonlinear deformation model in the strength analysis***

Most standards require not to take into account the behaviour of tensile concrete in the ultimate limit state analyses. However, in the case of the strength analysis of normal sections based on a nonlinear deformation model, some standards say that "*the resistance of concrete in the tension area **may** not be taken into account*" (see Sec. 6.1.2 of DBN V.2.6-98:2009 or Sec. 8.1.20 of SP 63.13330). This parameter enables to perform the analyses taking into account the resistance of tensile concrete.

- ***Effective Length along the X_1 (Y_1) Axis***

The design codes provide clear rules for taking into account buckling (second order effects) for bar elements, while the rules for plates are not so clear. Therefore, allowing for these effects in the program is optional. If the user has specified the effective length for the X_1 axis (similarly for Y_1), then if there is a compressive force N_x, the moment M_x will be increased by the buckling coefficient η.

- ***Random Eccentricity (Geometric Imperfections)***

The design codes provide clear rules for taking into account geometric imperfections (random eccentricity) for bar elements, while the rules for plates are not so clear. Therefore, allowing for these effects in the program is optional. When the analysis is performed according to EN 1992 and there is a compressive force N_x (similarly for N_y), a minimum eccentricity is used equal to e_0=max(h/30; 20 mm), where h is the plate thickness. When the analysis is performed according to SNiP, DBN, SP – e_0=max(L/600;h/30; 10 mm), where L is the respective effective length specified by the user (see above). If the effective lengths are not specified during the analyses according to these codes, then the value of the minimum eccentricity will be calculated without taking into account the L/600 term.

- ***Stress Limitation***

When performing the serviceability limit state analysis according to EN 1992-1-1 (Sec. 7.2) or DBN B.2.6-98:2009 (Sec. 5.2 of DSTU B B.2.6-156-2010) the stress shall be limited in some cases in order to avoid longitudinal micro-cracks. This option enables to calculate the respective factors during the design of reinforced concrete structures.

- ***Characteristics of materials under accidental actions***

When calculating for the action of accidental loads, by default characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017). Since this section was excluded by Change No. 2 to SP 296.1325800.2017, the user can (using the appropriate checkbox) choose which material properties (characteristic or design) to use.

7.4 Check

The check modes are used to perform a check of sections of reinforced concrete structural members for compliance with requirements of SNiP 2.03.01-84*, KMK 2.03.01-96 by the following factors[1]:

- ultimate moment strength of the section — Sec. 3.15–3.20, 3.27–3.28;
- strength for the ultimate longitudinal force of the section — Sec. 3.26, 3.28;
- longitudinal force with the deflection taken into account for slenderness L_0 / i >14 — Sec. 3.24, 3.6;
- strength for the ultimate longitudinal force with the reinforcement over the section height taken into account — Sec. 3.64 of the Guide to SNiP 2.03.01-84*;
- moment resisted by the section during cracking — Sec. 4.5;
- crack opening width (short-term) — Sec. 4.14, 4.15;
- crack opening width (long-term) — Sec. 4.14, 4.15;
- opening width of oblique cracks (short-term) — Sec. 4.17;
- opening width of oblique cracks (long-term) — Sec. 4.17;
- stresses in the transverse reinforcement — Sec. 4.17;
- strength in an oblique strip between oblique cracks — Sec. 3.30;
- strength for oblique sections with no transverse reinforcement — Sec. 3.32;
- strength in an oblique crack — Sec. 3.31 of SNiP, Sec. 3.31 of the Guide to SNiP 2.03.01-84*[2];

[1] The analysis for resistance to torque is based on formulas suggested by Moersch and Rausch (see [50]).

[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.

- strength accounting for the resistance of concrete in the tension area — Sec. 3.5, 3.8;
- strength of a section under torque;
- resistance of reinforcement to torque;
- resistance of side reinforcement to torque;
- resistance of transverse reinforcement to torque.

It should be noted that SNiP 2.03.01-84* provides two variants of strength analysis for flexural members: one for a so-called general case (Sec. 3.28*) and one based on simplified formulas which are applicable only for a uniaxial stress state. When checking flexural members the application always uses the general case, except for the **Strength of RC Sections** mode with the **Area of reinforcement** option (see below).

The reason for this is as follows. For 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 these rebars — as a second row of the lower/upper reinforcement or as the side reinforcement (the discrepancy may be up to 10 - 20%).

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

- strength for the ultimate longitudinal force of the section — Sec. 3.1–3.5 of SNiP;
- longitudinal force with the deflection taken into account for slenderness L0 / i >14 — Sec. 3.6, 3.24;
- strength for the ultimate longitudinal force in the analysis out of the bending plane — Sec. 1.21, 3.2, 3.3 of SNiP;
- ultimate moment strength of the section — Sec. 3.8;
- strength accounting for the resistance of concrete in the tension area — Sec. 3.5, 3.8;
- strength in an oblique strip between oblique cracks — Sec. 3.30;
- strength for oblique sections with no transverse reinforcement — Sec. 3.32;
- lateral force when there are no oblique cracks — Sec. 4.4 of the Guide to SNiP 2.03.01-84*.

If the analysis is performed according to SP 52-101-2003, the check of sections of reinforced concrete structural members will use the following factors:

- strength for the ultimate longitudinal force of the section — Sec. 6.2.19;
- strength of concrete in tension;
- ultimate moment strength of the section — Sec. 6.2.9-6.2.15;
- strains in compressed concrete — Sec. 6.2.21-6.2.31;
- strains in the tensile reinforcement — Sec. 6.2.21-6.2.31;
- height of the tension area of concrete — Sec. 4.1.2a, 6.2.30;
- strains in concrete in tension — Sec. 6.2.30, 6.2.31, 7.2.11;
- longitudinal force with the deflection taken into account for slenderness $L_0/i>14$ — Sec. 6.2.16;
- strength accounting for the resistance of concrete in the tension area — Sec. 6.1.9, 6.1.12;
- moment resisted by the section during cracking — Sec. 7.2.1;
- crack opening width (short-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- crack opening width (long-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- strength for the ultimate longitudinal force of the section — Sec. 6.2.3, 6.2.8, 6.2.16, Sec. 3.50 of the Guide to SP 52-101-2003 (further, the Guide to SP);
- strength for the ultimate longitudinal force with the reinforcement over the section height taken into account — Sec. 6.2.17, Sec. 3.58 of the Guide to SP;
- strength in a concrete strip between oblique sections — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength for the lateral force resisted only by concrete — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- strength for oblique sections with no transverse reinforcement — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;
- strength for an oblique section — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;

- lateral force when there are no oblique cracks — Sec. 4.28 of the Guide to SP;
- strength in a concrete strip between oblique sections for Q_z — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength in a concrete strip between oblique sections for Q_y — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength of a section under torque — Sec. 6.2.37;
- resistance of reinforcement S_1 to torque;
- resistance of reinforcement S_2 to torque;
- resistance of side reinforcement to torque;
- resistance of transverse reinforcement S_{Wz} to torque;
- resistance of transverse reinforcement S_{Wy} to torque;
- element strength between spatial sections — Sec. 6.2.41.

The sections of concrete structural members are checked using the following factors:

- strength for the ultimate longitudinal force of the section — Sec. 6.1.9-6.1.10;
- ultimate moment strength of the section — Sec. 6.1.12;
- strength of concrete in tension;
- strains in compressed concrete — Sec. 6.2.21–6.2.31;
- height of the tension area of concrete — Sec. 4.1.2a, 6.2.30;
- strains in concrete in tension — Sec. 6.2.30, 6.2.31, 7.2.11;
- longitudinal force with the deflection taken into account for slenderness L0/i>14 — Sec. 6.2.16;
- strength accounting for the resistance of concrete in the tension area — Sec. 6.1.9, 6.1.12;
- moment resisted by the section during cracking — Sec. 7.2.1;
- crack opening width (short-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- crack opening width (long-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- strength for the ultimate longitudinal force of the section — Sec. 6.2.3, 6.2.8, 6.2.16, Sec. 3.50 of the Guide to SP;
- strength in a concrete strip between oblique sections — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength for the lateral force resisted only by concrete — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- strength for oblique sections with no transverse reinforcement — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- strength for an oblique section — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;
- lateral force when there are no oblique cracks — Sec. 4.28 of the Guide to SP;
- strength in a concrete strip between oblique sections for Qz — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength in a concrete strip between oblique sections for Qy — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- strength of a section under torque — Sec. 6.2.37.

If the analysis is performed according to SP 63.13330, the check of sections of reinforced concrete structural members will use the following factors:

- strength for the ultimate longitudinal force of the section — Sec. 8.1.18;
- strength of concrete in tension;
- ultimate moment strength of the section — Sec. 8.1.8-8.1.14;
- strains in compressed concrete — Sec. 8.1.20-8.1.30;
- strains in the tensile reinforcement — Sec. 8.1.20-8.1.30;
- height of the tension area of concrete — Sec. 8.1.29;
- strains in concrete in tension — Sec. 8.1.29, 8.1.30, 8.2.14;
- longitudinal force with the deflection taken into account for slenderness L0/i>14 — Sec. 7.1.11, 8.1.15;
- strength accounting for the resistance of concrete in the tension area — Sec. 7.1.9, 7.1.12;
- moment resisted by the section during cracking — Sec. 8.2.4, 8.2.11;
- crack opening width (short-term) — Sec. 8.2.6, 8.2.15, 8.2.16;

- crack opening width (long-term) — Sec. 8.2.6, 8.2.15, 8.2.16;
- strength for the ultimate longitudinal force of the section — Sec. 8.1.15, 8.1.18;
- strength for the ultimate longitudinal force with the reinforcement over the section height taken into account — Sec. 8.1.16;
- strength in a concrete strip between oblique sections — Sec. 8.1.32, 8.1.34;
- strength for the lateral force resisted only by concrete — Sec. 8.1.33, 8.1.34;
- strength for oblique sections with no transverse reinforcement — Sec. 8.1.33, 8.1.34;
- strength for an oblique section — Sec. 8.1.33, 8.1.34;
- strength in a concrete strip between oblique sections for Qz — Sec. 8.1.32, 8.1.34;
- strength in a concrete strip between oblique sections for Qy — Sec. 8.1.32, 8.1.34;
- strength of a section under torque — Sec. 8.1.37;
- resistance of reinforcement S1 to torque;
- resistance of reinforcement S2 to torque;
- resistance of side reinforcement to torque;
- resistance of transverse reinforcement SWz to torque;
- resistance of transverse reinforcement S_{Wy} to torque;
- element strength between spatial sections — Sec. 8.1.41.

The sections of concrete structural members are checked using the following factors:

- strength for the ultimate longitudinal force of the section — Sec. 7.1.9-7.1.10;
- ultimate moment strength of the section — Sec. 7.1.12;
- strength of concrete in tension;
- strains in compressed concrete — Sec. 8.1.20–8.1.30;
- height of the tension area of concrete — Sec. 8.1.29;
- strains in concrete in tension — Sec. 8.1.29, 8.1.30, 8.2.14;
- longitudinal force with the deflection taken into account for slenderness $L_0/i>14$ — Sec. 7.1.11, 8.1.15;
- strength accounting for the resistance of concrete in the tension area — Sec. 7.1.9, 7.1.12;
- moment resisted by the section during cracking — Sec. 8.2.4, 8.2.11;
- crack opening width (short-term) — Sec. 8.2.6, 8.2.15, 8.2.16;
- crack opening width (long-term) — Sec. 8.2.6, 8.2.15, 8.2.16;
- strength for the ultimate longitudinal force of the section — Sec. 8.1.15;
- strength in a concrete strip between oblique sections — Sec. 8.1.32, 8.1.34;
- strength for the lateral force resisted only by concrete — Sec. 8.1.33, 8.1.34;
- strength for oblique sections with no transverse reinforcement — Sec. 8.1.33, 8.1.34;
- strength for an oblique section — Sec. 8.1.33, 8.1.34;
- strength in a concrete strip between oblique sections for Qz — Sec. 8.1.32, 8.1.34;
- strength in a concrete strip between oblique sections for Qy — Sec. 8.1.32, 8.1.34;
- strength of a section under torque — Sec. 8.1.37.

If SP 52-101-2003 or SP 63.13330 is selected, the analysis of normal sections and the calculations for serviceability limit state are performed on the basis of a non-linear deformation model, and the diagrams of concrete states are selected in compliance with the regulations of Sec. 5.1.21-5.1.24 of SP 52-101-2003 and Sec. 6.1.23-6.1.26 of SP 63.13330. Diagrams described in Sec. 5.2.11 of SP 52-101-2003 and Sec. 6.2.13 of SP 63.13330 are used as the diagrams of reinforcement behavior. The only exception is the **Strength of Sections** mode when the **Area of reinforcement** checkbox is checked. In this case the limit forces method is used.

Unlike SP 52-101-03 (Guide to this SP), SP 63.13330 does not specify the condition of the formation of oblique cracks. However, the program still performs this check provided that the 1st category of crack resistance is selected. The analysis is performed in accordance with the requirements of Sec. 4.28 of the Guide to SP 52-101-03.

When SP 52-101-2003 or SP 63.13330 is selected, the program calculates the following factors:

- Strength for the ultimate longitudinal force;
- Ultimate moment strength.

The first one is the ratio $\frac{|N|}{R_s A_{s,tot}}$ for the elements in tension and $\frac{|N|}{R_s A_{s,tot} + R_b A}$ for the elements in compression. Although technically there is no requirement to check this factor in the codes (with the exception of Sec. 6.2.19 of SP 52-101-2003 and Sec. 8.1.18 of SP 63.13330), in fact without verifying the fact that this value is less than one we cannot perform other calculations, for instance, calculate the coefficient φ_n in Sec. 8.1.34 of SP 63.13330. The *ultimate moment strength* has been introduced for similar reasons. It is a number which has to be divided by N, M_y, M_z, in order to obtain the maximum allowable values of the deformations of concrete or reinforcement.

SNiP method of strength analysis of oblique sections is focused on the work with a diagram of shear forces, using which it is possible to obtain a "shear force in an oblique section with the length of the projection C on the longitudinal axis of the element determined from all external forces, located on one side of the considered oblique section". It can be easily done in the case of one loading. However, if there are many variants of a loading, we have their envelope which is not a resultant of all external forces. No recommendations are given in the codes for this case. Therefore, the values of shear forces in the design sections are used to increase the reliability.

The procedure for the calculation of circular and ring cross-sections for the action of a transverse force is not provided in the codes. The formal analysis of the SNiP formulas makes it evident that two values b and h_0 are required. The h_0 value is clearly defined in the codes, but it is not clear what is the width of the section. The calculation in SCAD is performed in the following way. When the value $b \times h_0$ is required in the calculation for the action of a transverse force, a value equal to the area of a section with an outer radius R-a is used, where R is the outer radius of the considered cross-section, and a is the distance to the center of gravity of the reinforcement.

When the analysis is performed according to SP 52-101-03, Sec. 3.52 of the Guide specifies how to take into account the effect of a longitudinal force in the calculations for transverse forces. It should be noted that the longitudinal reinforcement is not taken into account in Sec. 3.52 in the formula for the value N_b=1,3$R_b A$. There is a letter from the authors of SP which describes an alternative method for calculating the value N_b taking into account the longitudinal reinforcement ($N_b = R_b A + R_s A_{s,tot}$). There is a special **Analysis** tab in the **Arbat** settings where the user can check the checkbox prompting the application to use this formula to determine N_b. Since there are two methods for calculating N_b (according to SP and according to the letter), the program calculates N_b in two different ways and uses the highest value.

7.4.1 Strength of RC Sections

This mode implements a function of determining the load-bearing capacity of any section available in the application depending on the arrangement, diameter (area), and class of reinforcement, the class of concrete, the conditions of operation, and the allowable crack opening width.

In the general case, the analyses are performed for the action of a longitudinal force, bending moments, torque, and shear forces acting in the principal planes of inertia.

SNiP 2.03.01-84* does not regulate any checks of reinforced concrete members for the serviceability limit states under the action of moments in two planes. Therefore, if the crack resistance mode is active (the selected design code being SNiP 2.03.01-84*), only the eccentric compression/tension with the eccentricity in one plane is considered.

General

The **General** tab depends on whether the **Area of reinforcement** checkbox of the **Strength of RC Sections** button of the main window is checked. If the checkbox is not checked, the information about the rebar arrangement will be used as the initial data, and the following information will be specified in the **General** tab (Fig. 7.4.1-1):

- the geometric and effective lengths of members;
- the random eccentricities;
- the shape and sizes of the section;

- the thickness of the concrete cover of the reinforcement;
- the service factors for reinforcement;
- the class of longitudinal reinforcement (reinforcement can be placed in two rows);
- the class of transverse reinforcement;
- the limit slenderness.

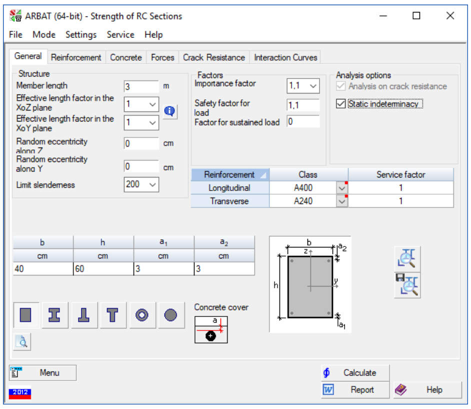


Figure 7.4.1-1. *The* **General** *tab* (the **Area of reinforcement** checkbox is not checked)

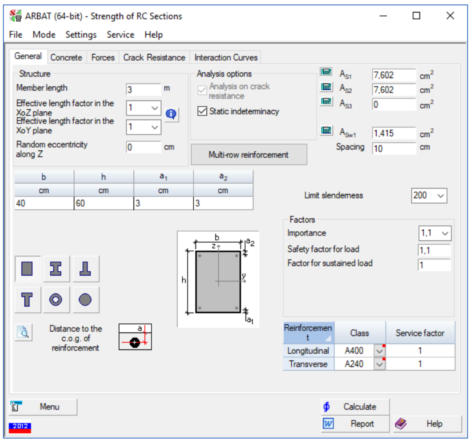


Figure 7.4.1-2. *The* **General** *tab* (the **Area of reinforcement** checkbox is checked)

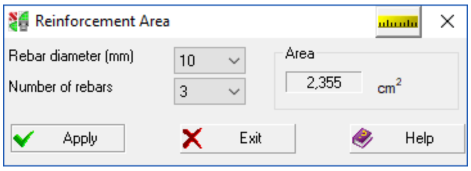


Figure 7.4.1-3. *The dialog box of the* **Reinforcement area** *calculator*

Checkboxes are used to indicate that a check has to be performed taking into account the serviceability limit state (**Analysis on crack resistance**), and that a section belongs to a statically determinate structure.

SP 52-101-2003 and SP 63.13330 enable to perform a check taking into account the crack resistance under moments in two planes, therefore this checkbox is always checked for these design codes (Fig. 7.4.1-1).

The analysis takes into account the specified importance factor, safety factor for load, and factor for sustained load.[3] The safety factor for load and the factor for sustained load specified in the first tab are used when plotting the interaction curves.

It should be noted that the value of the *factor for sustained load* can be even less than 0 or greater than 1. It can be seen from the following example. Suppose a certain combination includes a long-term loading with the longitudinal force of 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 these two loadings produces the overall longitudinal force of 0.9T, and the sustained part of this combination results in a longitudinal force of 1.0T. Thus the *factor for sustained load* 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 concrete cover are specified according to the rules described in Section 7.3.1.

The effective length factors are specified according to Sec. 3.25 of SNiP 2.03.01-84*, KMK 2.03.01-96 (Sec. 6.2.18 of SP 52-101-2003, Sec. 8.1.17 of SP 63.13330.2012). The button  enables to obtain the reference data on the effective lengths according to Sec. 3.25 of SNiP 2.03.01-84*, KMK 2.03.01-96 (Sec. 6.2.18 of SP 52-101-2003, Sec. 8.1.17 of SP 63.13330).

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*, KMK 2.03.01-96 (Sec. 4.2.6 of SP 52-101-2003, Sec. 7.1.7, 8.1.7 of SP 63.13330). If a nonzero random eccentricity is specified, then the greater of the two values will be used: either 1 cm or the specified value (according to Sec. 3.50 of the Guide to SNiP 2.03.01-84* and Sec. 3.6 of the Guide to SP 52-101-2003 and Sec. 7.1.7, 8.1.7 of SP 63.13330).

A standard group of controls and the **Reinforcement** tab are used to specify the characteristics of reinforcement and the information on the arrangement of rebars (see Section 7.3.3).

[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...». ARBAT takes into account this part of the loads by applying a *factor for sustained load*.

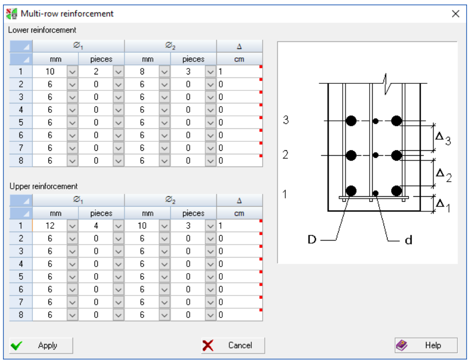


Figure 7.4.1-4. *The* **Multi-row reinforcement** *dialog box*

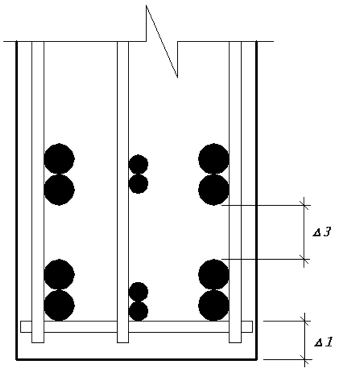


Figure 7.4.1-5. *Arrangement of rebars of different diameters*

If the **Area of reinforcement** checkbox of the **Strength of RC Sections** button of the main window is checked, then the initial data will include areas of longitudinal and transverse reinforcement (the **General** tab shown in Fig. 7.4.1-2). Moreover, instead of the thickness of the concrete cover, you have to specify the **distances to the centers of gravity** of reinforcement here.

The area of reinforcement can be calculated using the **Reinforcement area** calculator (Fig. 7.4.1-3). The calculator is invoked by clicking one of the buttons , on the left from the respective text fields.

To calculate the reinforcement area with the calculator, select a line with the rebar diameter from the top list and specify the number of rebars in the bottom list. Clicking the **Apply** button will close the calculator and the calculated value of the area will be entered into the respective text field of the **General** tab.

The tab contains the **Multi-row reinforcement** button that enables to enter data for the case of the multi-row reinforcement. Clicking this button will open a **Multi-row reinforcement** dialog box, which is shown in Fig. 7.4.1-4.

This dialog is used to determine the distance from the center of gravity of the longitudinal reinforcement section to the nearest face of the 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 (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. 7.4.1-5). The rebars with different diameters in the next rows are assumed to have their centers lying on one straight line.

The table is used to specify the diameter of rebars and their number in the current row, and the clear distance between the rebars of this row and those of the preceding row. The value of the concrete cover is specified for the rebars of the first row instead of the clear distance. If the distance between the rebars is set to zero ($\Delta = 0$), they are assumed to be adjacent to each other.

Once the **Multi-row reinforcement** dialog is closed by clicking the **Apply** button, the calculated results appear in the **Distance to the c.o.g. of reinforcement** text fields in the **General** tab. To edit the data, you can open the **Multi-row reinforcement** dialog box again, where you will find the information last entered.

Having prepared the initial data, you can start the analysis by clicking the **Calculate** button.

Figs. 7.4.1-6–7.4.1-8 show arrangements of longitudinal and transverse reinforcement and the legend of reinforcement used in drawings of various sections.

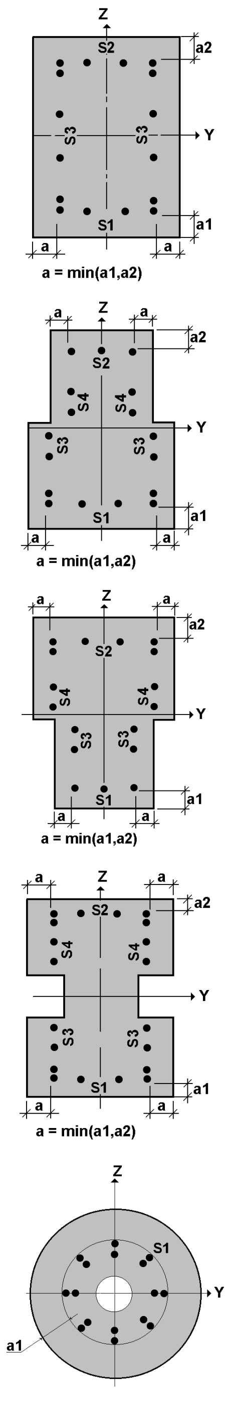


Figure 7.4.1-6. *Arrangements of longitudinal reinforcement*

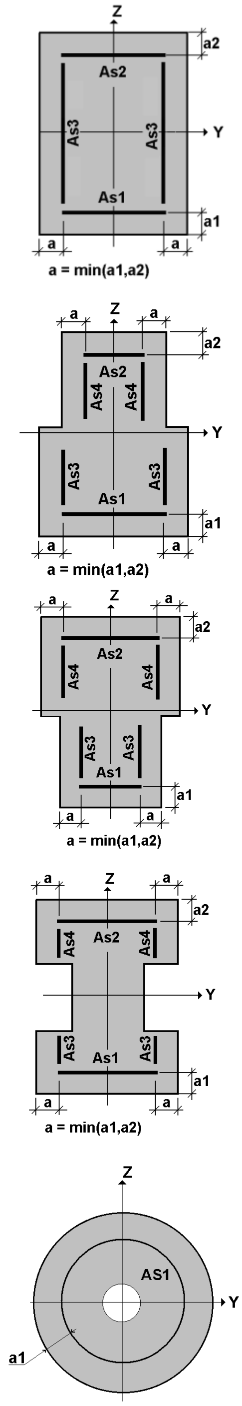


Figure 7.4.1-7. *"areas" of longitudinal reinforcement*

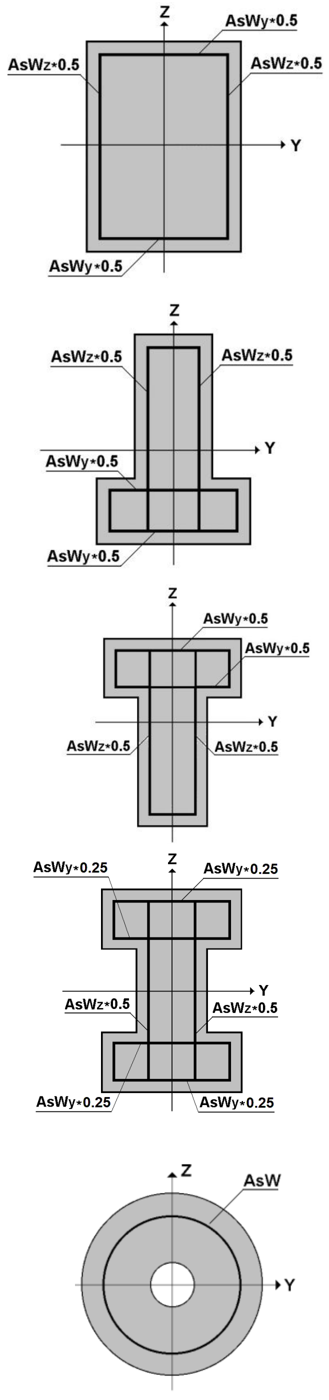


Figure 7.4.1-8. *Arrangements of transverse reinforcement at the known areas of transverse reinforcement*

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Forces

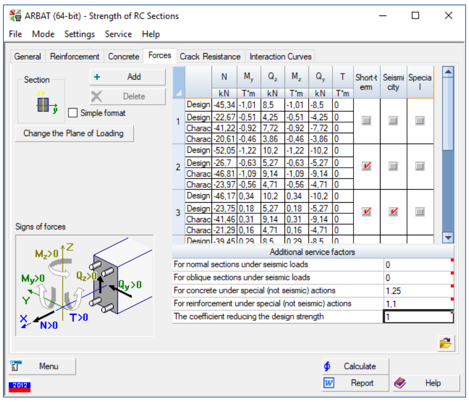


Figure 7.4.1-9. *The* **Forces** *tab*

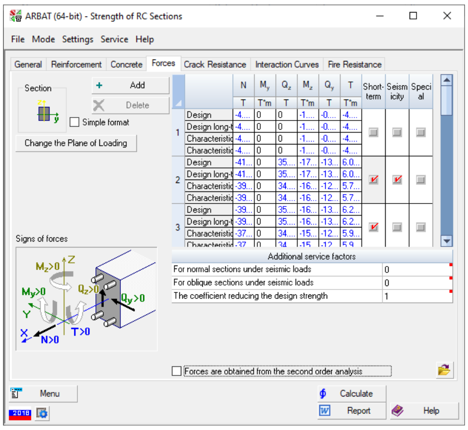


Figure 7.4.1-10. *The* **Forces** *tab (simple form)*

The **Forces** tab (Fig. 7.4.1-9, 7.4.1-10) is used to enter forces acting in a cross-section of the member. It shows a cross-section with principal axes of inertia and positive directions of forces. The tab contains a table used to specify forces which are acting in the section and are caused by one or more load cases.

The table can function in an expanded or simplified mode (the **Simple form** checkbox is used to switch between these modes). The design values of forces, safety factor for load and the factor for sustained load are specified for each load case in the simplified mode. In the expanded mode you have to specify the design, design long-term, characteristic and characteristic long-term values of forces for each load case. Moreover, the respective checkboxes can be used to specify each load case as seismic and/or short-term.

If the calculation is performed according to SP 16.13330, then the **Special** checkbox can be checked for a certain loading (a special non-seismic loading). In this case you will be able to specify a service factor for concrete and reinforcement (longitudinal and transverse) in accordance with Annex B of SP 296.1325800.2017 in the special table (moreover, the importance factor γ_n = 1.0 in accordance with Sec. 5.5 of SP 296.1325800.2017 will be used for such loadings). Furthermore, an additional service factor which reduces the design strength values and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017 can be taken into account.

When calculating for the action of accidental loads, by default characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017). Since this section was excluded by Change No. 2 to SP 296.1325800.2017, the user can (using the Advanced Settings) choose which material properties (characteristic or design) to use.

It should be noted that the codes do not provide data on the characteristic resistance of the transverse reinforcement. Therefore, the following value is used in the analysis $R_{sw}\ \gamma_s$,γ_s=1,15 (safety factor for reinforcement). The characteristic resistance of the compressive resistance of the longitudinal reinforcement is determined in a similar way.

In the case of seismic analysis, the user can specify the reduction factor of limit relative height of the compressed area, which is provided in some seismic codes (see, for example, Sec. 6.7.2 of SP 14.13330). Its default value is 0.85. The user must/can enter the value of this factor due to the fact that the codes provide its value only for some particular cases (design seismicity of 7, 8 or 9). The codes do not provide any recommendations for the cases when the seismicity is not expressed as an integer or it is expressed as a number greater than 9.

The table can be also filled by importing data from **SCAD** which describe design combinations of forces (DCFs) (if the combination includes a special (non-seismic) loading, then the characteristic and characteristic long-term forces/stresses will automatically be assumed equal to zero, since the serviceability limit state analysis does not have to be performed in this case). A file with the **.rsu** extension is created in the **Element Information** mode of **SCAD** (version 11) and can be imported by clicking the button . If **SCAD** (version 21) is used, a file with the **.rsu2** extension is created and is imported in the same way.

In this mode, the forces acting in the section of the member are specified on the basis of a static analysis performed externally, therefore you have to indicate the type of the design model used to calculate the forces — 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). In the case of a nonlinear analysis the **Forces are obtained from the second order analysis** checkbox should be checked and second-order effects (the effect of buckling) will not be taken into account.

The **Strength of RC Sections** mode provides a capability of changing the load plane (the **Change the Load Plane** button clicking which will replace M_y by M_z and Q_y by Q_z).

Crack Resistance

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

Interaction Curves

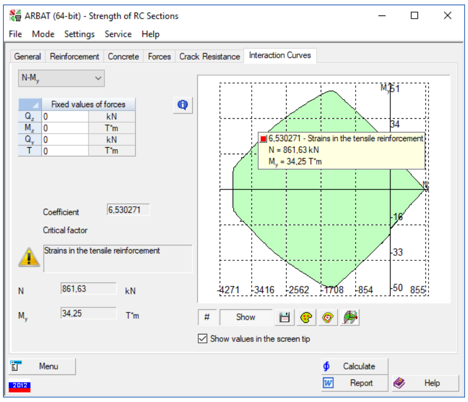


Figure 7.4.1-11. *The* **Interaction Curves** *tab*

Results of the analysis are displayed in the **Interaction Curves** tab (Fig. 7.4.1-11) and are represented as a curve enclosing an area of load-bearing capacity under the action of a pair of forces selected by the user. The pair of forces is assumed to be allowable if the load-bearing capacity utilization factor for the section is $K_{max} \le 1$. All the other forces are assumed to be equal to the values specified in the **Fixed values of forces** group.

When the check is performed according to SNiP 2.03.01-84* and the analysis takes into account the crack resistance, two pairs of forces 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 for the latter pair will be displayed only if the diameter and the spacing of transverse reinforcement have been specified in the **General** tab.

If the analysis does not take crack resistance into account or the checks have been performed according to SP 52-101-2003 or SP 63.13330, then the interaction curve can be plotted for the following pairs of forces:

N–M_y; N–M_z; M_y–M_z; N–Q_z; N–Q_y.

Using your mouse pointer, you can explore the area of the forces variation shown in the graph. Every position of the pointer corresponds to a pair of numerical values of forces, which are displayed in the fields on the left from the graph. The dialog also displays the maximum value of the utilization factor that corresponds to these values and the type of check in which it takes place.

Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the position of the pointer on the interaction curve.

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

7.4.2 Strength of Concrete Sections

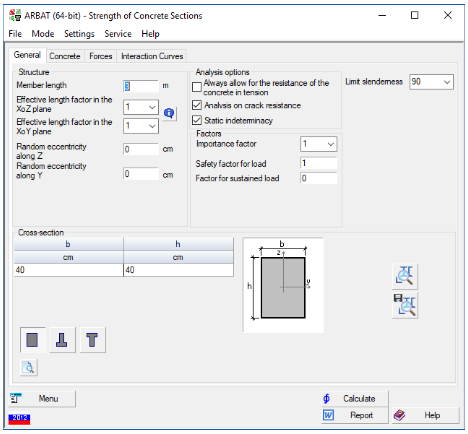


Figure 7.4.2-1. *The* **General** *tab*

This mode is similar to the **Strength of RC Sections** mode. A natural difference between them is that there is no need to specify the data on reinforcement (Fig. 7.4.2-1). The current version of the software implements the analysis of rectangular and T-sections.

Moreover, a special type of analysis is available which can be activated by the **Always allow for the resistance of the 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 to tension to be taken into account only in the case of the pure bending. However, on the other hand, if we followed the SNiP regulations directly, there could be a paradoxical situation when even a negligibly small longitudinal force would make it impossible to take into account the resistance of concrete to tension and thus there would be a stepwise decrease of the load-bearing capacity. We suggest the users to make their own decision whether to use this nonstandard approach or not.

It should be noted that it is required by the design codes to use the reduction factor γ_{b9} = 0,9 at the analysis of concrete sections. **Arbat** does not automatically multiply the respective parameters by the factor of 0,9. The user must take into account this requirement when specifying the value *Resulting factor without* γ_{b2} in the **Concrete** tab.

7.4.3 Beam Check

This mode enables to check the strength and crack resistance of a multi-span continuous beam of a constant cross-section in compliance with regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SP 52-101-2003, SP 63.13330). Bending of the beam in one load plane under distributed and concentrated loads is considered. Loads are combined into load cases which can be classified by their physical origin and properties into permanent, long-term, short-term, wind, and snow. Checks of all sections are performed for the automatically created design combinations of forces (DCF). DCF factors which take into account the nature of the load case are assigned by the application in compliance with the regulations of SNiP 2.01.07-85*. If the selected design code is SP 63.13330, then the loads are combined according to the regulations of SP 20.13330.

It is assumed that the beam is not subjected to longitudinal forces and only the following force actions are taken into account:

M — bending moment;

Q — shear force.

The analysis can be performed for a beam of a rectangular section, T-section or I-section. A particular arrangement of rebars should be specified. The number and diameter of rebars can be different on different segments along the beam length.

Data are prepared in the following tabs: **General**, **Loads**, **Concrete**, **Crack Resistance**, and **Segments**, and the results are analyzed in the **Results** tab.

General

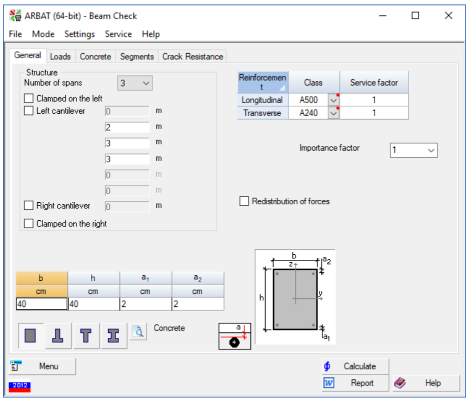


Figure 7.4.3-1. *The* **General** *tab*

The **General** tab (Fig. 7.4.3-1) is used to specify the number and lengths of the spans, the class and service factors for longitudinal and transverse reinforcement, the type and sizes of the section and the thickness of the concrete cover.

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

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

- set a number of spans in the **Number of spans** list (not more than five, not counting the cantilevers);
- the cantilevers (if there are any) are specified by checking the **Left cantilever** and/or **Right cantilever** checkboxes;
- enter the lengths of the spans and cantilevers in the respective text fields.

The shape of the section is selected according to the rules described in Section 7.3.1.

A special checkbox, Redistribution of forces**, enables to switch to a mode where the diagrams of moments and shear forces are calculated taking into account the redistribution of forces. The recommendation (see [13]) that sets the maximum level of moment (force) redistribution equal to 30% is taken into account. It should be noted that the phenomenon of redistribution deals with the formation of plastic hinges, i.e. an unlimited crack opening width. Therefore, when the** Redistribution of forces **checkbox is checked, the analysis on crack resistance is not performed.**

Loads

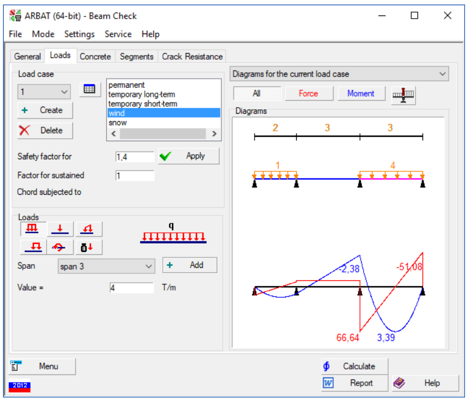


Figure 7.4.3-2. *The* **Loads** *tab*

The check of the specified reinforcement is performed for the DCFs formed from multiple load cases. The load cases are entered in the **Loads** tab (Fig. 7.4.3-2) following these steps:

- click the **Create** (a load case) button;
- select a type of the load case from the list (permanent, temporary long-term, temporary short-term, wind, or snow);
- specify the load type (click a button with a distributed or concentrated load);
- enter the design value of the load;
- specify the factor for sustained load for snow and short-term loads;
- specify the safety factor for load;
- use the **Span** drop-down list to select a loaded span or a cantilever (the selected span will be highlighted in red);
- click the **Add** button;
- create and add other loads included in the current load case.

The application also enables to specify the load from the self-weight of the beam (button ).

Depending on the type of the load, its properties may include:

- for distributed loads — the load intensity;
- for a load distributed over a part of the span — value of the load, distance from the beginning of the bar to the beginning of the application area, width of the load application area;
- for a concentrated force — the value of the force and its position in the span;

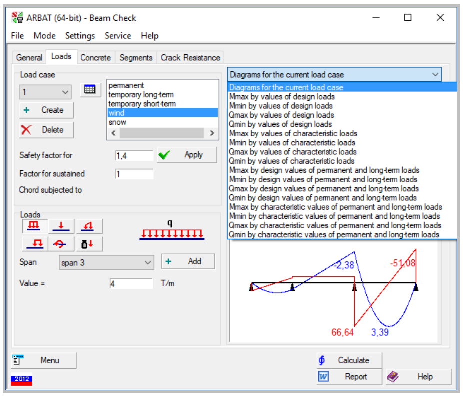


Figure 7.4.3-3. *The* **Loads** *tab with the opened list of combinations*

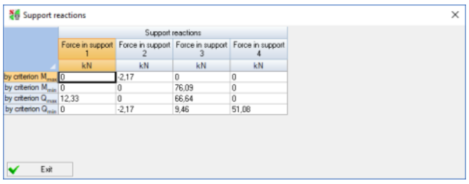


Figure 7.4.3-4. *The* **Support reactions** *dialog box*

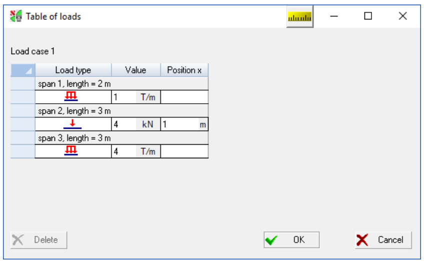


Figure 7.4.3-5. *The* **Table of loads** *dialog box*

- for a concentrated moment — the value of the moment and its position in the span;
- for a trapezoid load — value of the load at the beginning of the application area, distance from the beginning of the bar to the beginning of the application area, width of the load application area, value of the load at the end of the application area.

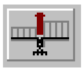
No additional data are required to specify the self-weight load.

All the specified loads will belong to the current load case until the next one is created. It should be noted that the **Factor for sustained load** and the **Safety factor for load** relate to the whole load case, and the calculation of DCF will use the last entered value. Having changed one of these factors, click the **Apply** button.

To delete the current load case, click the **Delete** button.

To edit a previously created load case, select its number from the list of numbers of load cases.

If the **Diagrams for the current load case** item is selected in the **Diagrams** field (in the list of displayed factors), then entering every new load will be followed by displaying the load arrangement and the diagrams of moments and shear forces for the current load case.

The **All**, **Force**, and **Moment** buttons enable you to select a mode for displaying the diagrams: diagrams of moments and shear forces together, only shear forces, and only moments respectively. Furthermore, clicking the button  will display the design values of support reactions (Fig. 7.4.3-4).

In addition to the diagrams of moments and shear forces for each of the considered load cases, the application calculates forces which can appear in every section of the beam from the combination of the specified loads. The rules for combining comply with SNiP 2.01.07-85*. The list of these combinations (Fig. 7.4.3-3) is located above the diagrams field and contains:

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

These combinations are determined for either design or characteristic loads, and also for the case when only permanent and long-term loads are applied at their design or characteristic values.

If you place the mouse pointer in the diagram field, values of the moment and shear force in a particular cross-section corresponding to the position of the pointer will be displayed (Fig. 7.4.3-2).

If you want to change the load value or delete the load from a load case, use the table of loads (the button  in the **Load case** group). The **Table of loads** dialog box (Fig. 7.4.3-5), which opens by clicking this button, displays the type, the value, and the position of the load. Changes made to the load parameters will be saved after you exit the table by clicking the **OK** button.

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Crack Resistance

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

Segments of a Beam

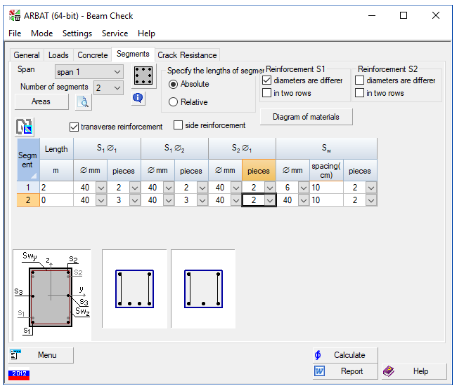


Figure 7.4.3-6. *The* **Segments** *tab*

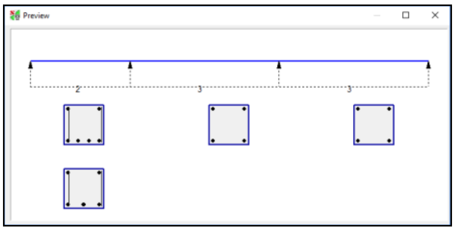


Figure 7.4.3-7. *The* **Preview** *dialog box*

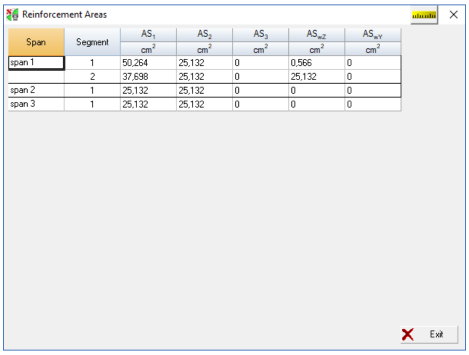


Figure 7.4.3-8. *The* **Reinforcement areas** *dialog box*

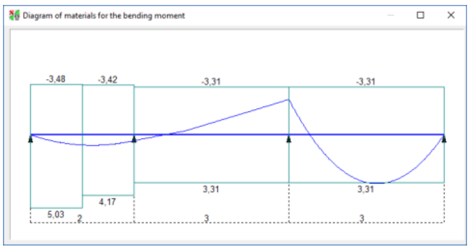


Figure 7.4.3-9. *Diagram of materials*

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 be constant 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. Initial data concerning the arrangement of reinforcement in the segments are specified in the **Segments** tab (Fig. 7.4.3-6). This information is entered separately for each span (cantilever). The dialog box contains the following groups of data:

- **Span**, where the number of segments for each span or cantilever is selected from the drop-down list;
- **Specify the lengths of segments**, where you have to define the way the lengths are specified using the following radio buttons:
  Absolute — the lengths of the segments will be specified in the units of length (the units of length for the spans are defined in the **Application Settings** dialog box);
  Relative — the lengths of the segments will be specified as percentage of the total span length;
- **Reinforcement S_1** — using the checkboxes of this group you can select the following options:
  Diameters are different — the rebars of the same row of the lower reinforcement have different diameters;
  In two rows — the lower reinforcement is installed in two rows (different diameters are allowed only in the first row of the lower reinforcement). If the **In two rows** checkbox is checked, you have to specify the clear distance between the rows of reinforcement (always in mm) in the respective text field;
- **Reinforcement S_2** — is used to describe the upper reinforcement and is similar to the **Reinforcement S_1** group;
- the **Transverse reinforcement** checkbox — the checked checkbox indicates that there is transverse reinforcement in the segment;
- the **Side reinforcement** checkbox — the checked checkbox indicates that there is side reinforcement in the segment;
- the table with the parameters of reinforcement, where the data are entered for each particular span selected from the **Span** list. The number of columns in the table depends on settings for the current span, and the number of lines depends on the number of segments this span is divided into.

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

The beam reinforcement layout thus specified can be verified in the **Preview** dialog box (Fig. 7.4.3-7), which opens after clicking the button . Reference information with examples of beam reinforcement can be obtained using the button .

"Uniform" reinforcement can be specified for beams of rectangular cross-section using the button (see Section 7.3.3).

The button enables to save the data on the longitudinal rebar arrangement in a format which can be used for importing the data into the AllPlan FT design software developed by Nemetchek. Clicking this button will open a standard Windows dialog box where you have to specify a folder and a filename. Once these data are selected, the information will be saved in a file with the **.a** extension.

Follow these steps to enter data in the **Segments** tab:

- select the name of a span (cantilever) from the **Span** list, for which the data will be entered;
- specify a number of segments for the selected span in the **Number of segments** list;
- select a method of length specification using the respective radio buttons in the **Specify the lengths of segments** group;
- check the checkboxes in the **Reinforcement S_1** and **Reinforcement S_2** groups to define the features of reinforcement in the span (if the reinforcement is arranged in two rows, specify the distance between the rows);
- check the **Transverse reinforcement** and/or **Side reinforcement** checkboxes if you want to specify transverse and/or side reinforcement;
- 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 specified the initial 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. 7.4.3-8), which appears after clicking the **Areas** button.

This tab also contains a button that invokes a dialog box displaying a diagram of materials (Fig. 7.4.3-9). The diagram is plotted under the assumptions that the safety factor for load is 1.1 and the factor for sustained load is 1.0.

Results

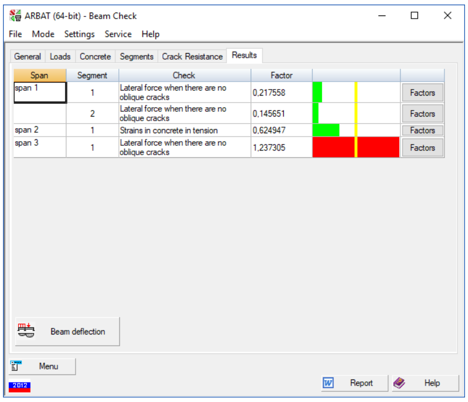


Figure 7.4.3-10. *The* **Results** *tab*

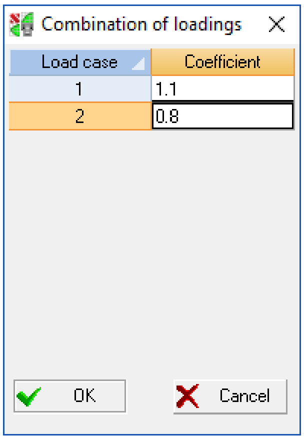


Figure 7.4.3-11. *The* **Combination of loadings** *dialog box*

This tab (Fig. 7.4.3-10) opens automatically once the check mode is activated (the **Calculate** button).

Results of the analysis are displayed in a table. The first and second columns of the table list spans and segments. For each segment, the **Check** column provides the name of a factor for which the utilization factor of restrictions achieves its maximum value, and the **Factor** column provides its value. The last column of the table contains a graphical representation of the factors, where red bars denote factors greater than one.

You can switch from this dialog box to the deflection analysis mode (by clicking the **Beam deflection** button) where the initial data are passed automatically. A combination of loadings specified in the check and reinforcement selection modes is used as the load for determining the deflections. The combination is specified in the respective dialog box (Fig. 7.4.3-11). 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 results of the calculation (the **Report** button).

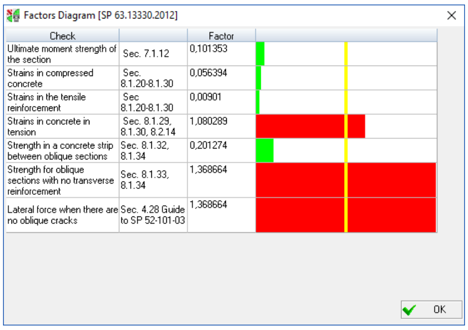


Figure 7.4.3-12. *The* **Factors Diagram** *dialog box*

Clicking the **Factors** button in the respective row of the table containing the results of the check of the segment opens the **Factors Diagram** dialog box (Fig. 7.4.3-12) with the detailed information about all checks performed for this segment and the respective factors.

7.4.4 Single-Span Beam Check

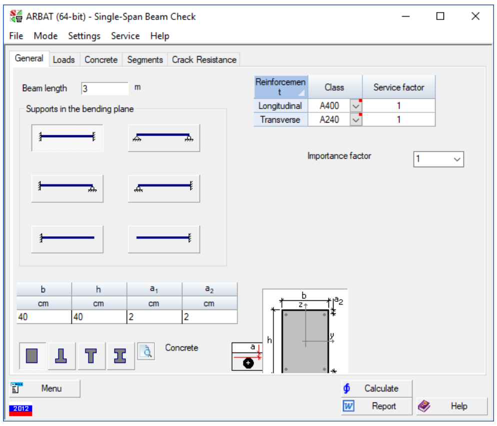


Figure 7.4.4-1. *The* **Single-Span Beam Check** *dialog box*

This multi-tab dialog box (Fig. 7.4.4-1) is used to perform a check of a single-span beam.

This mode is similar to the **Beam Check** mode (see Section 7.4.4). There are the following differences: only one length is specified for the beam, and the user has to select supports of the beam in its bending plane by clicking the respective button. Moreover, you do not have to specify the span to which the load is applied in the **Loads** tab.

7.4.5 Beam Deflection

This mode enables to calculate the deflections at the bending of a multi-span beam under a given load. The deflections are calculated for a rectangular section, T-section or I-section in compliance with Sec. 4.31 of SNiP 2.03.01-84*, KMK 2.03.01-96 (Sec. 7.3.4 of SP 52-101-2003, Sec. 8.2.21 of SP 63.13330). The curvature is determined taking into account cracks in the tension area according to Sec. 4.27 of SNiP 2.03.01-84*, KMK 2.03.01-96 (Sec. 7.3.7 of SP 52-101-2003, Sec. 8.2.24 of SP 63.13330). Initial data are prepared in the **General**, **Loads**, **Concrete**, **Conditions of Operation**, and **Segments** tabs, and results of the calculation are analyzed in the **Deflections** tab which opens once the calculation is performed (the **Calculate** button).

You can switch from the check and reinforcement selection modes for multi-span beams to the deflection mode. In this case the initial data specified in these modes will be transferred automatically.

General

The **General** tab is used to specify the number and lengths of the spans, the class and service factors for longitudinal and transverse reinforcement, the type and sizes of the section and the thickness of the concrete cover.

You also have to select (or enter) the maximum value of the ratio of the maximal deflection to the span length.

This tab is similar to the respective tab of the **Beam Check** mode (see Section 7.4.3).

Loads

The deflections are calculated for a single permanent load case that may include concentrated and distributed loads. The loads are specified in the respective tab, which is similar to the **Loads** tab of the **Beam Check** mode (see Section 7.4.3). The only difference is that since the analysis is nonlinear (it allows for the effect of cracks on the curvature), it is performed only for one load case.

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

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Conditions of Operation

Data on the conditions of operation are specified in the **Conditions of Operation** tab according to the rules described in Section 7.3.4.

Beam Segments

Segments of the beam are described in the respective tab according to the same rules as for the **Beam Check** mode (see Section 7.4.3).

Deflections

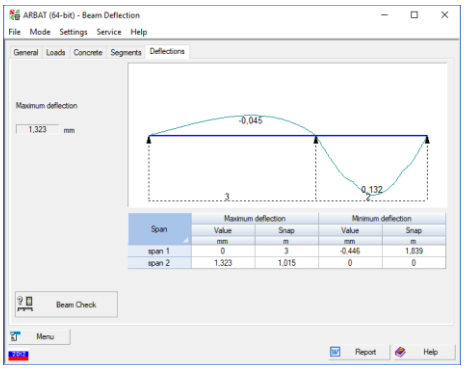


Figure 7.4.5-1. *The* **Deflections** *tab*

This tab (Fig. 7.4.5-1) opens automatically once the calculation mode is activated (the **Calculate** button). Results of the analysis will be displayed as a diagram or as a table (the **Table** button). The value of the maximum deflection is specified in the respective field. A report can be generated on the basis of the results of the calculation (the **Report** button); its generation is described in Section 2.5.

You can switch to the check mode by clicking the **Beam Check** button. All the data on the section, reinforcement and loads will be saved.

SNiP 2.03.01-84* (see Sec. 4.32-4.33) provides clear recommendations for taking into account the effect of shear forces on deflection. Other standards, for example, Sec. 8.2.21 of SP 63.13330, contain only the requirement to take into account shear angles without any specific instructions on how to determine them. Therefore, the shear is taken into account in the deflection analysis according to the recommendations of SNiP 2.03.01-84*.

7.4.6 Single-Span Beam Deflection

This mode is used to determine the deflections of a single-span beam. The mode is similar to the **Beam Deflection** mode (see Section 7.4.5). Peculiarities of the specification of the initial data are obvious — they follow from the difference between the structures of a single-span and a multi-span beam.

7.4.7 Column Check

This mode enables to check the strength and crack resistance of constant-section columns. An eccentric compression/tension with a biaxial eccentricity is considered. Checks of all sections are performed for the automatically created design combinations of forces (DCF). DCF factors which take into account the nature of the load case are assigned by the application in compliance with the regulations of SNiP 2.01.07-85*. If the selected design code is SP 63.13330, then the loads are combined according to the regulations of SP 20.13330.

SNiP 2-03-01-84* does not define any rules for the serviceability limit state analysis at the biaxial stress state. If this design code is selected, the following limitation takes place:

the check for the ultimate limit state takes into account the following force actions:

N — longitudinal force;

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

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

Q_z — shear force along the Z-axis;

Q_y — shear force along the Y-axis;

M_t — torque with its vector along the X-axis;

the checks for the ultimate and serviceability limit states take into account only the following force actions:

N — longitudinal force;

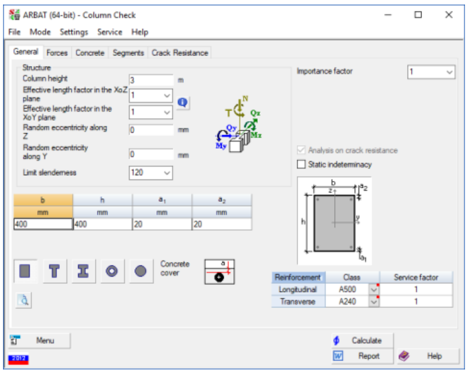
M_y — moment that bends the member in the XoZ-plane with its vector along the Y-axis;

Q_z — shear force along the Z-axis.

The analysis can be performed for a column of a rectangular section, T-section, I-section, ring, or round section. The arrangement of rebars in the section is assumed to be given and constant along the length of a particular segment. The user has to specify the number and lengths of segments the column is divided into.

Data are prepared in the following tabs: **General**, **Forces**, **Concrete**, **Segments**, and **Crack Resistance** tabs, and the results are analyzed in the **Results** tab.

General



Fogure 7.4.7-1. *The* **General** *tab*

The **General** tab (Fig. 7.4.7-1) is used to specify the height of a column, its effective length factors, values of the random eccentricity and the limit slenderness. Information on the section and the reinforcement classes is specified according to the rules described in Section 7.3.

Moreover, the **Analysis on crack resistance** checkbox defines whether the column should be analyzed for the serviceability 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 in the same way as in the **Strength of RC Sections** mode (see Section 7.4.1).

Forces

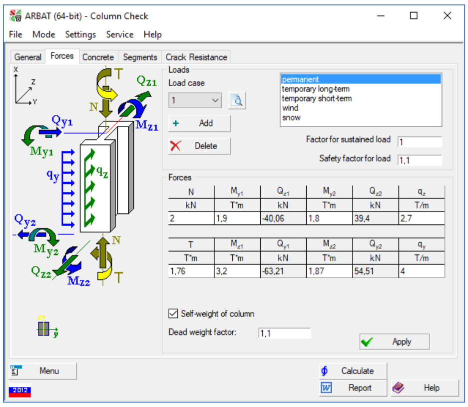


Figure 7.4.7-2. *The* **Forces** *tab*

The **Forces** tab (Fig. 7.4.7-2) is used to specify design values of forces applied as a longitudinal force and nodal moments in the end sections, which describe the interaction between the column and the rest of the structure, and a uniformly distributed lateral load along the whole length of the column and torque. General equilibrium conditions must be satisfied for these forces and moments. In particular, lateral forces Q_1 and Q_2, as well as nodal moments M_1 and M_2 have to be taken from the results of the calculation of the system as a whole. The equilibrium conditions are as follows:

$$Q_1 - Q_2 + qL = 0,$$

$$M_2 - M_1 - Q_1L - qL^2/2 = 0.$$

It should be noted that in this mode a positive longitudinal force corresponds to compression. Depending on whether you are dealing with a uniaxial or biaxial stress state, you can specify the whole set of forces or only the forces corresponding to the uniaxial eccentricity.

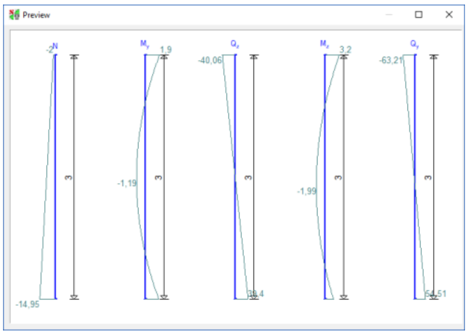


Figure 7.4.7-3. Diagrams of forces in a column

Follow these steps to specify forces for each load case:

- click the **Add** (a load case) button and its number will appear in the list of load cases;
- select a type of the considered load case in the list of loading types (permanent, temporary long-term, temporary short-term, wind, snow). The type of load case defines the combination factors according to SNiP 2.01.07-85*, which will be used with the loads of this load case when calculating the DCFs;
- enter the values of forces that correspond to the considered load case in the respective text fields;
- in the **Factor for sustained load** text field, enter a value of this factor for the considered load case;
- in the **Safety factor for load** text field specify a value of this factor;
- click the **Apply** button.

Once the latter operation is performed, the values of the nodal forces Q_1 and Q_2 which ensure the equilibrium will be calculated automatically.

The **Self-weight of column** checkbox enables to automatically add a load from the self-weight of the column to the current load case.

The button enables to view the diagrams of forces in the column for the current load case (Fig. 7.4.7-3).

To delete the current load case, click the **Delete** button.

To edit a previously created load case (the editing mode enables only to enter new loads and to delete the whole load case), select its number from the drop-down list of numbers of load cases. Once the new loads are added to the load case, click the **Apply** button.

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Column Segments

Segments of the column are described in the **Segments** tab according to the same rules as for the **Beam Check** mode (see Section 7.4.3). The only difference is that there is no list of spans, and the data are specified for all segments at once. Their number is selected from the **Number of segments** list. The numbering of segments in a column goes from the bottom to the top.

Design the Column

The application has a mode which enables to create automatically a set of working drawings for a column (the **Design the column** button). The drawings contain a specification, a steel consumption sheet, a specification of details, and a drawing of the cage with its specification (if the structure is reinforced by cages).

The following peculiarities and limitations should be taken into account when designing the column:

1. The design can be performed only for columns of a rectangular or ring section.
2. Longitudinal reinforcement in a column is always symmetric, i.e. $S_1=S_2$, and constant over its height. When entering the initial data, you have to specify only the diameter and the spacing of reinforcement S_1, and S_2 will be taken into account automatically.
3. Reinforcement S_1 and S_2 can be arranged only in one row (if the second row is specified, it will be ignored).
4. One row (S_1 and S_2) must not have less than two rebars.
5. A diameter and spacing must be specified for transverse reinforcement.

Note: Since the longitudinal reinforcement is constant along the whole length of the column, it is enough to specify it only for the first segment; and the diameter and spacing of the transverse reinforcement have to be specified for all the other segments.

Once the design mode is invoked, initial data is checked, and if there are no errors the **Design the column** dialog box appears where you have to use the radio buttons to select the reinforcement type: either with bars or with cages. Having made your choice, click the **OK** button, and the **Design the column** window (Fig. 7.4.7-4) will appear. It contains a toolbar with control buttons and a drawing field.

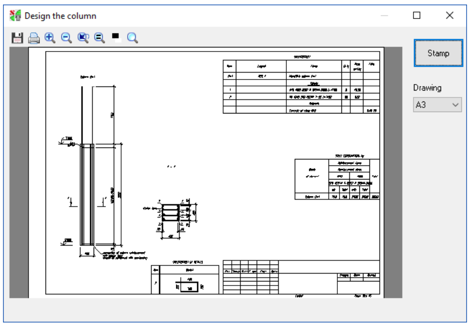


Figure 7.4.7-4. *The* **Design the column** *dialog box*

The toolbar contains buttons to perform the following actions:

— **Zoom Initial**, — **Zoom In**, and — **Zoom Out** — these three operations are used to change the scale of the image when viewing the documents. When the picture is zoomed in, scrollbars appear on the edges of the display field, which enable you to view the document. If your mouse has a wheel, it can be also used to *Zoom in* (roll forward) and *Zoom out* (roll backward) the image.

— **Undo View Change**— set the scale which was used before invoking any of the above commands;

— **Invert colors** — 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. 7.4.7-5);

— **Print** — prints the document (makes a hard copy). A standard dialog box, **Print**, appears where you have to select a device you want to use to print the document and change its properties if necessary;

— **Save** — if you want to improve 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 can specify a name and a directory to save the drawing, and its format (either DWG or DXF).

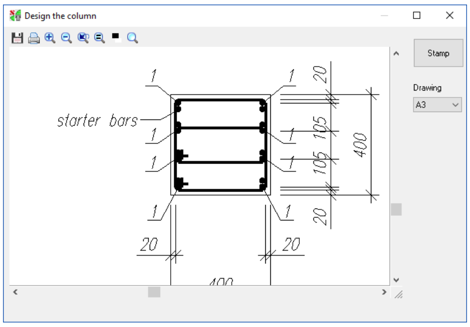


Figure 7.4.7-5. *The* **Design the column** *dialog box* — magnified view

The **Stamp** button opens the **Stamp** dialog box where you can fill in the fields of the stamp of form 3 according to GOST 21.1101-2013. (When the drawing of a cage is generated, you will be able to fill in the stamp of form 4).

Output documents can be generated in the A3 or A4 format. The cage drawing is always output in the A4 format. A format can be selected from the **Drawing** drop-down list, which includes the following items:

- *A3* — the drawing of the column with its sections, specification, steel consumption sheet, and specification of details is arranged on a single sheet of the A3 format;
- *Sheet 1 (A4)* — the first sheet of the A4 format contains the drawing of the column with its sections, specification of details, and the stamp;

- *Sheet 2 (A4)* — the second sheet of the A4 format contains the specification, the steel consumption sheet, and the stamp;
- *Cage (A4)* — if the structure is reinforced by cages, the sheet of the A4 format will contain a drawing of the cage with the specification and the stamp. If the structure is reinforced by individual bars, this option will not be available.

Crack Resistance

When SNiP 2.03.01-84*, KMK 2.03.01-96 is used, the **Analysis on crack resistance** checkbox in the **General** tab defines whether a check of the column reinforcement for crack resistance is needed. If the analysis is performed according to SP 52-101-03 and SP 63.13330, this checkbox is always checked.

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

Results

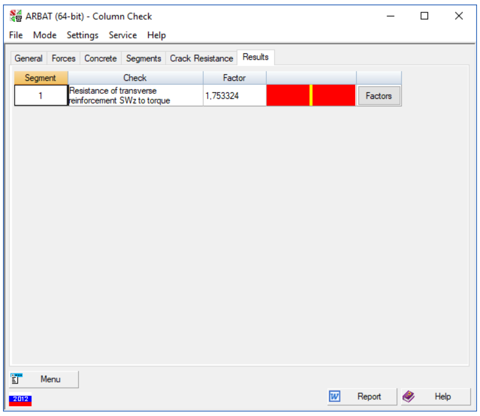


Figure 7.4.7-6. *The* **Results** *tab*

This tab (Fig. 7.4.7-6) opens automatically once the check mode is activated (the **Calculate** button).

Results of the analysis are displayed in a table. The first column of the table lists the numbers of segments. For each segment, the **Check** column provides the name of a factor for which the load-bearing capacity utilization factor achieves its maximum value, and the **Factor** column provides its value. The last column of the table contains a graphical representation of the factors where red bars denote factors greater than one.

A report can be generated on the basis of the results of the calculation (the **Report** button); its generation is described in Section 2.5.

Similarly to the **Beam Check** mode (see Section 7.4.3) the **Factors** buttons in the table enable to obtain the detailed information about all checks performed for each segment of the column.

7.4.8 Slab Check

This mode enables to perform a check of a given structure of a rectangular area of a solid slab. Depending on the ratio of the lengths of their sides slabs can be classified into those bent in one direction and those that can be bent in two directions. The area of a slab can be either a separate structural member of a building or a structure (e.g. a floor panel over a rectangular opening) or a part of a ribbed slab. The load-bearing capacity of the slab is determined from the limit equilibrium conditions according to a procedure given in the Guide to SNiP 2.08.01-85 [8] and the Instruction manual for the analysis of statically indeterminate reinforced concrete structures with the consideration of the redistribution of forces [6].

To perform the check of a slab, a limit value of the uniformly distributed load is compared with the total load from the specified load cases.

The application performs the check of:

- the load-bearing capacity of the slab for a bending moment caused by the total uniformly distributed load, including the case when the load-bearing capacity of the anchors is taken into account;
- the load-bearing capacity of the slab for a shear force caused by the total uniformly distributed load;
- the load-bearing capacity of the slab for the formation of cracks in the slab span and along the support lines;
- the maximum crack opening width in the span and in the support sections of the slab;
- the maximum deflection of the slab.

Peculiarities of the implementation

- For slabs bent in one direction the application implements the regulations of the Guide to SNiP 2.08.01-85 concerning the possibility of the increase of the limit bending moments in the span and on the supports by 20%;
- for slabs bent in two directions the application implements the regulation of the Instruction Manual [6] concerning the increase of the limit uniformly distributed load by 10%;
- only sections in the vicinity of supports are checked for the resistance to a shear force;
- the check of the formation of cracks and their opening width is performed only for sections normal to the longitudinal axis of the slab.

Boundary conditions of the slabs:

a) for slabs bent in one direction, the boundary conditions are specified only on two sides. At least one side of the slab should be clamped. The other side of the slab can be clamped, simply supported, or free from supports. This combination of the boundary conditions enables to model end and middle spans of continuous “beam-like” slabs. The second and the next spans of the continuous slab are not distinguished between one another. In all cases, the distribution of internal forces is assumed to be the same as that for the second span from the slab edge. Moreover, these boundary conditions enable to analyze the slab as a separate structural member in all practically important cases;

b) for slabs 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 in compliance with the regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SP 52-101-03, SP 63.13330).

Data are prepared in the following tabs: **General**, **Loads**, **Concrete**, and **Crack Resistance**.

General

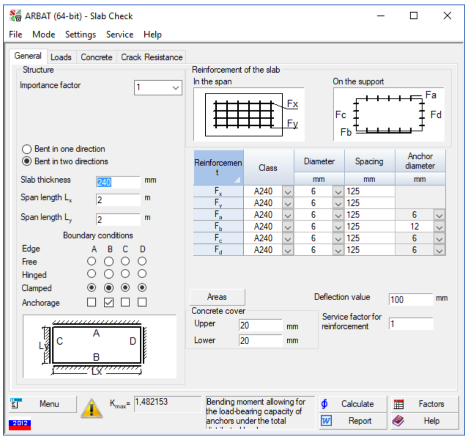


Figure 7.4.8-1. *The* **General** *tab*

This tab (Fig. 7.4.8-1) is used to enter the data which describe the structure of the slab, the diameter, class, and spacing of rebars, and the diameter of anchor bars (for slabs bent in two directions), the thickness of lower and upper concrete covers, service factor for reinforcement, and the allowable deflection value. Anchor bars can be installed only on the clamped sides of the slab. The reinforcement class for the anchor bars is assumed to be the same as that for the primary reinforcement. It is assumed that the anchors are made of single rebars installed in one row.

In order to select a structure of the slab, you have to use the respective radio button to indicate the type of its behavior (bent in one or two directions), specify the thickness 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 less than or equal to 3 (i.e., $\frac{L_x}{L_y} \le 3$ or $\frac{L_y}{L_x} \le 3$), the slab should be considered as bent in two directions. For a slab with one free edge, bent in two directions, the ratio of the length of the clamped edge to the length of the free edge must not exceed 1.5.

Boundary conditions for every face of the slab are specified by selecting the respective radio buttons: **Free** (edge), **Hinged** (edge), or **Clamped** (edge), which indicate the way the slab bears on its supporting structures or the way the slab area bears on the supporting ribs. For clamped faces, the **Anchorage** group of checkboxes is used to indicate the presence of anchors

Active columns and rows in the slab reinforcement table depend on the specified design.

The **Areas** button enables to obtain the reference information about the areas of reinforcement per running meter of the slab.

Loads

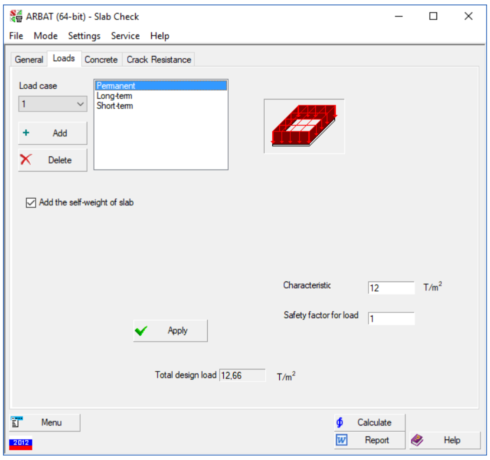


Figure 7.4.8-2. *The* **Loads** *tab*

The slab is checked for the action of only uniformly distributed loads over its whole area. The user has to reduce the loads to the uniformly distributed ones. This tab (Fig. 7.4.8-2) is used to specify the uniformly distributed loads. The slab is analyzed only for one combination of loads. The loads are specified in the same way as described in Section 7.4.7.

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

When specifying the loads their simplified classification is used (permanent, long-term, short-term). It is assumed that permanent and long-term loads do not have a short-term part, and the short-term load does not have the sustained one.

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Limitations

- the grade of concrete cannot be lower than B12,5;
- fine-grained concrete can be of either group A or B only;
- natural humidity is assumed for concrete. Water saturation, alternate water saturation and drying are not taken into account.

Crack Resistance

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

Results

The result of the check is a value of K_{max} (the maximal factor) which is displayed at the bottom of the dialog box. A complete list of all factors (checks) can be obtained by clicking the **Factors** button. This will open a dialog (see Section 1.3) with a list of all checks.

A report can be generated on the basis of the results of the calculation (the **Report** button).

7.4.9 Slab Check (Karpenko' theory)

This mode is used to check reinforced concrete elements of shells, slabs and deep beams using the criteria of strength and crack resistance in compliance with regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SNiP 52-01-2003, SP 52-101-2003, SP 63.13330.2012). Since the procedure for checking the reinforcement in two-dimensional elements is not given in the codes (except for SP 63.13330.2012), the method proposed by N.I. Karpenko [14] is used in **SCAD Office** (including **ARBAT**). The main guidelines of this method correspond to the general guidelines for the analysis of plane structures given in SNiP.

Initial data for the check of reinforcement are entered in the following tabs: **General**, **Stresses**, **Concrete**, and **Crack Resistance** (if the **Analysis on crack resistance** checkbox is checked), and results are analyzed in the **Results** tab.

General

The following data are specified in the **General** tab (Fig. 7.4.9-1):

- plate thickness;
- member type;
- importance factor;
- distance to the center of gravity of reinforcement;
- whether to perform the analysis on crack resistance;
- class and service factor for longitudinal reinforcement;
- class and service factor for transverse reinforcement;
- reinforcement parameters.

Member type — members of the shell, deep beam, or slab type are considered. Stresses N_x, N_y, T_{xy} for the members of slab type are taken as zero, and M_x, M_y, M_{xy} — for the members of deep beam type.

Importance factor — the value is selected from the drop-down list. Depending on the selected safety codes the importance factor will be applied either to design, or to design and characteristic values of forces/stresses.

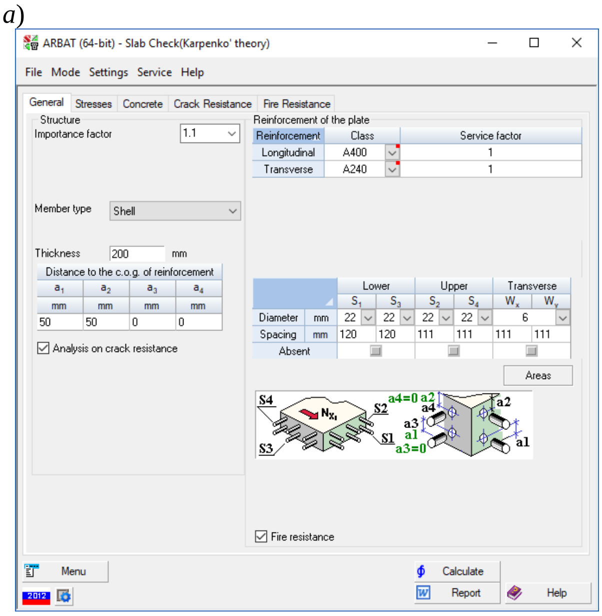
Distance to the c.o.g. of reinforcement — can be defined by two or four numbers. In the first case, the value $\mathbf{a_1}$ corresponds to the distance to the center of gravity of the lower reinforcing mesh, and $\mathbf{a_2}$ corresponds to the distance to the center of gravity of the upper reinforcing mesh. In the second case, $\mathbf{a_1}$ and $\mathbf{a_2}$ are specified for the reinforcement placed respectively below and above along the X_1 axis, and $\mathbf{a_3}$ and $\mathbf{a_4}$ are specified for the reinforcement along the Y_1 axis.

Analysis on crack resistance — if the respective checkbox is checked (it is checked by default), the analysis will be performed taking into account the crack resistance.

Plate reinforcement — class and service factor for longitudinal and transverse reinforcement are specified in this table.

Reinforcement — diameter and spacing of longitudinal and transverse reinforcement are specified in this table in accordance with the reinforcement layouts given in this tab.

a)



b)

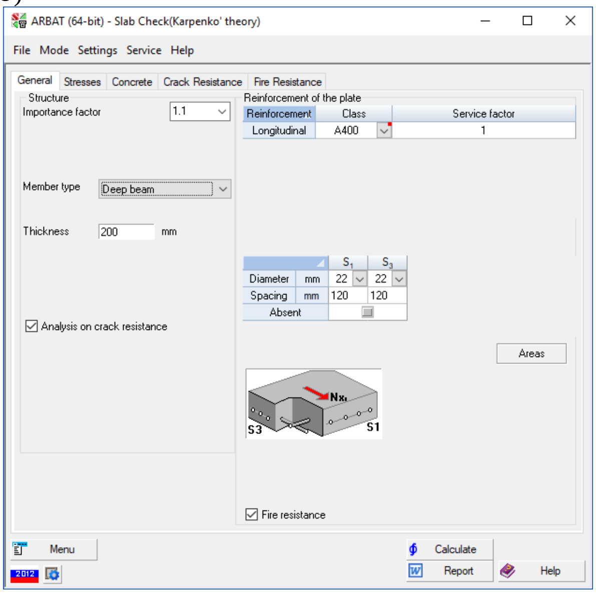


Figure 7.4.9-1. *The* **General** *tab of the* **Slab Check (Karpenko' theory)** *dialog box*
a) shell/ slab *b*) deep beam

Stresses

The **Stresses** tab (Fig. 7.4.9-2) enables to specify stresses in an element. Stresses can be specified by the user or loaded from a file with the **.rsu2** extension (button ) with the values of the design combinations of forces (DCF) calculated in SCAD. The file is generated in the **DCF** section of the **Information on the Element** mode by clicking the **Export** button.

Each set of force factors must contain design, characteristic, and characteristic long-term stress values. Moreover, depending on the actions included in the combination they can be specified as seismic, short-term or special using the respective checkboxes.

The **Add** and **Delete** buttons enable to add new rows to the table or to delete the existing ones, respectively.

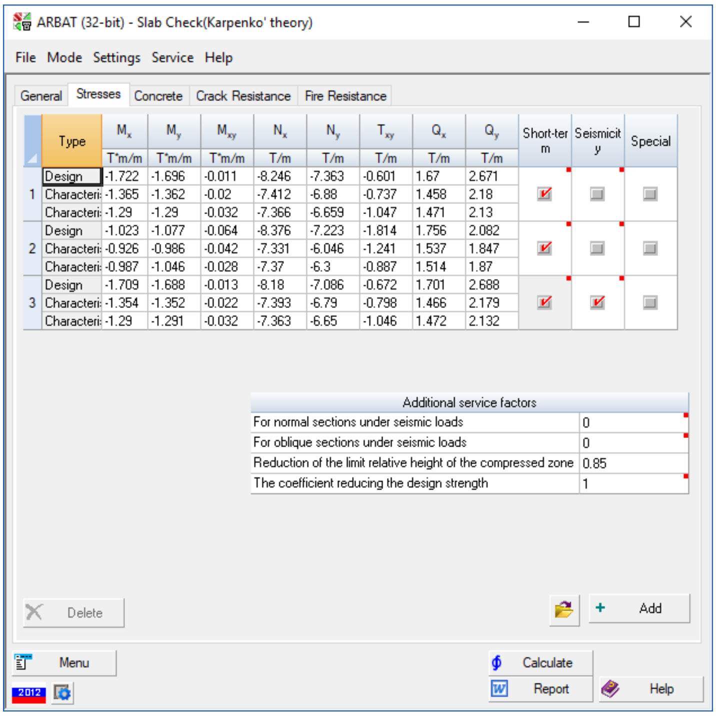


Figure 7.4.9-2. *The* **Stresses** *tab of the* **Slab Check (Karpenko' theory)** *dialog box*

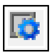
Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Crack Resistance

This tab appears when the **Analysis on crack resistance** checkbox is checked in the **General** tab. Data on crack resistance are specified according to the rules described in Section 7.3.4.

Additional Settings

Clicking the button  invokes the **Additional Settings** dialog box (Fig. 7.4.9-3), which enables to define the rules for applying some guidelines of the codes, in particular, limiting the maximum distance between cracks (the ***Limit the distance between cracks to min(40 cm, 40∅)*** checkbox) and the rules for considering shear forces (the ***Use formula (8.106) from SP 63.13330*** checkbox). These rules are considered in more detail in Section 7.3.7.

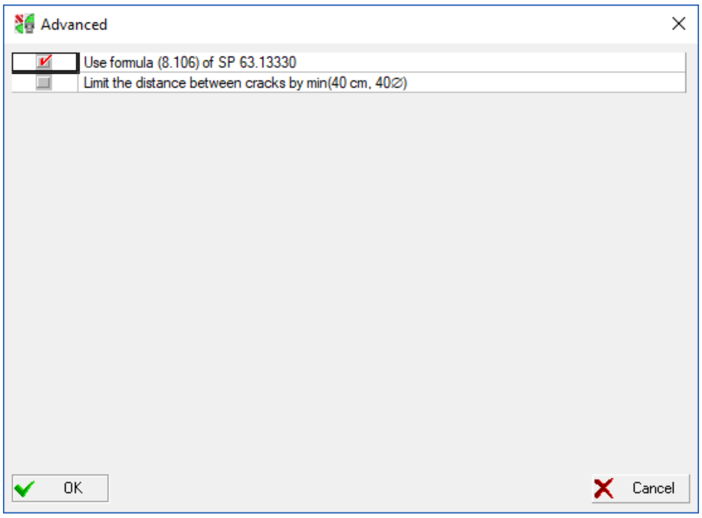


Figure 7.4.9-3. The **Additional Settings** dialog box

Display Results of the Check

The analysis is invoked by clicking the **Calculate** button. The results of the checks are displayed as a factors diagram after the clicking the **Factors** button. The following values are checked depending on the type of the member: strength of the plate section, strength for the shear forces Q_x, and Q_y , short-term and long-term crack opening width, etc.

7.4.10 Ultimate Slab Load

This mode enables to determine the ultimate uniformly distributed load on the slab.

According to the theory [1] it is assumed that at the limit equilibrium stage the slab is broken into disks connected along the fracture lines by linear plastic hinges. It is assumed that the ultimate distributed moments act in plastic hinges in the limit state. Compared to them, the effect of shear forces and torques is small and therefore neglected.

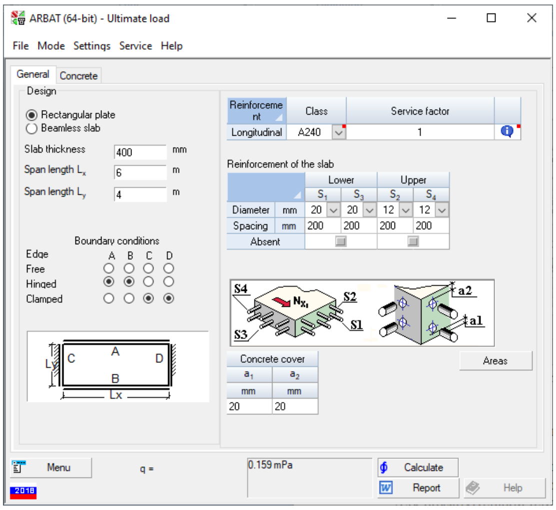
Plastic deformations along the fracture lines are accompanied by elastic bending of the slab elements. Due to the smallness of elastic deformations in comparison with plastic ones, they are neglected and the slab elements are considered as flat disks, and the fracture lines are considered as rectilinear hinges.

This mode enables to analyze rectangular cells of reinforced concrete slabs with different boundary conditions and flat slabs. The area of a slab can be either a separate structural member of a building or a structure (e.g. a floor panel over a rectangular opening) or a part of a ribbed slab. The result of the calculation is the value of the ultimate uniformly distributed load. Design formulas for its determination were obtained based on the data given in [2-5].

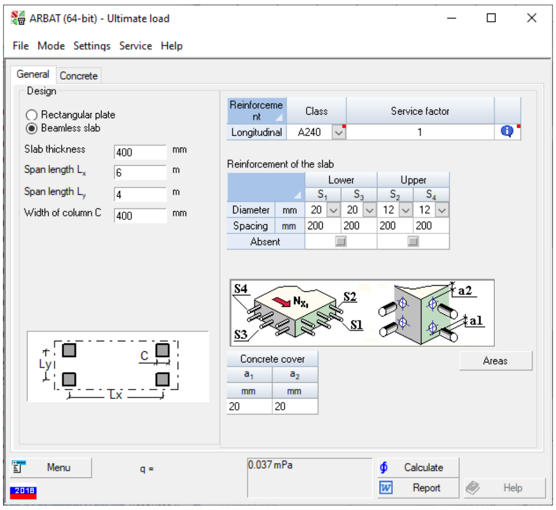
Data are prepared in the following tabs: **General** and **Concrete**.

General

This tab (Fig. 7.4.10-1) is used to enter the data which describe the structure of the slab, the diameter, class, and spacing of rebars, the thickness of lower and upper concrete covers, service factor for reinforcement.



Rectangular plate



Flat slab

Fig. 7.4.10-1. The **General** tab

A type of slab can be selected using the respective radio buttons:

- rectangular plate;
- flat slab.

If a rectangular plate is selected, you have to specify its sizes L_x,L_y and thickness. And in the case of a flat slab you have to additionally specify the column width.

Boundary conditions for every face of the slab are specified by selecting the respective radio buttons: **Free** (edge), **Hinged** (edge), or **Clamped** (edge), which indicate the way the slab bears on its supporting structures or the way the slab area bears on the supporting ribs.

Active columns and rows in the slab reinforcement table depend on the specified design.

The **Areas** button enables to obtain the reference information about the areas of reinforcement per running meter of the slab.

Limitations

Limitations of the boundary conditions:

- four free edges are not allowed;
- if three edges are free, the fourth one must be clamped.

Reinforcement limitation – if the slab is clamped, upper reinforcement is required.

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Results

The result of the calculation is the value of the ultimate uniformly distributed load, which is displayed at the bottom of the dialog box.

A report can be generated on the basis of the results of the calculation (the **Report** button).

7.5 Local Strength

All modes of this group perform a check of reinforced concrete structural members, including inserts, for the local action of loads in compliance with the regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 and taking into account requirements and recommendations of the Guide to SNiP 2.03.01-84* (the Guide to SP 52-101-03), Manual on designing of concrete and reinforced concrete structures made of heavy-weight concrete [11], Manual on designing of concrete and reinforced concrete structures made of heavy-weight concrete [12], Recommendations on designing of steel inserts for reinforced concrete structures [9].

The design is assumed to be known, i.e. we know the area of load application, junction between members etc., and we also know what additional transverse reinforcement is needed to ensure the local strength.

It is assumed in all modes of the local strength analysis, that there are long-term loads and the values of the design stength of concrete are multiplied by the values of the factor γ_{b2} (for SNiP calculations) or γ_{b1} (for SP calculations) specified by the user. In order to perform the analysis of the local strength at the short-term loads, it is necessary to enter the respective value of the factor γ_{b2} (γ_{b1}) when specifying the data on concrete.

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

7.5.1 Local Compression (SNiP 2.03.01-84*)

This mode implements checks of reinforced concrete structural members for local compression in compliance with the regulations of Sec. 3.39 - 3.41 of SNiP 2.03.01-84*, KMK 2.03.01-96. All structural schemes of reinforced concrete members given in Drawing 15 of SNiP 2.03.01-84* are implemented.

The checks are performed for a longitudinal force applied to the members both with the additional lateral reinforcement by transverse meshes and without the lateral reinforcement. Depending on the presence of the lateral reinforcement, one of the following factors is calculated:

- strength from conditions of local compression for a non-reinforced section;
- strength from conditions of local compression for a section reinforced by meshes.

General

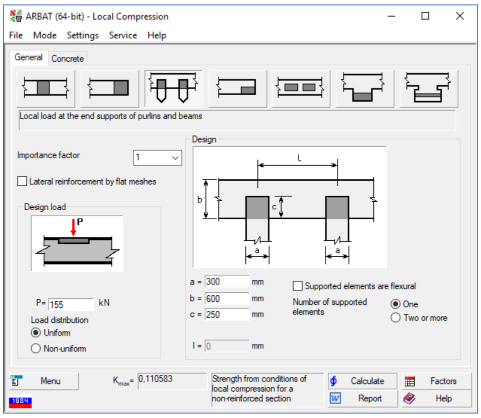


Figure 7.5.1-1. *The* **General** *tab* (SNiP 2.03.01-84*)

This tab (Fig. 7.5.1-1) is used to specify:

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

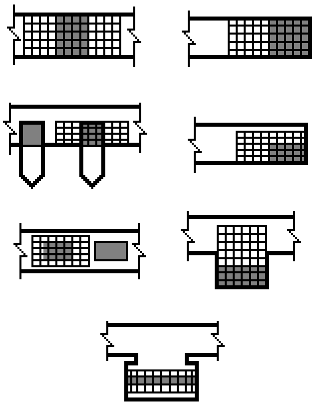


Figure 7.5.1-2. *Load arrangements (according to* SNiP 2.03.01-84*)

The load arrangement (Fig. 7.5.1-2) is selected by clicking on the respective icon and corresponds to one of the schemes given in Drawing 15 of SNiP 2.03.01-84*. Modifications of the load arrangements assumed according to the Drawing 15 are described in the **Peculiarities of the implementation** section.

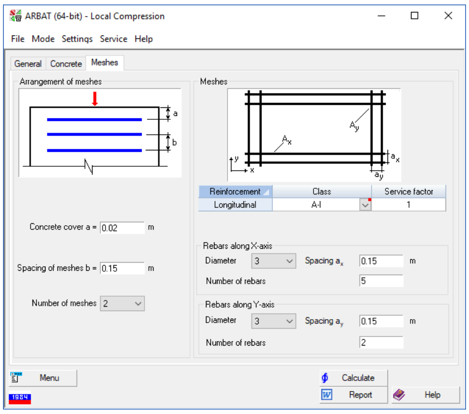


Figure 7.5.1-3. *The* **Meshes** *tab*

When the **Lateral reinforcement by flat meshes** checkbox is checked, the **Meshes** tab becomes accessible for entering data (Fig. 7.5.1-3).

This tab is used to specify:

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

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Peculiarities of the implementation

Load arrangements (*c*) and (*d*) of Drawing 15 of SNiP 2.03.01-84* are combined in one image.

Load arrangement (*f*) of Drawing 15 of SNiP 2.03.01-84* has been modified. The application enables to specify multiple application points of equal loads (see Fig. 57) similarly to schemes (*c*) and (*d*). In this case the rules for determining the local area of bearing are assumed to be the same as those for schemes (*c*) and (*d*).

The vertical spacing of the meshes and the number of the meshes are checked using formulas (198) and (199) from Section 3.94 of the Guide to SNiP 2.03.01-84*.

Limitations

The limitations deal with the following circumstances:

- whether it is legitimate to perform the check for local compression (it depends on the ratio of the lateral sizes of the checked member and the sizes of the load application area);
- the assumed arrangement of lateral reinforcement by flat meshes;
- structural limitations of the used classes and diameters of rebars, the spacing between rebars in meshes and between the meshes, and other limitations given in documents [7], [11], [12].

Limitations related to the legitimacy of a check are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of meshes, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

- the number of lateral reinforcing meshes is from 2 to 4;
- the thickness of the concrete cover is from 10 to 20 mm;
- the vertical distance between the meshes (the spacing of meshes) is from 60 to 150 mm;
- the reinforcement classes are Bp-I, A-I, A-II, A-III according to SNiP 2.03.01-84* and A400C according to TSN 102-00**;**
- the diameters of rebars depending on their class are from 3 to 14 mm;
- the distance between the bars of the meshes (the spacing of bars) in every direction is from 50 to 100 mm .

7.5.2 Local Compression (SP 52-101-2003, SP 63.13330)

This mode implements checks of reinforced concrete structural members for local compression in compliance with the regulations of Sec. 6.2.43–6.2.45 of SP 52-101-2003, Sec. 8.1.43-8.1.45 of SP 63.13330. All structural schemes of reinforced concrete members given in Fig. 6.11 of SP 52-101-2003 and Fig. 8.9 of SP 63.13330 are implemented.

General

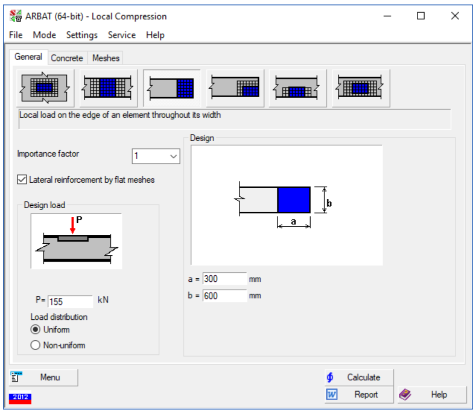


Figure 7.5.2-1. *The* **General** *tab*

This tab (Fig. 7.5.2-1) is used to specify:

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

The checks are performed for a longitudinal force applied to the members both with the additional lateral reinforcement by transverse meshes and without the lateral reinforcement. Depending on the presence of the lateral reinforcement, one of the following factors is calculated:

- strength from conditions of local compression for a non-reinforced section;
- strength from conditions of local compression for a section reinforced by meshes.

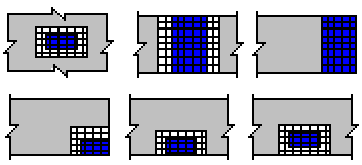


Figure 7.5.2-2. Load arrangements according to SP 52-101-03, SP 63.13330

The load arrangement (Fig. 7.5.2-2) is selected by clicking on the respective icon and corresponds to one of the schemes given in Fig. 6.11 of SP 52-101-03 and Fig. 8.9 of SP 63.13330.

When the **Lateral reinforcement by flat meshes** checkbox is checked, the **Meshes** tab becomes accessible for entering data (Fig. 7.5.2-1). Its structure is similar to that of the respective tab in the analysis according to SNiP 2.03.01-84* (Fig. 7.5.1-3).

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Peculiarities of the implementation

The vertical spacing of the meshes and the number of the meshes are checked using formulas (198) and (199) from Section 3.94 of the Guide to SNiP 2.03.01-84*.

Limitations

The limitations deal with the following circumstances :

- whether it is legitimate to perform the check for local compression (it depends on the ratio of the lateral sizes of the checked member and the sizes of the load application area);
- the assumed arrangement of lateral reinforcement by flat meshes;
- structural limitations of the used classes and diameters of rebars, the spacing between rebars in meshes and between the meshes, and other limitations given in documents [7], [11], [12].

Limitations related to the legitimacy of a check are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of meshes, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application :

- the number of lateral reinforcing meshes is from 2 to 4;
- the thickness of the concrete cover is from 10 to 20 mm;
- the vertical distance between the meshes (the spacing of meshes) is from 60 to 150 mm;
- the reinforcement classes are A240, A300, A400, A500, B500, A600C;
- the diameters of rebars depending on their class are from 10 to 14 mm;
- the distance between the bars of the meshes (the spacing of bars) in every direction is from 50 to 100 mm.

7.5.3 Punching (SNiP 2.03.01-84*)

This mode implements a check of slab structures (without transverse reinforcement) for punching in compliance with the regulations of Sec. 3.42 of SNiP 2.03.01-84*, KMK 2.03.01-96 and Sec. 3.98 of the Manual on designing of concrete and reinforced concrete structures made of heavy-weight concrete [12]. Both structures with the additional lateral reinforcement by vertical bars located within the bearing pyramid and structures without any lateral reinforcement can be checked here.

Depending on the presence of lateral reinforcement, one of the following factors is calculated:

- strength from conditions of punching without additional reinforcement ;
- strength from conditions of punching with additional reinforcement.

General

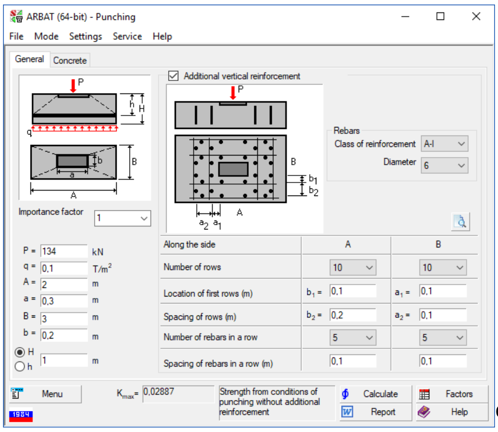


Figure 7.5.3-1. *The* **General** *tab*

The **General** tab (Fig. 7.5.3-1) is used to specify :

- the sizes of the slab base;
- the sizes of the load application area;
- the height of the slab (*H*), or the effective height of the section (*h*) where:
- *h* is the distance from the slab top to the center of gravity of the lower reinforcement (it can be selected using the respective radio button). If the slab height parameter *H* is used, the value of *h* is taken as 0.9*H*;
- the resultant of the punching load applied to the center of the load application area;
- a uniform load (including the zero load) applied to the base of the slab and resisting punching.

When the **Additional vertical reinforcement** checkbox is checked, the following data have to be specified:

- the class and the diameter of vertical rebars that make up the meshes parallel to the sides of the slab;
- the number of rows of vertical bars parallel to the sides of the slab;
- the location 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 the arrangement of rebars according to the parameters of the specified additional vertical reinforcement.

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Peculiarities of the implementation

The requirement of Sec. 3.42 of SNiP 2.03.01-84*, KMK 2.03.01-96 that the punching force should act on a “limited area” is implemented in the following way: the area is considered to be “limited” if the inclination of the side faces of the bearing pyramid to the horizon does not exceed 68°. This limitation complies with the fifth paragraph of Sec. 3.42 of SNiP 2.03.01-84*.

The punching is analyzed only for slab structures. Steps in the slabs (such as those of the foundation) are not taken into consideration.

The vertical punching load and the load applied to the base of the slab are assumed to be uniformly distributed over their application areas. Eccentricities of the application of loads are not taken into account.

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 meshes with the effective vertical rebars. The meshes are arranged parallel to the sides of the slab, symmetrically, and do not fall within the load application area. The location of meshes nearest to the load application area is such that they always fall within the bearing prism.

The requirement of Sec. 5.29 of SNiP 2.03.01-84*, KMK 2.03.01-96 concerning the installation of additional reinforcement at the punching analysis is implemented.

When there is no additional reinforcement the application automatically applies the coefficient of 0,9 to the value R_{bt} on the basis of the requirements of item 9 of table 15 in the case when only a concrete section is considered.

Limitations

The limitations deal with the following circumstances:

- whether it is legitimate to perform the check for punching (it depends on the ratio of 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;
- structural limitations of the used classes and diameters of rebars, the spacing between rebars in meshes and between the meshes, and other limitations given in documents [7], [11], [12].

Limitations related to the legitimacy of a check and the ratios of loads are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of rebar rows, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

- the reinforcement classes are Bp-I, A-I, A-II, A-III according to SNiP 2.03.01-84* and A400C**;**
- the diameters of rebars depending on their class are from 3 to 14 mm;
- the minimum spacing between the rows of rebars is defined by Sec. 5.29 of SNiP 2.03.01-84*;
- the distance from the first row of rebars to the load application area must be not less than 30 mm and not greater than the spacing between the rows;
- the total number of rebar rows along each side is from 2 to 10;
- the number of rebars in each row must be the same for the rows along each side and must not be less than 2 and not greater than 10.

The check determines the number of rows and the number of rebars in the rows which fall within the bearing prism. If some of the rebar rows or rebars in the rows do not fall within the bearing prism, the respective messages are displayed on the screen and included in the report.

7.5.4 Punching (SP 52-101-2003, SP 63.13330)

This mode implements a check of slab structures for punching in compliance with the regulations of Sec. 6.2.49-6.2.52 of SP 52-101-2003, Sec. 8.1.46-8.1.51 of SP 63.13330.2012. Both structures without the additional vertical reinforcement and those with the additional lateral reinforcement by vertical bars arranged in the cross-section of a slab fragment uniformly or in a cross can be checked here. In the general case, the analysis is performed for a combined action of a compressive force and a bending moment.

All checks defined by Sec. 6.2.49-6.2.52 of SP 52-101-03 and Sec. 8.1.46-8.1.51 of SP 63.13330 are implemented.

General

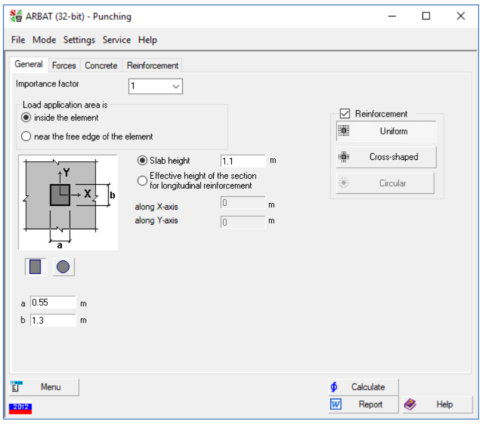


Figure 7.5.4-1. *The* **General** *tab*

This tab (Fig. 7.5.4-1) is used to specify:

- the sizes of the slab in plan;
- the shape of the load application area (rectangular or round (when SP 63.13330 is used));
- the sizes of the load application area and, if necessary, its snap to the slab edges;
- the height of the slab (*H*), or the effective heights of the sections for longitudinal reinforcement along the X and Y axes (they can be selected using the respective radio buttons). If the slab height *H* is used as the initial data, the values of the effective heights are approximately equal to 0.9*H*;
- the location of the load application area — near the free edge of the slab (Fig. 7.5.4-2) or in its middle part (Fig. 7.5.4-3).

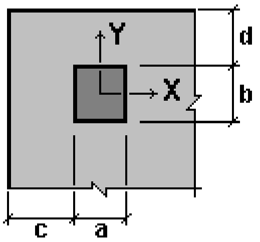


Figure 7.5.4-2. *Load application area near the free edge of the slab*

If the **Reinforcement** checkbox is checked, you have to specify the arrangement of reinforcement — uniform (Fig. 7.5.4-5, 7.5.4-6), cross-shaped (Fig. 7.5.4-7, 7.5.4-8), or circular (in the case of a round load application area) (Fig. 7.5.4-9, 7.5.4-10).

The rebars specified by the user are placed symmetrically with the given spacing from the center of gravity of the load application area (along the X and Y axes). This can be seen by clicking the **Preview** button , in the **Reinforcement** tab.

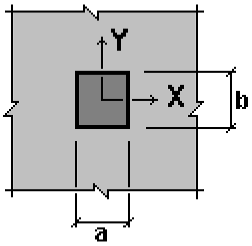


Figure 7.5.4-3. *Load application area in the middle part of the slab*

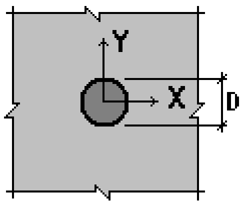


Figure 7.5.4-4. *Round load application area*

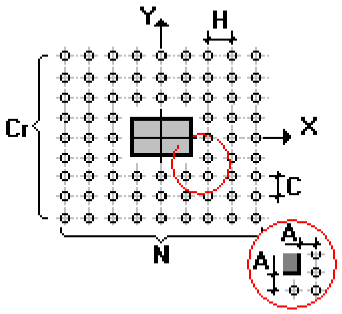


Figure 7.5.4-5. *Uniform arrangement of reinforcement*

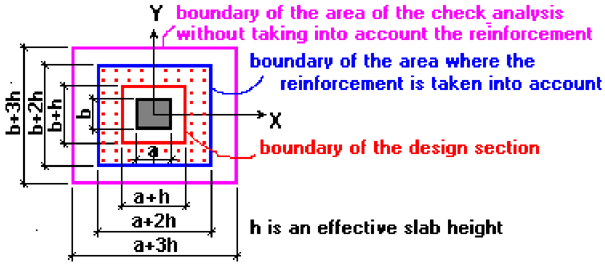


Figure 7.5.4-6. *Areas of the slab for the checking analysis, uniform arrangement of reinforcement*

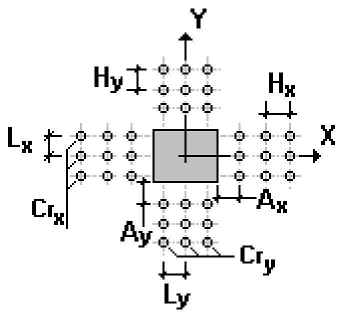


Figure 7.5.4-7. *Cross-shaped arrangement of reinforcement*

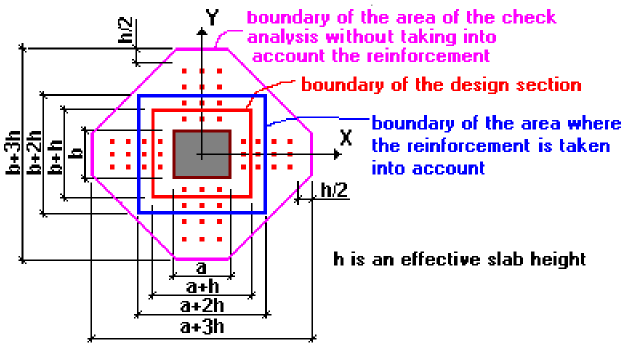


Figure 7.5.4-8. *Areas of the slab for the checking analysis, cross-shaped arrangement of reinforcement*

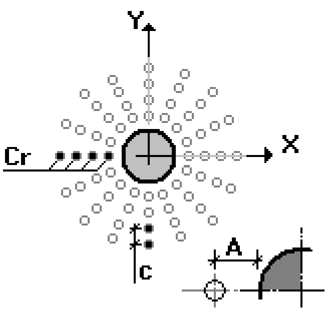


Figure 7.5.4-9. *Circular arrangement of reinforcement*

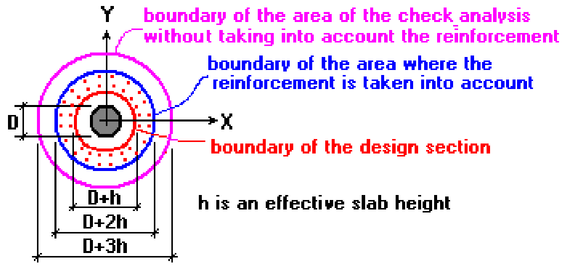


Figure 7.5.4-10 *Areas of the slab for the checking analysis circular arrangement of reinforcement*

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Forces

The resultants of the punching load (the longitudinal force and the bending moments) applied to the center of the load application area are specified in the **Forces** tab. This tab is similar to the *Forces* tab of the mode *Strength of RC Sections*.

Reinforcement

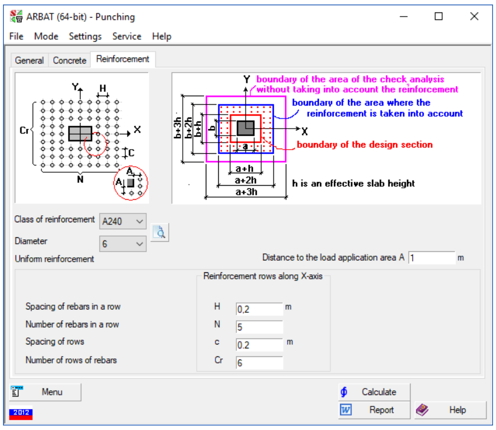


Figure 7.5.4-11. *The* **Reinforcement** *tab*

The **Reinforcement** tab (Fig. 7.5.4-11) is used to specify:

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

Peculiarities of the implementation

The punching analysis is implemented only for slab structures.

The distance to the free edges of a slab (for various schemes of punching) is taken into account only when the distance from the boundary of the load application area to the free edge does not exceed 3 effective heights (h_0) of longitudinal reinforcement of the slab along one of the axes, X, Y (free edge of the slab), or both axes X, Y (free corner of the slab). The punching analysis cannot be performed when the free edge of the slab falls within an area located at the distance of less than half the effective height of the slab to the load application area.

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

Transverse reinforcement is taken into account in the analysis within the full bearing pyramid with an offset of h_0 from the load application area to each side.

If the distance from the transverse rebars to the load application area is large enough, their presence has little effect on the ultimate force resisted by the transverse reinforcement at punching.

Since the application does not limit the number of the specified reinforcing bars in a row, the following limitation has been introduced: to accept the boundary of the area of the check analysis without taking into account the reinforcement at the distance of 0,5 h_0 not from the boundary of the location of the transverse reinforcement specified by the user (next-to-last paragraph of Sec. 6.2.48 of SP 52-101-2003 or Sec. 8.1.48 of SP 63.13330), but from the boundary of the area within which the analysis takes into account the transverse reinforcement.

Limitations

Limitations dealing with the design codes:

- the punching analysis cannot be performed if the ratio of the bending moments used in the analysis to the ultimate bending moments exceeds the ratio of the forces used in the analysis to the ultimate force resisted by concrete (see Sec. 6.2.46 of SP 52-101-2003, Sec. 8.1.46 of SP 63.13330)[4];
- lateral vertical reinforcement is implemented only in the case when the load application area is located in the middle part 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 always taken into account;
- in the implementation of the last paragraph of Sec. 6.2.52 of SP 52-101-03, Sec. 8.1.52 of SP 63.13330 the resisting moments of transverse reinforcement are assumed to be equal to those of concrete;
- structural limitations of the classes and diameters of rebars, of the spacing between rebars in meshes and between meshes, and other limitations given in documents [7], [11], [12].

Additional lateral reinforcement is implemented only as vertical meshes with effective vertical rebars. The meshes are parallel to the sides of the slab and do not fall within the load application area. Meshes nearest to the load application area are arranged in such a way that they always fall within the bearing prism.

Limitations related to the legitimacy of a check and the ratios of loads are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of rebar rows, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

* the reinforcement classes are A240, A300, A400, A500, B500, A600C according to SP 52-101-2003 (SP 63.13330);
* the diameters of rebars depending on their class are from 6 to 40 mm;
- the rebars are arranged in compliance with Sec. 8.3.15 of SP 52-101-03, Sec. 10.3.17 of SP 63.13330.

7.5.5 Punching Analysis of Wall Ends and Corners (SP 63.13330)

This mode implements a check of slab structures for punching in compliance with the regulations of Sec. 8.1.46-8.1.51 of SP 63.13330. Both structures without the additional vertical reinforcement and those with the additional lateral reinforcement by vertical bars uniformly arranged in the cross-section of a slab fragment can be checked here. In the general case, the analysis is performed for a combined action of a compressive force and a bending moment.

[4] This limitation is introduced to avoid the following absurd situation, which occurs if you follow the regulations of Sec. 8.1.46 of SP 63.13330.2012 literally, i.e., at a small ratio of values F ans F_{ult} arbitrarily large value of the bending moment happens to be allowable.

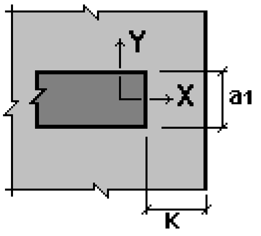
All checks defined by Sec. 8.1.46-8.1.51 of SP 63.13330 are implemented.

General

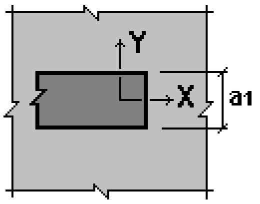
This tab is used to specify:

- the sizes of the slab in plan;
- the shape of the load application area (rectangular or L-shaped) (SP 430.1325800);
- the sizes of the load application area and, if necessary, its snap to the slab edges;
- the height of the slab (*H*), or the effective heights of the sections for longitudinal reinforcement along the X and Y axes (they can be selected using the respective radio buttons). If the slab height *H* is used as the initial data, the values of the effective heights are approximately equal to *0.9H*;
- the location of the load application area — near the free edge of the slab or in its middle part.

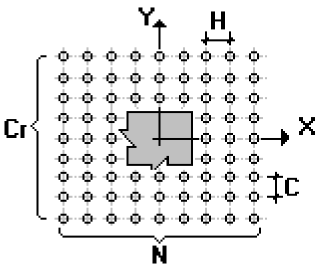
If the **Reinforcement** checkbox is checked, you have to specify the arrangement of reinforcement.



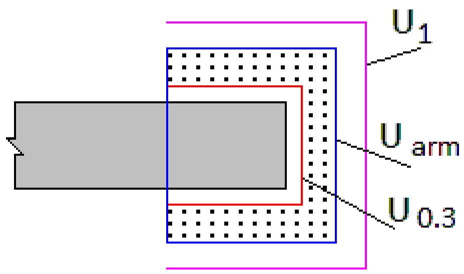
Load application area near the free edge of the slab



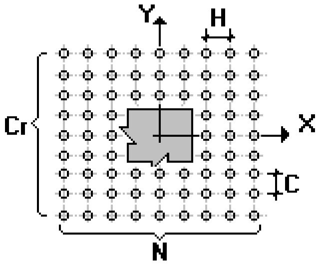
Load application area in the middle part of the slab



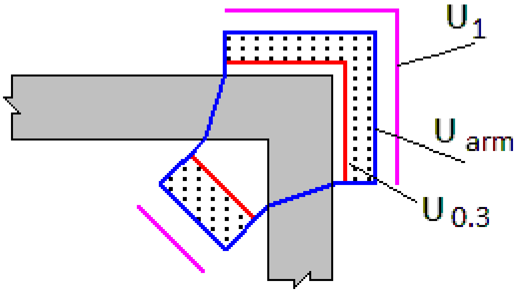
Uniform arrangement of reinforcement



Areas of the slab for the checking analysis, uniform arrangement of reinforcement near the wall end



Uniform arrangement of reinforcement



Areas of the slab for the checking analysis, uniform arrangement of reinforcement near the wall corner

The rebars specified by the user are placed symmetrically with the given spacing from the center of gravity of the load application area (along the X and Y axes). This can be seen by clicking the **Preview** button  in the **Reinforcement** tab.

Concrete

Properties of concrete are specified in the standard **Concrete** tab.

Forces

This tab is used to specify the resultants of the punching load (the longitudinal force and the bending moments) applied to the center of the load application area.

The punching analysis of the wall end is performed for the longitudinal force P and the out-of-plane bending moment Mx.

The punching analysis of the wall corner is performed only for the longitudinal force P without taking into account the moment Mx.

The forces are specified in the same way as in the **Strength of RC Sections** mode.

Reinforcement

The **Transverse Reinforcement** tab is used to specify:

- the class and the diameter of transverse rebars in a slab;
- the distance between the rebars in a row;
- the number of rebars in a row;
- the spacing between the rebar rows;
- the number of rebar rows.

Peculiarities of the implementation

The punching analysis is implemented only for slab structures.

The distance to the free edges of a slab (for various schemes of punching) is taken into account only when the distance from the boundary of the load application area to the free edge does not exceed 3 effective heights (h_0) of longitudinal reinforcement of the slab along one of the axes, X, Y (free edge of the slab), or both axes X, Y (free corner of the slab). The punching analysis cannot be performed when the free edge of the slab falls within an area located at the distance of less than half the effective height of the slab to the load application area.

Lateral reinforcement is parallel to the X and Y axes.

Transverse reinforcement is taken into account in the analysis within the full bearing pyramid with an offset of h_0 from the load application area to each side.

The given transverse reinforcement is arranged in such a way so that its center coincides with the center of mass of the punching contour located at a distance of 0,5 h_0.

If the distance from the transverse rebars to the load application area is large enough, their presence has little effect on the ultimate force resisted by the transverse reinforcement at punching.

Since the application does not limit the number of the specified reinforcing bars in a row, the following limitation has been introduced: to accept the boundary of the area of the check analysis without taking into account the reinforcement at the distance of 0,5 h_0 not from the boundary of the location of the transverse reinforcement specified by the user (Sec. 8.1.48 of SP 63.13330.2012), but from the boundary of the area within which the analysis takes into account the transverse reinforcement.

Limitations

Limitations dealing with the design codes:

- the punching analysis cannot be performed if the ratio of the bending moments used in the analysis to the ultimate bending moments exceeds the ratio of the forces used in the analysis to the ultimate force resisted by concrete (see Sec. 8.1.46 of SP 63.13330);
- lateral vertical reinforcement is implemented only in the case when the load application area is located in the middle part of the slab;
- in the implementation of the last paragraph of Sec. 8.1.52 of SP 63.13330.2012 the resisting moments of transverse reinforcement are assumed to be equal to those of concrete;
- structural limitations of the classes and diameters of rebars, of the spacing between rebars in meshes and between meshes, and other limitations.

Additional lateral reinforcement is implemented only as vertical meshes with effective vertical rebars. The meshes are parallel to the sides of the slab and do not fall within the load application area. Meshes nearest to the load application area are arranged in such a way that they always fall within the bearing prism.

Limitations related to the legitimacy of a check and the ratios of loads are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of rebar rows, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

- the reinforcement classes are A240, A300, A400, A500, B500, A600C according to SP 63.13330;
- the diameters of rebars depending on their class are from 6 to 40 mm;
- the rebars are arranged in compliance with Sec. 10.3.17 of SP 63.13330.

7.5.6 Punching (EN 1992-1-1)

This mode implements a check of slab structures for punching in compliance with the regulations of Sec. 6.4 of EN 1992-1-1. Both structures without the additional vertical reinforcement and those with the additional lateral reinforcement by vertical bars arranged in the cross-section of a slab fragment uniformly, in circles or in a cross can be checked here. In the general case, the analysis is performed for a combined action of a vertical shear force and longitudinal forces.

All checks defined by Sec. 6.4.3-6.4.5 of EN 1992-1-1 are implemented.

General

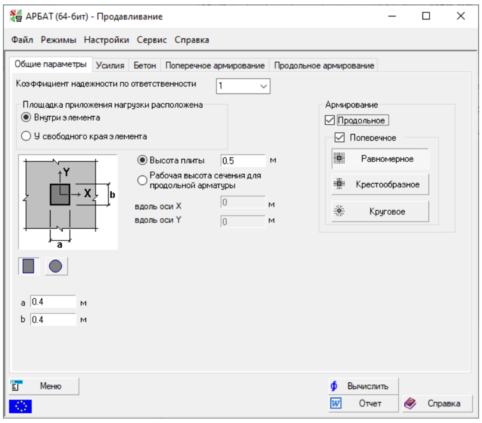
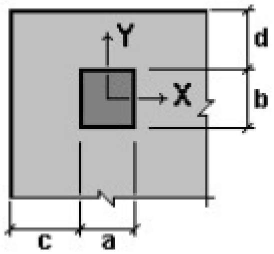


Figure 7.5.6-1. *The* **General** *tab*

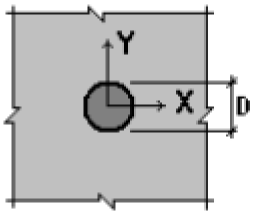
The **General** tab (Fig. 7.5.6-1) is used to specify:

- the sizes of the slab in plan;
- the shape of the load application area (rectangular or round);
- the sizes of the load application area and, if necessary, its snap to the slab edges;
- the height of the slab (*H*), or the effective heights of the sections for longitudinal reinforcement along the X and Y axes (they can be selected using the respective radio buttons). If the slab height *H* is used as the initial data, the values of the effective heights are approximately equal to 0,9*H*;
- the location of the load application area — near the free edge of the slab or in its middle part.

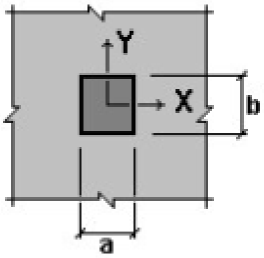
If the **Reinforcement** checkbox is checked, you have to specify the arrangement of reinforcement — uniform, cross-shaped, or circular (in the case of a round load application area).



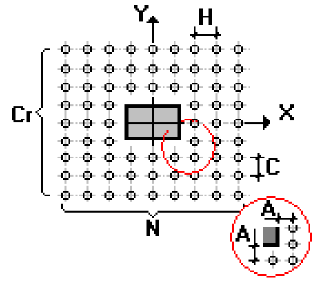
Load application area near the free edge of the slab



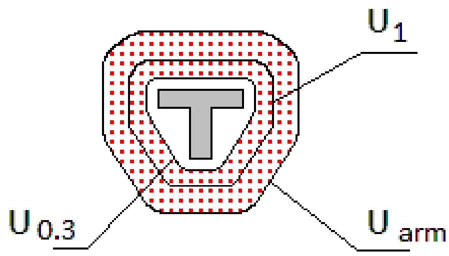
Round load application area



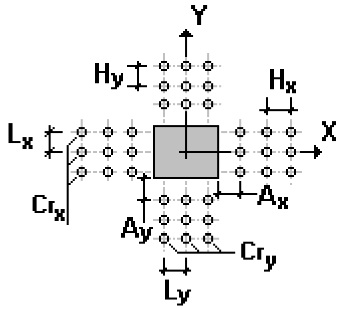
Load application area in the middle part of the slab



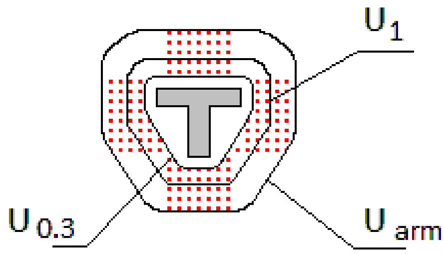
Uniform arrangement of reinforcement



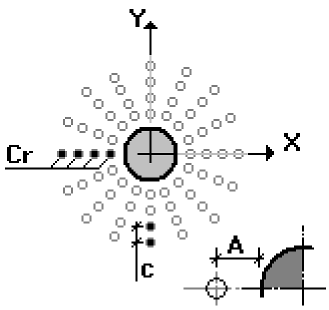
Areas of the slab for the checking analysis, uniform arrangement of reinforcement



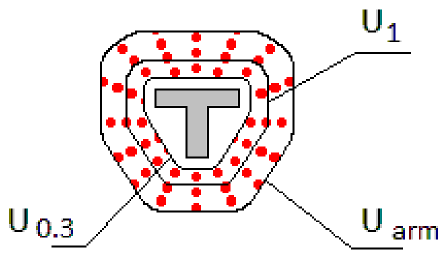
Cross-shaped arrangement of reinforcement



Areas of the slab for the checking analysis, cross-shaped arrangement of reinforcement



Circular arrangement of reinforcement



Areas of the slab for the checking analysis circular arrangement of reinforcement

The rebars specified by the user are placed symmetrically with the given spacing from the center of gravity of the load application area (along the X and Y axes). This can be seen by clicking the **Preview** button in the **Reinforcement** tab.

Concrete

Properties of concrete are specified in the standard **Concrete** tab.

Forces

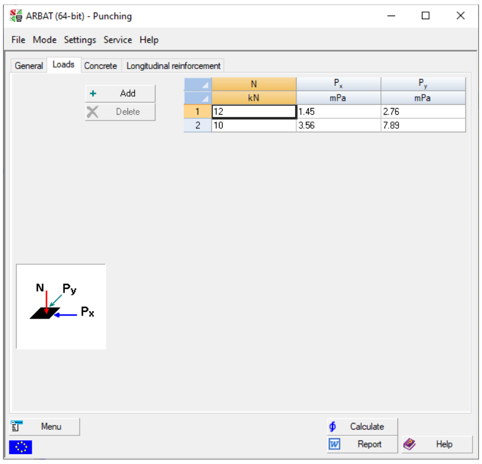


Figure 7.5.6-2. *The* **Forces** *tab*

The **Loads** tab (Fig. 7.5.6-2) is used to specify punching load and stresses in the concrete in the X and Y directions. This tab is similar to that of the **Strength of RC Sections** mode.

Reinforcement

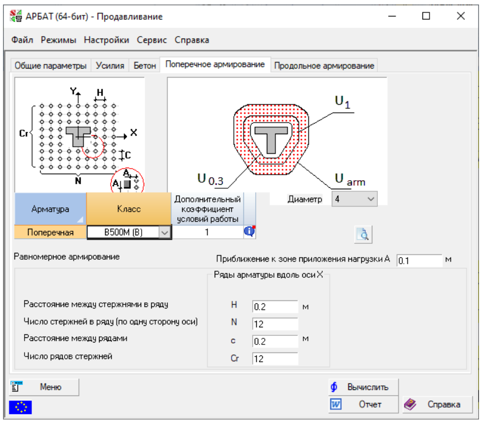


Figure 7.5.6-3. *The* **Transverse Reinforcement** *tab*

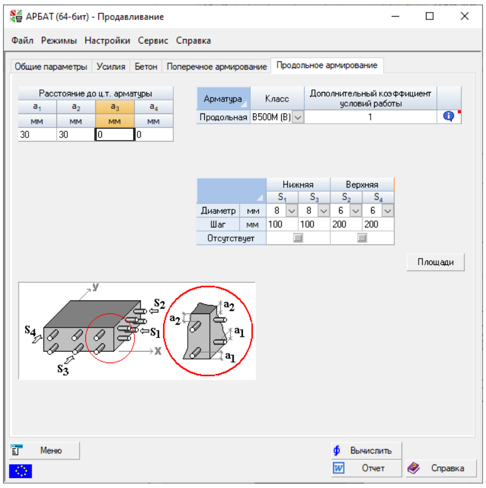


Figure 7.5.6-4. *The* **Longitudinal Reinforcement** *tab*

Information on longitudinal and transverse reinforcement in the loaded area is required to perform the punching analysis.

We need the information on longitudinal reinforcement in the slab to determine the reduced reinforcement ratio for longitudinal reinforcement along the X and Y axes according to Sec. 6.4.4.

We need the information on transverse reinforcement to determine the punching shear resistance according to Sec. 6.4.5.

The **Transverse Reinforcement** tab (Fig. 7.5.6-3) is used to specify:

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

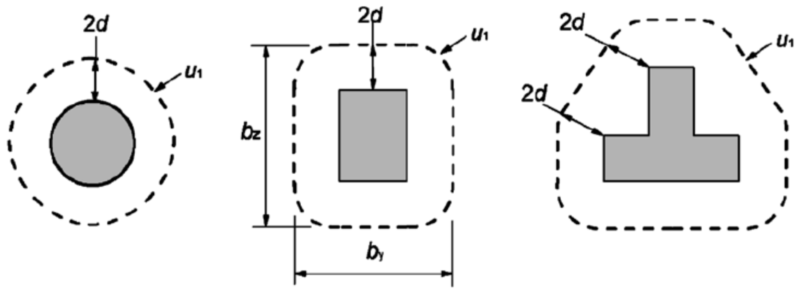
The **Longitudinal Reinforcement** tab (Fig. 7.5.6-4) is used to specify:

- the class and the diameter of longitudinal rebars (S1, S2, S3, S4), that make up the meshes parallel to the top and bottom faces of the slab;
- the distances to the center of gravity of reinforcement.

Peculiarities of the implementation

The punching analysis is implemented only for slab structures.

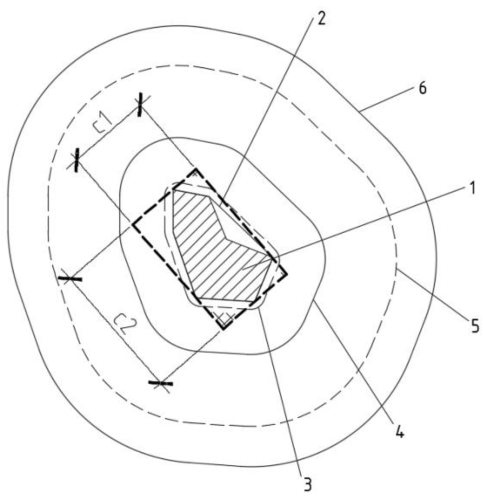
The distance to the free edges of a slab (for various schemes of punching) is taken into account only when the distance from the boundary of the load application area to the free edge does not exceed 3 effective heights (d) of longitudinal reinforcement of the slab along one of the axes, X, Y (free edge of the slab), or both axes X, Y (free corner of the slab). The punching analysis cannot be performed when the free edge of the slab falls within an area located at the distance of less than half the effective height of the slab to the load application area.



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

Transverse reinforcement is taken into account in the analysis within the full bearing pyramid with an offset of *2d* from the load application area to each side.

In the general case a convex hull ***Conv*** of the loaded area is drawn.



$Conv < u_{03} < u_1 < u_{arm} < u_{out}$ – perimeters

u_0 is the length of the convex hull (for an interior column) or the length of column periphery with c1 and c2 being the column dimensions (for a corner column or an edge column)

c1 and c2 are the dimensions of the minimum circumscribed rectangle with sides parallel to the principal axes of inertia of the column section.

1 – column section;
2 – convex hull of a column section ***Conv***;
3 – perimeter u_{03} at a distance 0,3*d* from *Conv*;
4 – punching perimeter u_1 at a distance 2*d* from *Conv*;

5– perimeter u_{arm} (equidistant) is drawn within the perimeter u_{out} with an offset of 1,5*d* from it;

6 – equidistant to the convex hull *Conv* with a length equal to u_{out}, the transverse reinforcement is considered only within its limits.

The reinforcement is considered in the punching analysis taking into account the transverse reinforcement within ($u_{arm} - u_{03}$**).**

If the distance from the transverse rebars to the load application area is large enough, their presence has little effect on the ultimate force resisted by the transverse reinforcement at punching.

Since the application does not limit the number of the specified reinforcing bars in a row, the following limitation has been introduced: to accept the boundary of the area of the check analysis without taking into account the reinforcement at the distance of 0,3*d* not from the boundary of the location of the transverse reinforcement specified by the user, but from the boundary of the area within which the analysis takes into account the transverse reinforcement.

Limitations

Limitations dealing with the design codes:

- lateral vertical reinforcement is implemented only in the case when the load application area is located in the middle part of the slab;
- structural limitations of the classes and diameters of rebars, of the spacing between rebars in meshes and between meshes, and other limitations.

Additional lateral reinforcement is implemented only as vertical meshes with effective vertical rebars. The meshes are parallel to the sides of the slab and do not fall within the load application area. Meshes nearest to the load application area are arranged in such a way that they always fall within the bearing prism.

Limitations related to the legitimacy of a check and the ratios of loads are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of rebar rows, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

7.5.7 Tearing

This mode implements checks of joints of members of reinforced concrete structures for tearing in compliance with Sec. 3.43 of SNiP 2.03.01-84*, KMK 2.03.01-96, Sec. 3.97 of the Guide to SNiP 2.03.01-84*, and Sec. 3.121 of the Manual on designing of concrete and reinforced concrete structures made of heavy-weight concrete [12].

Additional reinforcement is always present in the area where the joint is checked for tearing.

The strength from conditions of local tearing is calculated in the result of the check.

General

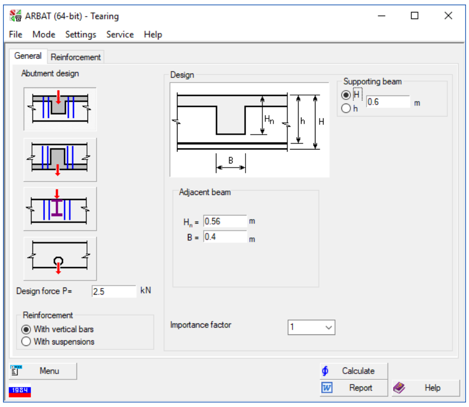


Figure 7.5.7-1. *The* **General** *tab*

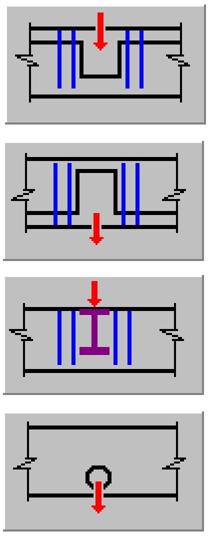


Figure 7.5.7-2. *Designs checked for tearing*

The **General** tab (Fig. 7.5.7-1) is used to select one of four designs checked for tearing (Fig. 7.5.7-2).

First design is a monolithic reinforced concrete floor where beams join in the upper area.

Second design is a monolithic reinforced concrete floor where beams join in the lower area.

Third design is an adjacent steel beam (an I-beam is shown in the figure). The adjacent beam is defined by the height and width of its cross-section.

Fourth design is the application of a concentrated force transferred by suspending a weight through a hole in the beam.

The respective radio buttons of the **Reinforcement** group are used to select whether the area of tearing is reinforced with vertical bars or with suspensions.

Once the abutment design is selected, you have to specify:

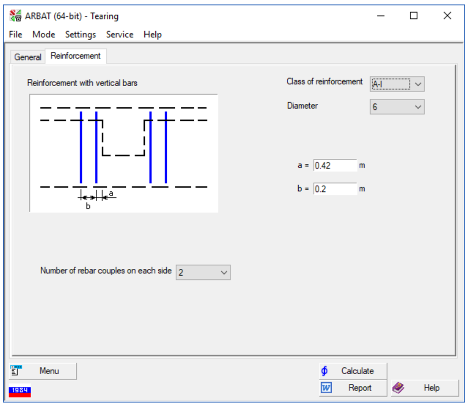
- the design load;
- the sizes of the supporting and adjacent beams (for designs 1 - 3) and the location of the adjacent beam (for design 3);
- the sizes of the supporting beam, the location and the diameter of the hole (for design 4).

For any checked design you have to specify the height of the supporting beam (*H*), or the effective height of its section (*h*) where *h* is the distance from the top of the beam to the center of gravity of the lower reinforcement (it can be selected using the respective radio button). If the beam height parameter *H* is used, the value of *h* is taken as 0.9*H*.

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

Reinforcement

a



b

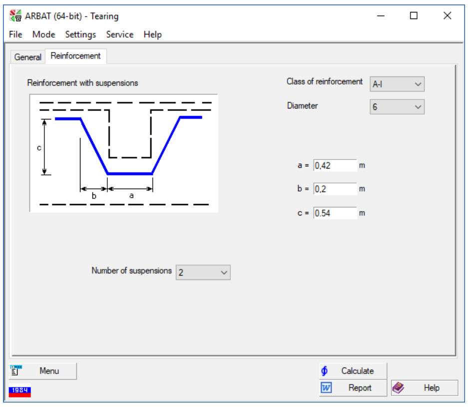


Figure 7.5.7-3. *The* **Reinforcement** *tab*

Depending on the selected reinforcement, this tab is used to specify the data on vertical bars (Fig. 7.5.7-3, *a*) or suspensions (Fig. 7.5.7-3, *b*).

The following data has to be specified for the reinforcement with vertical bars:

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

The following data has to be specified for the reinforcement with suspensions:

- the class and the diameter of reinforcement;
- the number of suspensions;
- the sizes of the suspensions (in the check of the fourth design, the size of the horizontal part of the suspension does not have to be specified).

Peculiarities of the implementation

There is an important problem of transferring the load from the adjacent beam to the supporting one. The following approach is used:

- in the check of designs 1, 2 (joints between reinforced concrete beams), the load is transferred at the level of the center of gravity of the compressed area of concrete of the adjacent beam. The height of the compressed area of concrete is assumed by default to be equal to 40% of the height of the adjacent beam (you can change the height of the compressed area);
- in the check of the design 3 (adjacent steel beam), the load is transferred uniformly over the height of the adjacent beam;
- in the check of the design 4 (suspension of a weight through a hole in the beam), the load is transferred at the level of the hole bottom, and the width of the load transfer area is assumed to be equal to the half of the hole diameter.

Additional reinforcement (vertical bars, suspensions) is arranged symmetrically with respect to the joint.

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

Limitations

The following reinforcement can be used for various structural designs of the joints:

- reinforcement with vertical bars can be used for designs 1, 2, 3;
- reinforcement with suspensions can be used for designs 1, 3, 4.

Limitations related to the design arrangement of a joint are terminal: once they are violated, the respective analysis cannot be performed.

In most cases limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the number of rebars, the reinforcement class etc. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

- if vertical bars are used, the number of rebar couples on each side of the joint is from 1 to 3;
- if suspensions are used their number is from 1 to 3;
- the distance between vertical bars is not less than 50 mm;
- the reinforcement classes are Bp-I, A-I, A-II, A-III according to SNiP 2.03.01-84* and A400C according to TSN 102-00**;**
- the diameters of the rebars (depending on the class) are from 3 to 14 mm.

The check determines the number of suspensions and bars which fall within the area of tearing. If any of the bars do not fall within the area of tearing, the respective message is displayed on the screen and included in the report.

7.5.8 Inserts

This mode implements checks of anchors of welded steel inserts which consist of a flat plate and normal or/and oblique anchors welded to it. The outer side of the plate of the insert is located in one plane with the outer surface of the reinforced concrete member.

The checks are performed in compliance with the regulations of Sec. 3.44 - 3.46 of SNiP 2.03.01-84*, KMK 2.03.01-96, Recommendations on designing of steel inserts [9] and Annex B of SP 63.13330.

Three types of anchors of the inserts are checked:

- normal anchors tee-welded to the plate;
- normal anchors tee-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 be reinforced on their ends.

The strength of an insert plate is not checked because there is no information about the design of a bearing part supported by it (a table, a rib etc.). Only the compliance of the plate thickness and the anchor diameters to the technological welding requirements is checked. The plate of an insert is not provided with additional retaining parts and devices transferring some part of the lateral load onto concrete.

The following factors are checked for all the considered types of anchors:

- strength of the most stressed anchor;
- embedment length of the anchor in tension;
- bearing of concrete under the reinforcement of the most compressed anchor;
- spalling of concrete under an anchor in tension at the edge of the reinforced concrete element (if the distance from the edge of the insert to the edge of the element is specified).

General

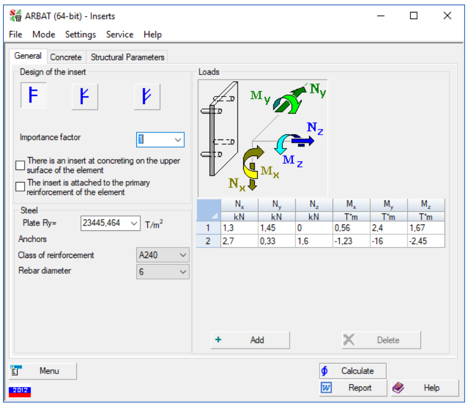


Figure 7.5.8-1. *The* **General** *tab*

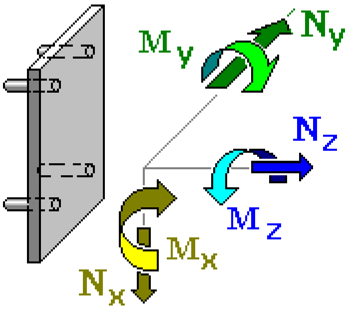


Figure 7.5.8-2. *Loads on inserts*

The **General** tab (Fig. 7.5.8-1) is used to specify:

- the type of an insert;
- attachment of the anchors of the insert to the primary reinforcement of the element (if there is an attachment, only the strength of the anchors will be checked);
- the class of steel for the plate of the insert, the class and diameter of the anchor reinforcement (anchors of the same type are assumed to have the same diameter and the same class of reinforcement);
- loads on the insert applied to the center of the outer surface of the plate.

An insert of the first type can transfer 6 force components to a reinforced concrete member: 3 forces along the coordinate axes and 3 moments with their vectors parallel to the respective axes. Positive directions of the forces and moment vectors are shown in Fig. 7.5.8-2.

An insert of the second or third type can transfer only three force components lying in the XoZ plane to a reinforced concrete member (Fig. 7.5.8-2).

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 6.4.2.

Structural Parameters

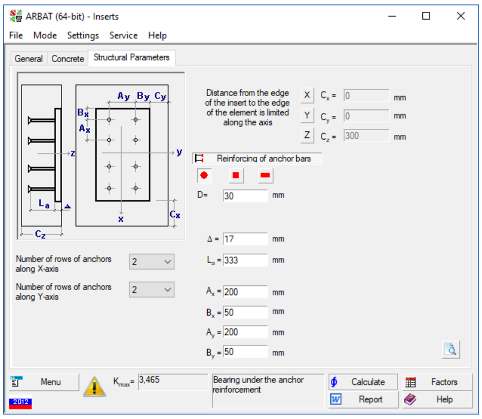


Figure 7.5.8-3. *Structural parameters for an insert of the 1st-type*

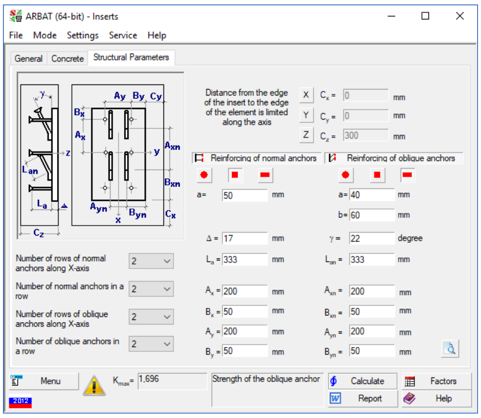


Figure 7.5.8-4. *Structural parameters for an insert of the 2nd-type*

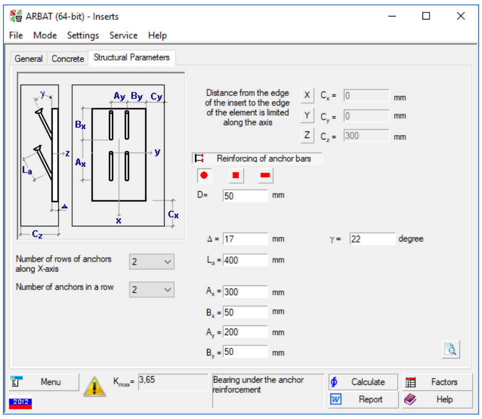


Figure 7.5.8-5. *Structural parameters for an insert of the 3rd-type*

This tab (Fig. 7.5.8-3–7.5.8-5) is used to specify the following data for each type of inserts:

- distances from the edges of the insert to the faces of the reinforced concrete element along the coordinate axes (if it has been indicated in the **General** tab that there are such limitations);
- reinforcing of anchor bars, types of reinforcing (round, square or rectangular plates) and sizes;
- thickness of the insert plate;
- lengths of the anchors;
- the number and arrangement of anchors on the insert;
- the angle of inclination of the anchors (only for inserts of the 2nd and 3rd type).

Limitations

The limitations of the current version are based on the structural limitations of the classes and diameters of rebars used for anchors, distances between the anchors, ratios of the plate thickness and the diameters of the anchor bars, the minimum number of anchors, and other limitations given in SNiP 2.03.01-84*, KMK 2.03.01-96 and Recommendations on designing of steel inserts [9].

Limitations related to the legitimacy of a check are terminal: once they are violated, the respective analysis cannot be performed.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the reinforcement class. You only receive warnings in the reports and on the screen about the violation of other limitations.

The following structural limitations are implemented in the application:

- anchor bars can be made of reinforcement of the following classes: A-I, A-II, A-III, A-IV, and A-400;
- diameters of reinforcement can be from 8 to 25 mm depending on the reinforcement class;
- limitations on the arrangement of reinforcement comply with Sec. 5.5 of the Recommendations [9]**;**
- the number of rows of anchor bars can be from 1 to 4 depending on the insert type. The number of anchor bars in each of the rows is also from 1 to 4;
- not less than two normal or oblique anchors must be installed in the plane of the bending moment (M_x , M_y). Not less than two normal anchors must be installed to resist the torque (M_z);
- if the anchors are made of the reinforcement of class A-I (A240), then they are always reinforced.

Peculiarities of the implementation

Peculiarities of the implementation are based on the assumptions and limitations for checks defined in SNiP 2.03.01-84*, KMK 2.03.01-96 and the Recommendations [9]. They are as follows:

- when checking concrete for bearing under the anchor reinforcement, it is assumed that the effective area of concrete is always considerably greater than the area of the anchor reinforcement. The φ_6 factor according to Sec. 3.39 of SNiP 2.03.01-84*, KMK 2.03.01-96 is always 2.5;
- when checking concrete for bearing under the anchor reinforcement, it is assumed that a crack can always appear in concrete along the anchor;
- it is assumed in the check of the embedment length of the tensile anchor in concrete that the whole anchor is in the tension area of concrete. Formula (62) of the Recommendations [9] assumes ω_{an} = 0,7 and φ_c = 1;
- the checks for punching of concrete under anchors in tension are not performed.
- if the anchor is reinforced, it is assumed that the required length of the bar anchorage does not exceed ten diameters of the anchor (see Sec. 5.8 of the Recommendations [10]).
- the concrete spalling check is performed for the edge of the concrete element, i.e. under "constrained" conditions for the formation of the anchor spalling prism (when the specified values Cx, Cy are insufficient). If these values are not specified, it is considered that there is a sufficient concrete mass and the spalling does not occur.

7.5.9 Short Cantilevers

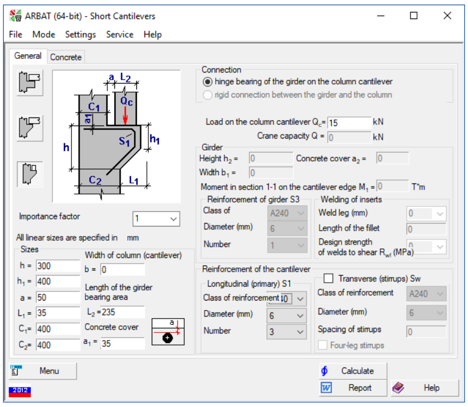


Figure 7.5.9-1. *The* **General** *tab*

This mode is used to check short cantilevers for the action of a lateral force to ensure the strength in a compressed oblique strip between the load and the support in compliance with Sec. 3.34 of SNiP 2.03.01-84*, KMK 2.03.01-96 and Sec. 3.99 of the Guide to SNiP [7], and Annex G of SP 63.13330.

The check is performed once all the necessary conditions and requirements to the short cantilevers are satisfied. The main condition is $S_1 \leq 0{,}9h_0$, where $h_0=h-a_1$ is the effective height of the support section of the cantilever, S_1 is the distance from the inner face of the lower part of the column to the end of the bearing area of the beam. Only single-sided cantilevers of three types are considered: a rectangular cantilever, a cantilever with a slope to support a girder, and a cantilever with a slope to support a crane beam. Click on the respective icon to select the type of the cantilever.

General

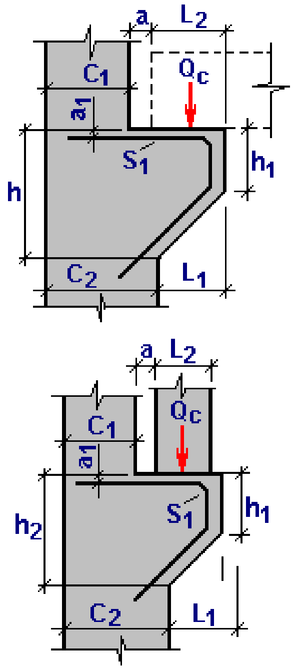


Figure 7.5.9-2. *Types of short cantilever*

The **General** tab (Fig. 7.5.9-1) is used to specify:

- a joint type (Fig. 7.5.9-2):
  (a) a crane beam is supported by the cantilever;
  (b) a floor beam (girder) is supported by the cantilever;
- the sizes of the cantilever and the column:
  the height of the cantilever in the support section, **h**; the height of the free end of the cantilever, $\mathbf{h_1}$; the cantilever overhang length, $\mathbf{L_1}$; the actual length of the bearing area of the beam along the cantilever overhang, $\mathbf{L_2}$; the distance from the inner face of the column to the side face of the beam, **a**; the height (of the section) of the upper part of the column, $\mathbf{c_1}$; the height (of the section) of the lower part of the column, $\mathbf{c_2}$; the width of the column or the cantilever, **b**; the concrete cover of the longitudinal reinforcement, $\mathbf{a_1}$;
- the type of connection between the girder and the column cantilever;
- the capacity of the crane, **Q**, which works in the span between the columns, if a crane beam is supported by the cantilever;
- the load, $\mathbf{Q_c}$, on the cantilever caused by the imposed weight;
- the moment in the girder section on the cantilever edge, if the girder and the cantilever are connected rigidly;
- the class, number and diameter of the longitudinal (primary) reinforcement;
- parameters of the welding of inserts:
  weld leg, length of the fillet weld, design strength of welds to shear;
- the class, diameter, and spacing of the transverse reinforcement (stirrups).

Concrete

The **Concrete** tab is generally similar to those described in the previous modes, but you do not have to specify the conditions of hardening here, and a service factor for concrete, γ_{b2}, has to be entered.

Peculiarities of the implementation

Cantilevers with haunches with an arbitrary angle of slope are considered. If the height of a cantilever is 100 mm or less, or if its overhang is $\mathbf{L_1}$ = 100…150 mm, it can be designed as rectangular.

The dimensions of the cantilevers are specified on the following basis: the height of the cantilever, **h**, in its support section must not be less than 250 mm; the height of the free edge, $\mathbf{h_1}$, of cantilevers bearing precast crane beams is taken depending on the design capacity of the crane, **Q**, according to the following rules:

$Q \le 5$ 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 in the oblique compressed strip between the load and the support is checked irrespective of whether there is or there is no transverse reinforcement.

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

The width of the cantilever, **b**, is set equal to the width of the column. The width of the girder, **b_1**, is set to be less than or equal to the width of the cantilever.

Two types of connections between the beam and the column are considered: a hinge connection, and a rigid one with joint grouting and welding of the lower reinforcement of the girder to the cantilever reinforcement through inserts. If, in the rigid connection, the **M_1** moment puts the lower face of the girder in tension, then it should be used with the “minus” sign.

Transverse reinforcement for the cantilevers consists of:

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

where **C** is the distance from the inner face of the lower part of the column to the application point of force **Q_c**.

The spacing of stirrups must (in all cases) be not greater than **h**/4 and not greater than 150 mm.

By default, the transverse reinforcement of the cantilever is implemented as two-leg 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-leg stirrups (when the respective checkbox is checked), the design formulas will include the quadruple area of the transverse reinforcement.

In all cases if the cantilever overhang, **L_1**, is less than the length of the bearing area, **L_2**, the analysis takes into account only the load on the cantilever located within the cantilever overhang.

Concrete of the cantilever under the bearing area is checked; the bearing stresses in the areas where the load is transferred onto the cantilever must not exceed **$R_{b,loc}$** (Sec. 3.39 of SNiP II-23-81*), otherwise it is necessary to increase the class of concrete or the load transfer area. In the check for local compression, the bearing area is taken equal to its design value.

In the check of the longitudinal reinforcement, if the girder and the column are connected rigidly, one of the limitations of the horizontal force, **N_s**, depends on the height and length of the fillet weld connecting inserts of the girder and those of the cantilever. The minimum weld leg, **K_f**, is assigned in compliance 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 welded members, and varies between 3 and 12 mm. The design strength of the fillet welds for shear in the metal, **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 fillet weld, **l_w**, connecting the inserts of the girder and those of the column is determined by the sizes of the inserts, but must not exceed the double length of the load transfer area along the cantilever overhang, 2**L_2**, and must be not less than 4**K_f**.

Limitations

Limitations related to the design arrangement of the joint are terminal: once they are violated, the respective analysis cannot be performed.

Since the application operates only with the class and diameter of longitudinal and transverse reinforcement, it is the user who is fully responsible for the design arrangement of reinforcement in the cantilever, installation of the proper anchorage (Sec. 5.14 of SNiP 2.03.01-84*, KMK 2.03.01-96 and Sec. 5.45 of the Manual [7]) for the longitudinal (primary) reinforcement, and the type of transverse reinforcement (see recommendations above).

The application of rigid reinforcement to reinforce a cantilever of a limited height is not considered.

Limitations related to the structural requirements are not terminal: the analysis can be performed, but the user gets a warning that the limitations have been violated. Some of these limitations are implemented in the following way: you can select certain values only from a limited list, e.g., the minimum weld leg, 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 transverse rebars should be of class A-I.

7.5.10 Keys

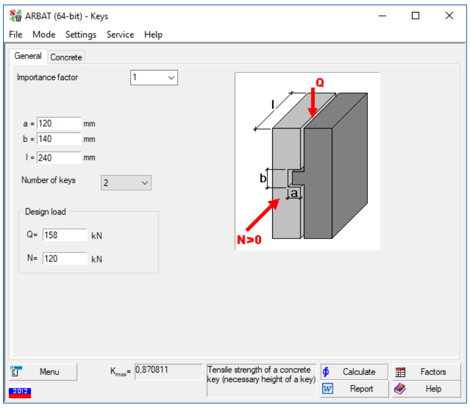


Figure 7.5.10-1. *The* **General** *tab*

This mode implements checks of concrete keys with the specified parameters under the action of shear and longitudinal forces. Parameters of keys (depth, height and length) and their ratios are not checked.

Checks are performed in compliance with the regulations of Sec. 3.115 of the Guide to SNiP 2.03.01-84 * and Sec. F.1 of SP 63.13330.

This mode enables to check the compressive strength of a concrete key (necessary depth of a key) and its tensile strength (necessary height of a key) at its constant length and the specified class of concrete (see Fig. 7.5.10-1)

The number of keys entered in the calculation must be not more than three. However, if the tensile force is acting on the structure, it is allowed to take into account the behavior of five keys (Note to Sec. 3.115 of the Guide to SNiP 2.03.01-84 * or the last paragraph of Sec. F.1 of SP 63.13330).

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

7.5.11 Shear between Web and Flanges of T-sections (EN 1992-1-1)

This mode enables to perform the shear analysis of the flange of a T-section according to EN 1992-1-1. We consider a part of the flange with axial forces N_1, N_2 and bending moments M_1,M_2 applied at the ends in such a way so that the flange is either in compression at both ends or in tension at both ends. The length of the considered part is Δx. The stress state in a section is determined on the basis of a nonlinear deformation model (with bilinear stress-strain diagrams for concrete and reinforcement) without taking into account the behavior of concrete in tension. The checks are performed according to Sec. 6.2.4 of EN 1992-1-1.

Initial data are specified in the multi-tab dialog box.

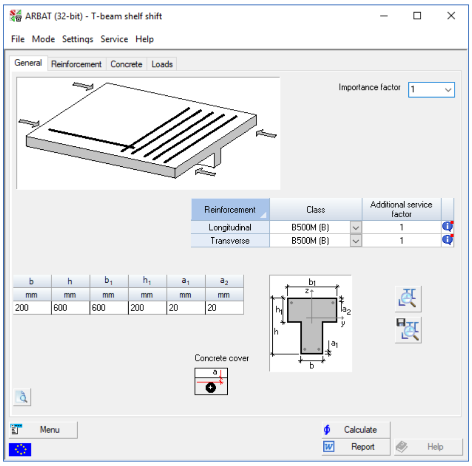


Figure 7.5.11-1. *The* **General** *tab*

The **General** tab (Fig. 7.5.11-2) is used to specify the dimensions of the T-section, the classes of longitudinal and transverse reinforcement and the importance factor.

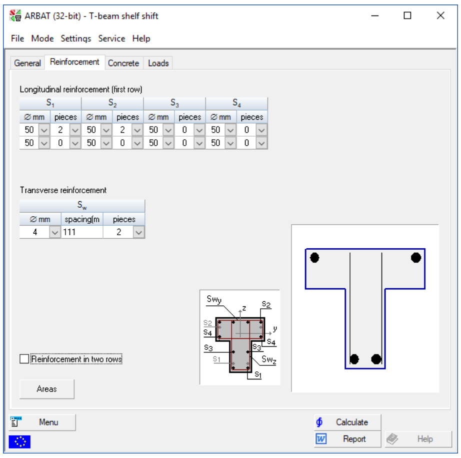


Figure 7.5.11-2. The **Reinforcement** tab

The information on the arrangement of longitudinal and transverse reinforcement is specified in the **Reinforcement** tab (Fig. 7.5.11-2) in the same way as in all other modes.

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 6.4.2.

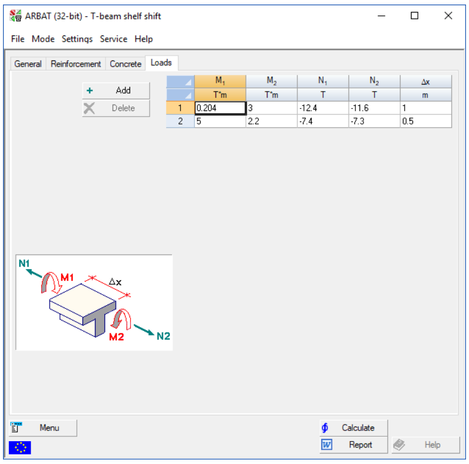


Figure 7.5.11-3. The **Forces** tab

The **Forces** tab (Fig. 7.5.11-3) is used to specify internal forces and moments in different parts of the flange. This tab is similar to the respective tab of the **Strength of Sections** mode. The only difference is that you also have to specify the moments and axial forces at the ends of each part and the length of the considered part.

7.5.12 Shear at the interface between concrete cast at different times (EN 1992-1-1)

This mode enables to check the design shear resistance at the interface between concrete cast at different times in compliance with Sec. 6.2.5 of EN 1992-1-1. It is assumed that the requirements of Sec. 6.2.1-6.2.4 of EN 1992-1-1 are satisfied for the considered section.

Initial data are specified in a multi-tab dialog box.

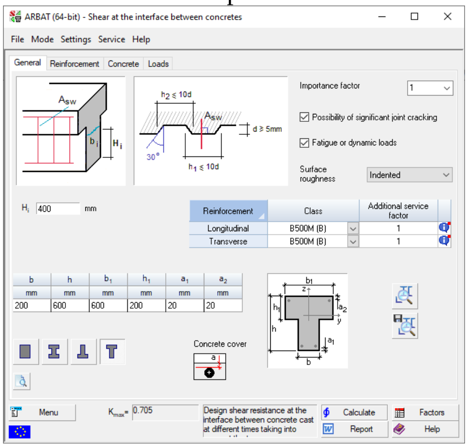


Figure 7.5.12-1. The **General** tab

The **General** tab (Fig. 7.5.12-2) is used to specify the cross-sectional dimensions, the classes for longitudinal and transverse reinforcement, and the importance factor. You also have to select the type of surface roughness (very smooth, smooth, rough or indented), and specify the possibility of significant joint cracking and of fatigue or dynamic loads.

The standard **Reinforcement** tab is used to specify the information on the arrangement of longitudinal and transverse reinforcement.

Properties of concrete are specified in the standard **Concrete** tab.

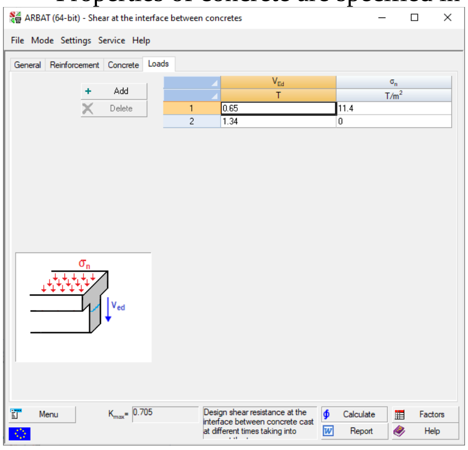


Figure 7.5.12-2. The **Forces** tab

The **Forces** tab (Fig. 7.5.12-2) is used to specify forces acting in the considered section. Transverse shear force V_{Ed} is acting in the section, as well as the normal stress σ_n, caused by the external normal force across the interface that can act simultaneously with the shear force, positive for compression and negative for tension.

7.6 Selection of Reinforcement

The program does not check the structural limitations on the placement of reinforcement during its selection, assuming that the user can perform this check in an informal way. Moreover, having obtained the results of the selection and ascertained that the structural limitations cannot be met, the user can decide to correct the formwork dimensions.

7.6.1 Selection of Beam Reinforcement

This mode is used to select reinforcement in multi-span continuous beams of a constant cross-section using the criteria of strength and crack resistance in compliance with regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SNiP 52-01-2003, SP 52-101-2003, SP 63.13330). Bending of the beam in one load plane under distributed and concentrated loads combined into load cases is considered. The latter can be classified by their physical origin and properties into permanent, temporary long-term, temporary short-term, wind, and snow. The selection is performed for the automatically created design combinations of forces (DCF). DCF factors which take into account the nature of the load case are assigned by the application in compliance with the regulations of SNiP 2.01.07-85*. If the selected design code is SP 63.13330, then the loads are combined according to the regulations of SP 20.13330.

It is assumed that the beam is not subjected to longitudinal forces and only the following force actions are taken into account:

M – bending moment;

Q – shear force.

The analysis can be performed for a beam of a rectangular section, T-section or I-section. The areas of upper and lower reinforcement in the segments, and the area and spacing of the bars of transverse reinforcement are obtained in the result of the analysis. The selected reinforcement is assumed to be the same along a particular segment, and the user should specify the number and lengths of segments the beam is divided into.

Initial data for the selection of reinforcement are entered in the following tabs: **General**, **Loads**, **Concrete**, and **Crack Resistance**, and results are analyzed in the **Results** tab.

General

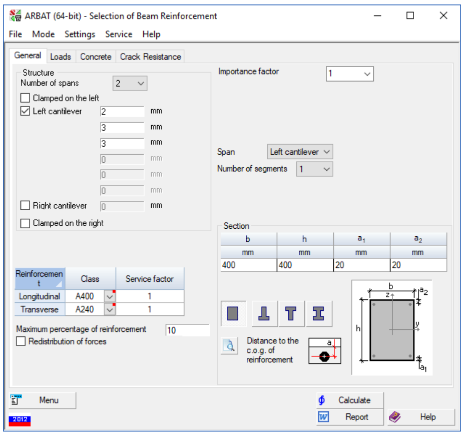


Figure 7.6.1-1. *The* **General** *tab*

The **General** tab (Fig. 7.6.1-1) is used to specify the geometric properties of a multi-span beam and the type of its section, to enter the sizes of the section and the distance to the center of gravity of reinforcement, to specify the number and lengths of segments in the span, and to indicate the class and service factors for longitudinal and transverse reinforcement. Entering all this information is similar to working in the **Beam Check** mode (see Section 7.4.3).

The number of segments for each span (cantilever) is selected from the **Number of segments** drop-down list. The number of span is selected from the **Span** list.

The **Specify the lengths of segments** group enables to select the way the lengths are specified using the following radio buttons:

- **Absolute** — the lengths of the segments will be specified in the units of length;
- **Relative** — the lengths of the segments will be specified as percentage of the total span length.

Depending on the way the lengths are specified, you have to fill the table with either the lengths of the segments or their percent ratios for each span.

The shape of the section is selected according to the standard rules (see Section 7.3.1). It should only be noted that the distances to the center of gravity of reinforcement, $\mathbf{a_1}$ and $\mathbf{a_2}$, are specified here.

The arrangement of longitudinal and transverse reinforcement in the considered sections is shown in Figs. 7.6.1-2 and 7.6.1-3, respectively.

This tab is also used to specify the maximum percentage of reinforcement; when this value is exceeded, it means the reinforcement could not be selected.

The way the forces are redistributed if the respective checkbox is checked is described in the **Beam Check** section.

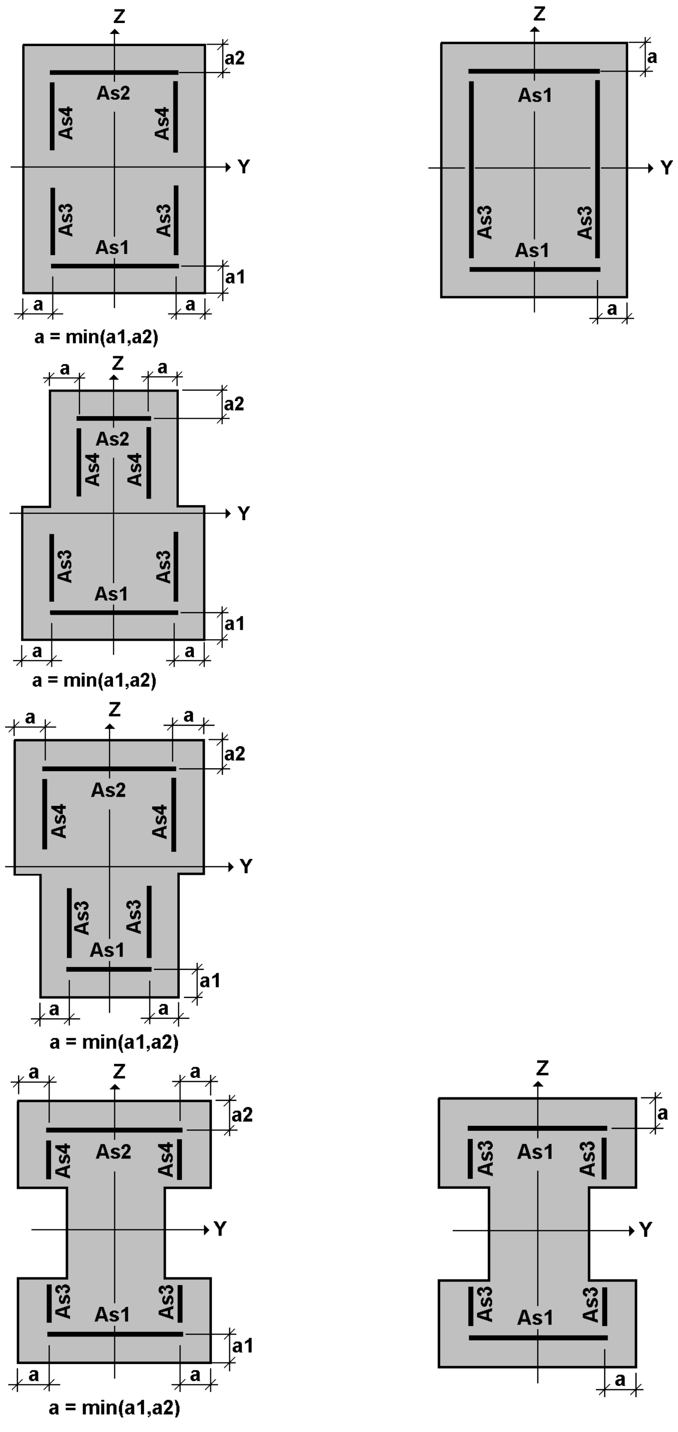


Figure 7.6.1-2. *Arrangement of “areas” of longitudinal reinforcement*

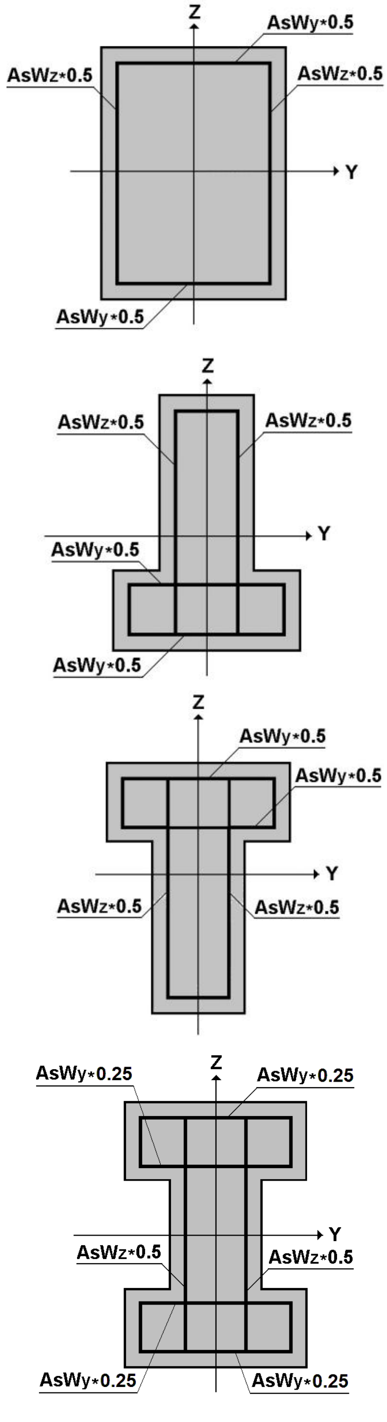


Figure 7.6.1-3. *Arrangement of transverse reinforcement*

Loads

The loads are specified according to the same rules as in the **Beam Check** mode (see Section 7.6.1).

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Crack Resistance

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

Moreover, 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 outer row of longitudinal reinforcement is greater than that to the center of gravity of reinforcement specified in the **General** tab, it means that the reinforcement has to be installed in two rows and the reinforcement will be selected for the two-row arrangement.

By using the results of the selection of the reinforcement area and the **Rebar Calculator** service tool (see Section 17.1.3), you can determine the necessary diameter and the number of rebars. If the diameter of rebars turns out to be different from the one specified in the **Crack Resistance** tab, you will have to adjust the diameter of longitudinal rebars and perform a new calculation.

Results

Once the initial data are prepared, click the **Calculate** button to select the reinforcement. The **Results** tab will open (Fig. 7.6.1-4) and display the results of the selection in the form of diagrams. The type of reinforcement in the diagrams, the percentage of reinforcement, and the crack opening width are selected from the drop-down list in the top right corner of the window. For segments highlighted in red, the reinforcement could not be selected. Information about the reasons for this can be obtained from the table of results (this result can also depend on the specified maximum percentage of reinforcement).

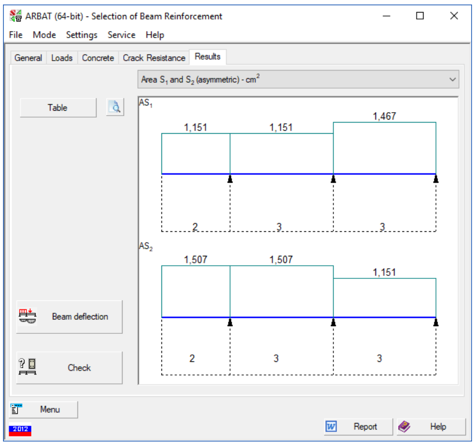


Figure 7.6.1-4. *The* **Results** *tab*

Depending on the item selected in the list, the diagrams of reinforcement can be displayed separately for each type of reinforcement or in pairs. For example, you can have the diagrams AS_1 and AS_2 displayed together. Clicking the button  will open the **Preview** dialog box which shows areas of the selected longitudinal reinforcement in sections of each span of the beam. If the reinforcement could not be selected, the respective section is displayed in red.

Tabular data are displayed in a separate dialog box, **Results** (Fig. 7.6.1-5), which opens by clicking the **Table** button. Results of the selection for each segment are given in one line if the crack resistance criterion does not require any additional reinforcement or in two lines if such reinforcement is required. The first line displays 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 selected for a particular segment, the respective line of the **Type** column will display information about reasons why the error has occurred.

Depending on the option selected in the **Display the transverse**

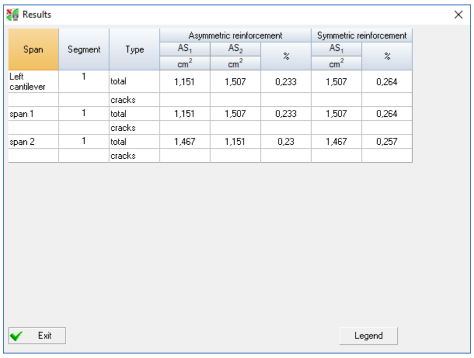


Figure 7.6.1-5. *The* **Results** *dialog box*

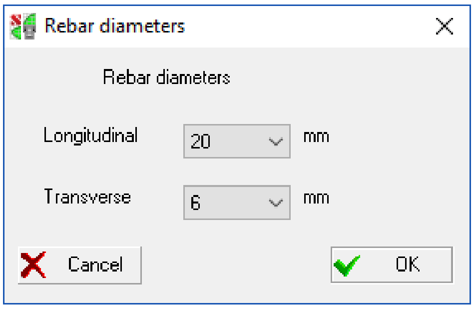


Figure 7.6.1-6. *The* **Rebar diameters** *dialog box*

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 selection of reinforcement or for the spacing specified by the user. In the latter case, click the **Apply** button after you enter the spacing.

A report can be generated on the basis of the results of the reinforcement selection (the **Report** button), which includes a model of the beam, the arrangements of loadings, sizes of the section, properties of concrete and reinforcement, diagrams of force actions for each load case, and diagrams and a table with the results of the selection.

The application enables to transfer the results of the reinforcement selection to the deflection calculation mode (the **Beam deflection** button) or to the check mode (the **Check** button). These two modes work with a particular reinforcement arrangement for which the rebars are defined automatically on the basis of diameters specified in the **Crack Resistance** tab (it is assumed that 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 performed for the first category of crack resistance, then the diameter of the rebars is specified by the user in the **Rebar diameters** dialog box (Fig. 7.6.1-6) that opens once the above modes are invoked.

7.6.2 Selection of Single-Span Beam Reinforcement

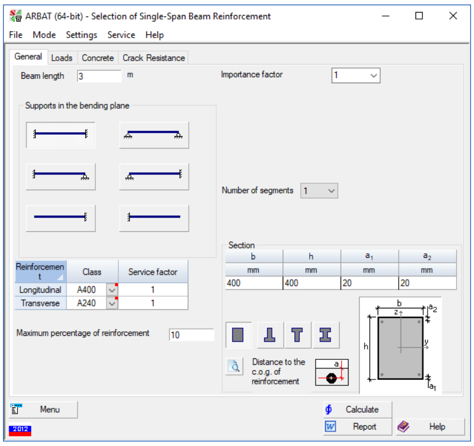


Figure 7.6.2-1. *The* **General** *tab of the* **Selection of Single-Span Beam Reinforcement** *mode*

This mode is similar to **Selection of Beam Reinforcement** mode (see the previous section). There are the following differences:

- only the length is specified for the beam (Fig. 7.6.2-1);
- the user has to select supports of the beam in its bending plane by clicking the respective button;
- redistribution of forces is not taken into account;
- you do not have to specify the span to which the load is applied in the **Loads** tab.

7.6.3 Selection of Column Reinforcement

This mode is used to select reinforcement in a column of a constant cross-section using the criteria of strength and crack resistance in compliance with regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SNiP 52-01-2003, SP 52-101-2003, SP 63.13330). An eccentric compression/tension with a biaxial eccentricity is considered. Checks of all sections are performed for the automatically created design combinations of forces (DCF). DCF factors which take into account the nature of the load case are assigned by the application in compliance with the regulations of SNiP 2.01.07-85*. If the selected design code is SP 63.13330, then the loads are combined according to the regulations of SP 20.13330.

If the analysis is performed according to SNiP 2-03-01-84* taking into account ultimate limit state only, the reinforcement is selected only for the following set of force actions:

N – longitudinal force;
M_y – moment that bends the member in the XoZ-plane with its vector along the Y-axis;
M_z – moment that bends the member in the XoY-plane with its vector along the Z-axis;
Q_z – shear force along the Z-axis;
Q_y – shear force along the Y-axis;
M_t – torque with its vector along the X-axis.

Otherwise (with serviceability limit state) only the following force actions are taken into account:
N — longitudinal force;
M_y — moment that bends the member in the XoZ-plane with its vector along the Y-axis;
Q_z — shear force along the Z-axis.

If the reinforcement is selected according to SNiP 52-01-2003 (SP 63.13330), the serviceability limit state can be taken into account for an arbitrary set of *N*, M_y, and M_z.

The analysis can be performed for a column of a rectangular section, T-section, I-section, ring, or round section. The results of the analysis include the areas of symmetric and/or asymmetric longitudinal reinforcement, and the area and spacing of transverse reinforcement on the segments of the column. The selected reinforcement is assumed to be the same along a particular segment, and the user should specify the number and lengths of segments the column is divided into.

Data are prepared in the following tabs: **General**, **Forces**, **Concrete**, and **Crack Resistance**, and the results are analyzed in the **Results** tab.

General

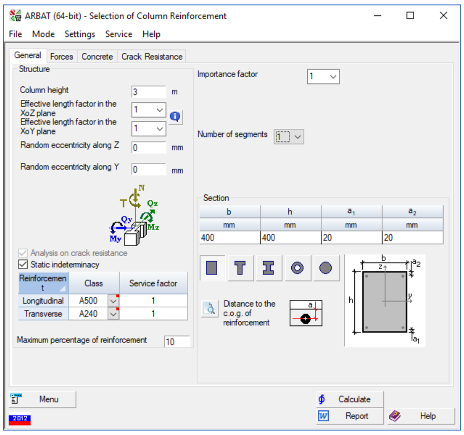


Figure 7.6.3-1. *The* **General** *tab*

The **General** tab (Fig. 7.6.3-1) is used to specify the height of a column, the type and sizes of its section, the distances to the center of gravity of reinforcement, the effective length factors, and values of the random eccentricities. You also have to select the number of segments of the column, and assign classes and service factors for longitudinal and transverse reinforcement. Moreover, the **Analysis on crack resistance** checkbox defines whether the column should be analyzed for the serviceability limit state, and the **Static indeterminacy** checkbox defines whether the column belongs to a statically determinate or indeterminate structure.

The number of segments of the column is selected from the **Number of segments** drop-down list.

The **Specify the lengths of segments** group enables to select the way the lengths are specified using the following radio buttons:

- **Absolute** — the lengths of the segments will be specified in the units of length;
- **Relative** — the lengths of the segments will be specified as percentage of the total column height.

Depending on the way the lengths are specified, you have to fill the table with either the lengths of the segments or their percent ratios. The numbering of the segments goes from the bottom to the top.

To select the shape of a section, click the respective button and enter the sizes of the section and the distances to the center of gravity of reinforcement, **a₁** and **a₂**, in the respective text fields. Clicking the button enables you to view the section shape in the **Preview** dialog box.

The arrangement of longitudinal and transverse reinforcement in the considered sections is shown in Figs. 7.6.3-2 and 7.6.3-3, respectively.

The effective length factors and the random eccentricities are specified in the same way as in the **Strength of RC Sections** mode (see Section 7.4.1).

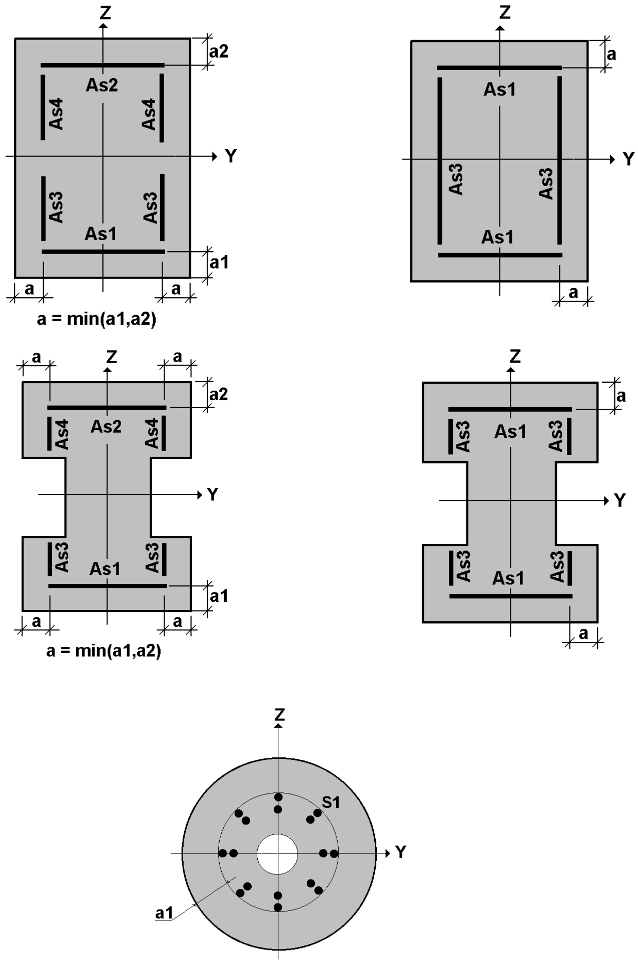


Figure 7.6.3-2. *Arrangement of "areas" of longitudinal reinforcement*

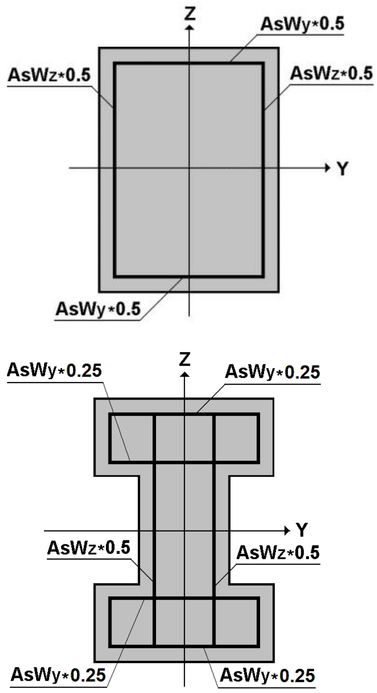


Figure 7.6.3-3. *Arrangement of transverse reinforcement*

Forces

The parameters of load cases are specified in the **Forces** tab according to the same rules as in the **Column Check** mode (see Section 7.4.7).

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Crack Resistance

Data on crack resistance are specified in the **Crack Resistance** tab according to the rules described in Section 7.3.4.

The tab is always accessible for the analyses according to SNiP 52-01-2003 (SP 63.13330) and when the **Analysis on crack resistance** checkbox (SNiP 2.03.01-84*) is checked in the **General** tab.

Results

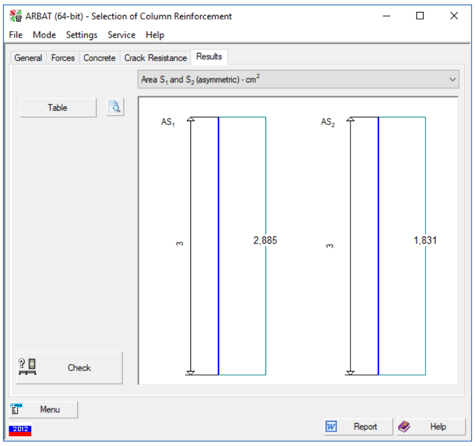


Figure 7.6.3-4. *The* **Results** *tab*

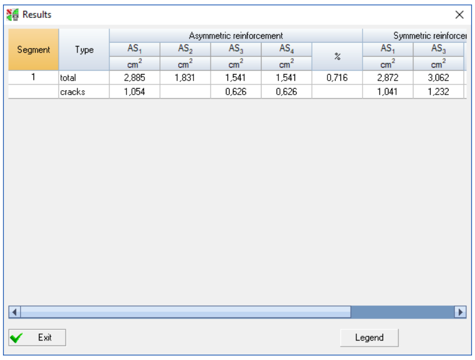


Figure 7.6.3-5. *The* **Results** *dialog box*

Once the initial data are prepared, click the **Calculate** button to select the reinforcement. The **Results** tab will open (Fig. 7.6.3-4), and display the results of the selection (the area of reinforcement, the percentage of reinforcement, the crack opening width) in the form of diagrams. The type of the displayed results is selected from the drop-down list in the top right corner of the window. For segments highlighted in red, the reinforcement could not be selected (this result depends on the specified maximum percentage of reinforcement). Information about the reasons for this can be obtained from the tables of results.

Depending on the item selected in the list, the diagrams of reinforcement can be displayed separately for each type of reinforcement or in pairs. For example, you can have the diagrams AS_1 and AS_2 or AS_3 and AS_4 displayed together. Clicking the button  will open the **Preview** dialog box which shows areas of the selected longitudinal reinforcement in sections of each segment of the column. If the reinforcement could not be selected, the respective section is displayed in red.

Tabular data are displayed in a separate dialog box, **Results** (Fig. 7.6.3-5), which opens by clicking the **Table** button. Results of the selection for each segment are given in one line if the crack resistance criterion does not require any additional reinforcement or in two lines if such reinforcement is required. The first line displays 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 selected for a particular segment, the respective line of the **Type** column will display information about reasons why the error has occurred.

Depending on the option selected in the **Display the 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 selection of reinforcement or for the spacing specified by the user. In the latter case, click the **Apply** button after you enter the spacing.

A report can be generated on the basis of the results of the reinforcement selection (the **Report** button), which includes a model of the column, the parameters of load cases, sizes of the section, properties of concrete and reinforcement, and a table with the results of the selection.

The application enables to transfer the results of the reinforcement selection to the check mode (the **Check** button). The number of rebars is determined according to the same rules as in the **Selection of Beam Reinforcement** mode (see Section 7.6.1).

Peculiarities of the implementation

Analysis of eccentrically compressed members

According to Section 3.24 of SNiP 2.03.01-84*, KMK 2.03.01-96, the structural analysis is based on the non-deformed model, where at the flexibility of l_0/i > 14 the effect of the member deflection on its strength has to be taken into account (l_0 is an effective length of the member, i is a radius of gyration in the load plane of the member cross-section).

Analysis of lightly reinforced sections

When calculating the crack opening width for a lightly reinforced section, at $M = M_{crc}$ the cracks open too widely at once. The application reduces the value of a_{crc} in the range $M_{crc} \leq M \leq M_0$ by multiplying it by a factor that allows for the behavior of concrete in tension above the crack [7].

Limitations

If SNiP 2.03.01-84*, KMK 2.03.01-96 is selected:

The analysis of reinforced concrete members made of cellular, porous, or self-stressing concrete is not implemented.

The analysis of prestressed reinforced concrete members is not performed.

The fatigue analysis is not performed.

The analysis for the serviceability limit state does not include the analysis of crack closing.

The flexibility limitations are not taken into account during the selection of longitudinal reinforcement.

7.6.4 Selection of Reinforcement in a Section

This mode enables to perform the selection of reinforcement according to the strength and crack resistance requirements of SNiP 2.03.01-84*, KMK 2.03.01-96 (SNiP 52-01-2003, SP 52-101-2003, SP 63.13330). This mode is similar to those described above for the selection of reinforcement in structural members (beam, column, ...). The main difference is that the selection is based on the forces in a section. The forces are specified in the same way as in the **Resistance of Sections** mode (including the possibility to import the design combinations of forces calculated by **SCAD**).

7.6.5 Selection of Slab Reinforcement (Karpenko' theory)

This mode is used to select reinforcement for shells, slabs and deep beams using the criteria of strength and crack resistance in compliance with regulations of SNiP 2.03.01-84*, KMK 2.03.01-96 (SNiP 52-01-2003, SP 52-101-2003, SP 63.13330.2012). Since the procedure for checking the reinforcement in two-dimensional elements is not given in the codes (except for SP 63.13330.2012), the method proposed by N.I. Karpenko [14] is used in **SCAD Office** (including **ARBAT**). The main guidelines of this method correspond to the general guidelines for the analysis of plane structures given in SNiP.

Initial data for the selection of reinforcement are entered in the following tabs: **General**, **Stresses**, **Concrete**, and **Crack Resistance** (if the **Analysis on crack resistance** checkbox is checked), and results are analyzed in the **Results** tab.

General

The following data are specified in the **General** tab (Fig. 7.6.5-1):

- plate thickness;

- member type;
- importance factor;
- distance to the center of gravity of reinforcement;
- whether to perform the analysis on crack resistance;
- whether to take into account the code requirements for minimum percentage of reinforcement;
- maximum percentage of reinforcement;
- class, service factor and diameter of longitudinal reinforcement;
- class, service factor and diameter of transverse reinforcement;
- minimum reinforcement parameters.

Member type — members of the shell, deep beam, or slab type are considered. Stresses N_x, N_y, T_{xy} for the members of slab type are taken as zero, and M_x, M_y, M_{xy}, Q_x, Q_y — for the members of deep beam type.

Importance factor — the value is selected from the drop-down list. Depending on the selected safety codes the importance factor will be applied either to design, or to design and characteristic values of forces/stresses.

Distance to the c.o.g. of reinforcement — can be defined by two or four numbers. In the first case, the value $\mathbf{a_1}$ corresponds to the distance to the center of gravity of the lower reinforcing mesh, and $\mathbf{a_2}$ corresponds to the distance to the center of gravity of the upper reinforcing mesh. In the second case, $\mathbf{a_1}$ and $\mathbf{a_2}$ are specified for the reinforcement placed respectively below and above along the X_1 axis, and $\mathbf{a_3}$ and $\mathbf{a_4}$ are specified for the reinforcement along the Y_1 axis.

Analysis on crack resistance — if the respective checkbox is checked (it is checked by default), the analysis will be performed taking into account the crack resistance.

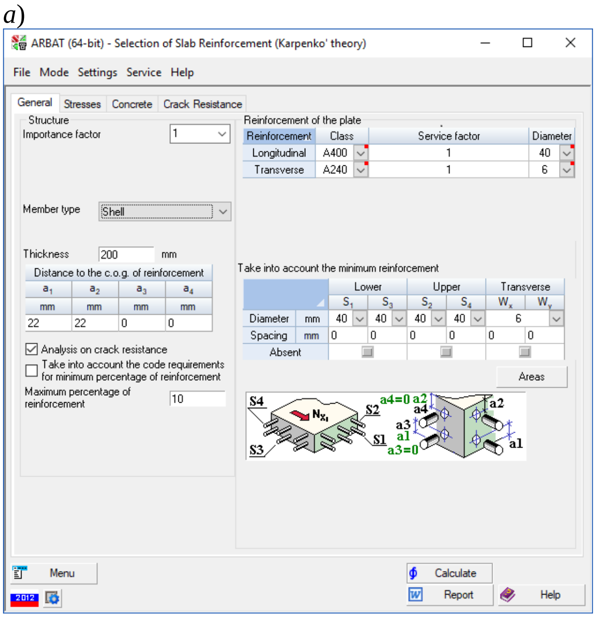
Take into account the code requirements for minimum percentage of reinforcement —when the respective checkbox is checked the area of the reinforcement per running meter provided in the result of the selection is not less than the area necessary to meet the requirements of the codes for the minimum percentage of reinforcement.

Minimum reinforcement — the data on reinforcement specified in the respective table are considered as minimum reinforcement so that the calculated reinforcement area is not less than the area of the given reinforcement.

Maximum percentage of reinforcement — if this value is exceeded, it will be impossible to select reinforcement.

Plate reinforcement — class, service factor and diameter of longitudinal and transverse reinforcement are specified in this table (the diameter is specified only in the case of the analysis on crack resistance).

a)



b)

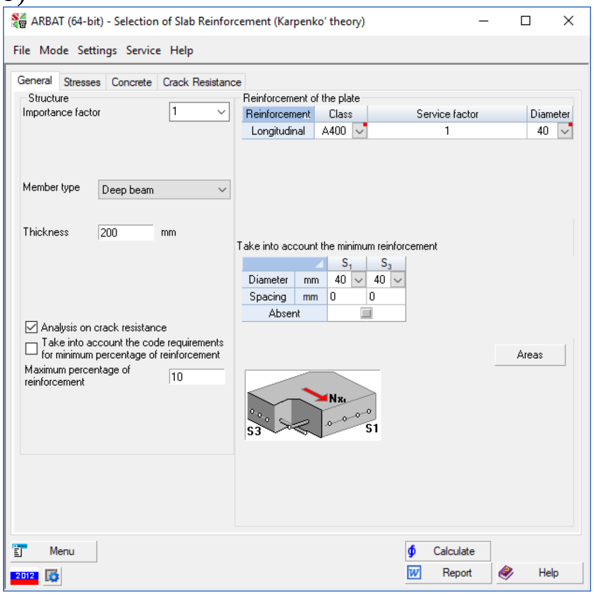


Figure 7.6.5-1. *The* **General** *tab of the* **Selection of Slab Reinforcement (Karpenko' theory)** *dialog box*

a) shell/ slab *b*) deep beam

Stresses

The **Stresses** tab (Fig. 7.6.5-2) enables to specify stresses in an element. Stresses can be specified by the user or loaded from a file with the **.rsu2** extension (button) with the values of the design combinations of forces (DCF) calculated in SCAD. The file is generated in the **DCF** section of the **Information on the Element** mode by clicking the **Export** button.

Each set of force factors must contain design, characteristic, and characteristic long-term stress values. Moreover, depending on the actions included in the combination they can be specified as seismic, short-term or special using the respective checkboxes.

The **Add** and **Delete** buttons enable to add new rows to the table or to delete the existing ones, respectively.

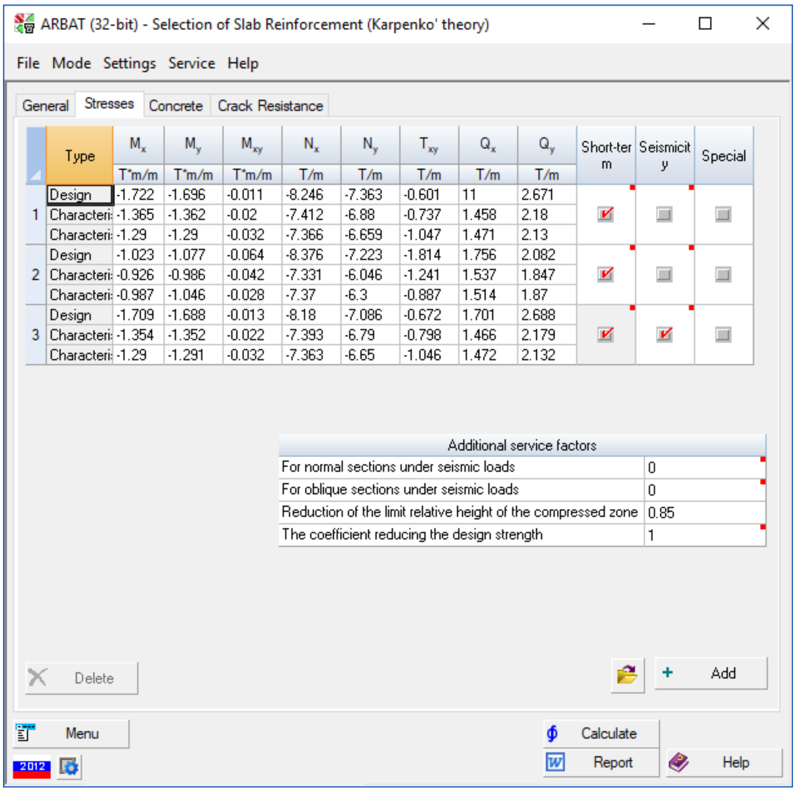


Figure 7.6.5-2. *The* **Stresses** *tab of the* **Selection of Slab Reinforcement (Karpenko' theory)** *dialog box*

Sign convention for the stresses presented on the Fig. 7.6.5-3.

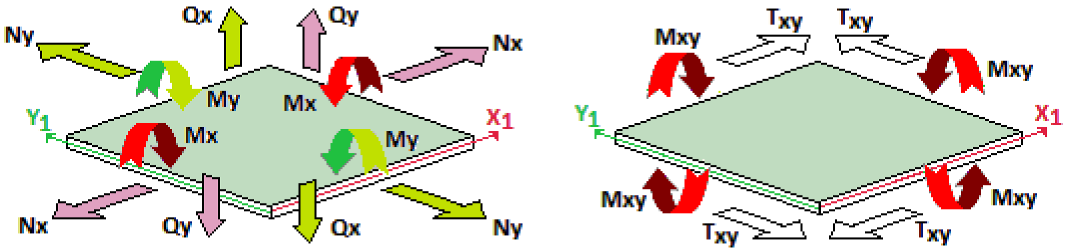


Figure 7.6.5-3. Sign convention

Concrete

Properties of concrete are specified in the **Concrete** tab in the same way as described in Section 7.3.2.

Crack Resistance

This tab appears when the **Analysis on crack resistance** checkbox is checked in the **General** tab. Data on crack resistance are specified according to the rules described in Section 7.3.4.

Additional Settings

Clicking the button invokes the **Additional Settings** dialog box (Fig. 7.6.5-4), which enables to define the rules for applying some guidelines of the codes, in particular, limiting the maximum distance between cracks (the ***Limit the distance between cracks to min(40 cm, 40∅)*** checkbox) and the rules for considering shear forces (the ***Use formula (8.106) from SP 63.13330*** checkbox). These rules are considered in more detail in Section 7.3.7.

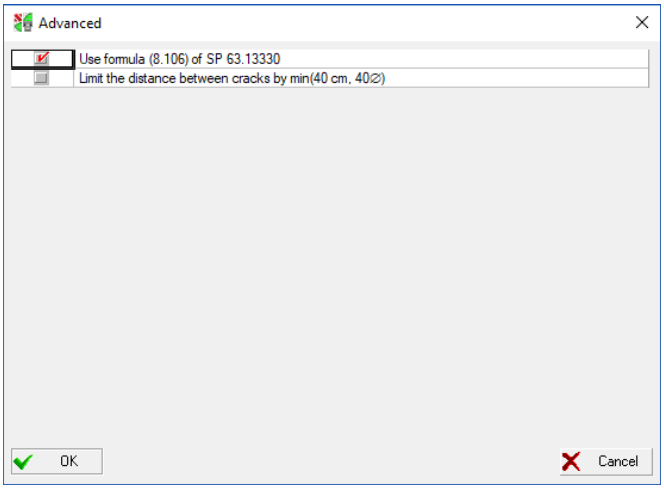


Figure 7.6.5-4. *The* **Additional Settings** *dialog box*

Results

The **Results** tab (Fig. 7.6.5-5) appears once the analysis is invoked by clicking the **Calculate** button. The **Selected reinforcement** table provides the data on diameter and spacing of longitudinal and transverse reinforcement per running meter, as well as the corresponding values of the area of reinforcement — *design* (without the assortment), *crack resistance* and *taking into account the assortment*.

The following values are checked during the reinforcement selection depending on the type of the member: strength of the plate section, strength for the shear forces Q_x, and Q_y , short-term and long-term crack opening width, etc.

Clicking the **Check** button invokes the **Check of Reinforcement of Two-dimensional Elements** mode, which enables to perform the checks of the selected reinforcement and plot the factors diagram.

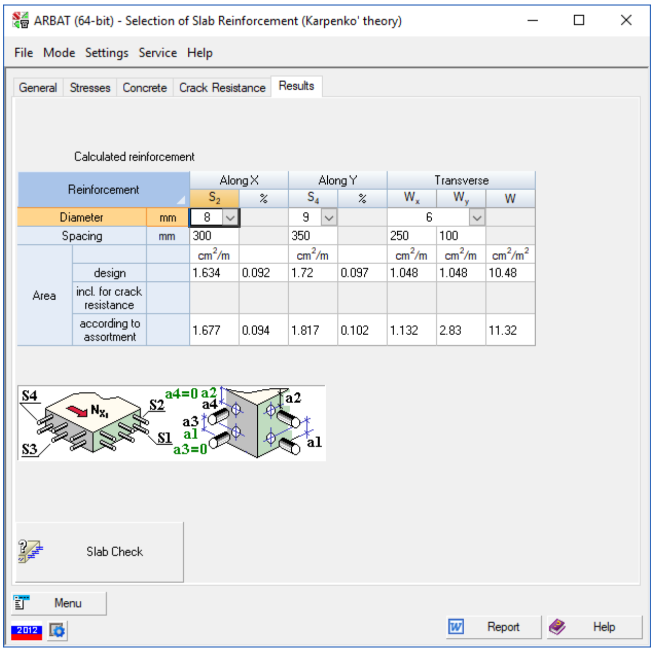


Figure 7.6.5-5. *The* **Results** *tab of the* **Selection of Slab Reinforcement (Karpenko' theory)** *dialog box*

7.7 Geometric Properties

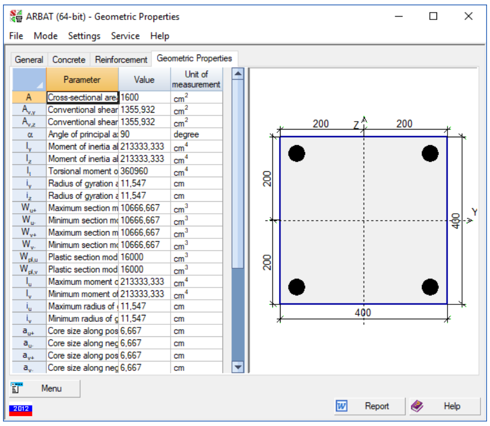


Figure 7.7-1. *The* **Geometric Properties** *dialog box*

This mode is used to calculate the geometric properties of concrete and reinforced concrete sections. The section itself and the data on concrete and reinforcement are specified according to the rules described above (see Section 7.3). Results are displayed in the **Geometric Properties** tab in the form shown in Fig. 7.7-1. They include the data on the area, moments of inertia of the concrete and reduced sections, and the percentage of reinforcement. These data can be used in a static or dynamic structural analysis. For example, if you specify the stiffness of a bar element in **SCAD** in the numerical parametric form and use the data obtained from the **Geometric Properties** mode, the effect of reinforcement will be taken into account.

7.8 Reinforcement Anchorage

This mode enables to perform the calculation of the anchorage (embedment) length of the reinforcement in concrete and the overlap length (lapped reinforcement) of rebars of the given diameter depending on the selected class and performance conditions of the reinforcement (in concrete in tension or in compression).

The effective length of the reinforcement anchorage can be adjusted by multiplying it by the ratio of the required reinforcement and the actual cross-sectional area.

The result of each selection of the anchorage or overlap length of the rebars can be saved in an initial data file with the **.sav** extension, and a report with the results of the calculation can be generated.

If the analysis is performed according to SNiP 2.03.01-84* (with Ukrainian modifications), the reinforcement anchorage and the overlap of rebars are calculated in accordance with the requirements of DSTU B V.2.6-156:2010.

The adhesion conditions, anchorage method (straight or bent bars), the presence of reinforcement transverse to the anchored one and many other factors are taken into account when calculating the anchorage and overlap length.

The anchorage length is calculated both for single and for double bars. The double bars are the bars with a clear distance less than the diameter of these bars.

The value of the compressive stress in concrete acting perpendicular to the anchored bar is determined by dividing the support reaction by the bearing area of the element.

The cross-sectional area of the transverse reinforcement along the effective length of the anchorage area is calculated by the method of successive approximations on the basis of the given spacing and diameter of transverse reinforcement.

7.9 Additional Information about ARBAT

7.9.1 On Seismic Actions

The concept of “seismic load” is absent from some of the **ARBAT** modes (*Beam Check*, *Column Check*, ...). This is due to the fact that special information on the logical relations between the loadings (which is actually realized in the DCF mode of **SCAD**) has to be specified to take seismicity into account correctly.

It should be noted that Section 2.14 of SNiP II-7-81*, Construction in Seismic Regions requires using an additional service factor which is greater than one (see Table 7 of SNiP II-7-81*). When a structure is being analyzed for a simultaneous action of multiple loads (which is nearly always the case for building structures), one of which is seismic, SNiP formally requires to use this factor even in cases when the fraction of the seismic action is small in comparison with other (long-term) loads. A requirement like this may lead to “dangerous results”.

The user can always take seismicity into account by, for example, specifying the service factor (according to Table 7 of SNiP II-7-81*) in the **Arbat** settings. Without these “auxiliary” operations the analysis will be conservative.

7.9.2 Factors in Stress-Strain Models

When comparing the results of the calculation for the limit forces and the nonlinear deformation model, the factors (utilization factors of restriction) can differ by many times. In order to explain the reason for this discrepancy, let’s consider the strength analysis of the normal cross-section of a bar under central tension.

When performing the calculation for limit forces, the following restriction must be satisfied (see Sec. 8.1.18 of SP 63.13330)

$$N \le R_s A_{s,tot},$$

where N is a longitudinal tensile force from external loads;

R_s is the design tensile strength of the reinforcement;

$A_{s,tot}$ is a cross-sectional area of all longitudinal reinforcement.

In this case, the respective factor is equal to

$$k_{force} = N / R_s A_{s,tot},$$

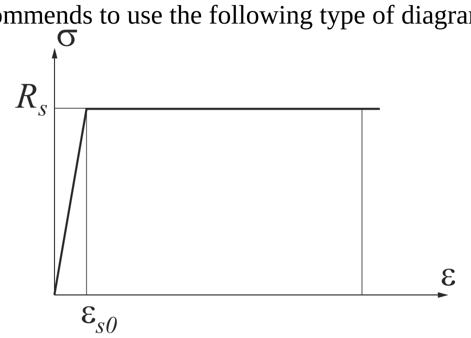
i.e. k_{force} depends linearly on N.

When using a nonlinear deformation model, the following restriction must be satisfied (see Sec. 8.1.24 of SP 63.13330)

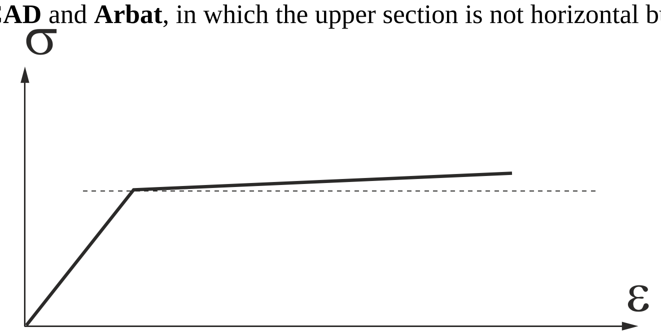
$$k_{def}=\varepsilon_{s,max}/\varepsilon_{s,ult}\leq 1,$$

where $\varepsilon_{s,max}$ is a relative deformation of a rebar under the greatest tension in the normal section of the element from the action of the external load;
$\varepsilon_{s,ult}$ is the limit value of the relative reinforcement elongation.

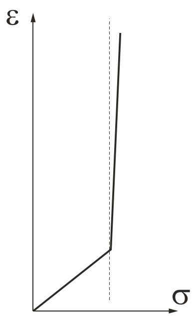
So, knowing the value of N (or the stress in the reinforcement $\sigma_s=N/A_{s,tot}$), it is necessary to determine the relative deformation of the reinforcement ε on the basis of the state diagram and compare it with the maximum allowable one ($\varepsilon_{s,ult}$). SP 63.13330 recommends to use the following type of diagram



If $N/A_{s,tot} < R_s$, then no problems arise, but even when $N/A_{s,tot} = R_s$, the diagram recommended by SP does not allow to determine ε unambiguously (ε can take any value greater than ε_{s0}) and obtain a factor allowing to make a conclusion about how much it is necessary to increase the load-bearing capacity of the section (or to reduce the value of the longitudinal force) in order to meet the requirements of the codes. Therefore, a modified diagram is used in the calculations in **SCAD** and **Arbat**, in which the upper section is not horizontal but slightly inclined:

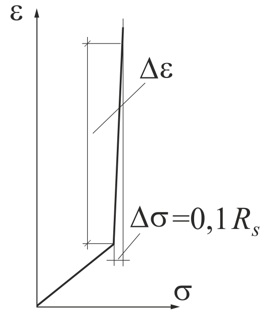


Let’s redraw this diagram as follows



Suppose that the longitudinal force exceeds the maximum allowable value given in Sec. 8.1.18 by 10%, i.e. k_{force} = 1,1.

If we use a nonlinear deformation model, we will get the following picture,



i.e., the relative deformation has increased significantly (much more than 10%) and we get the factor k_{def} >> 1,1.

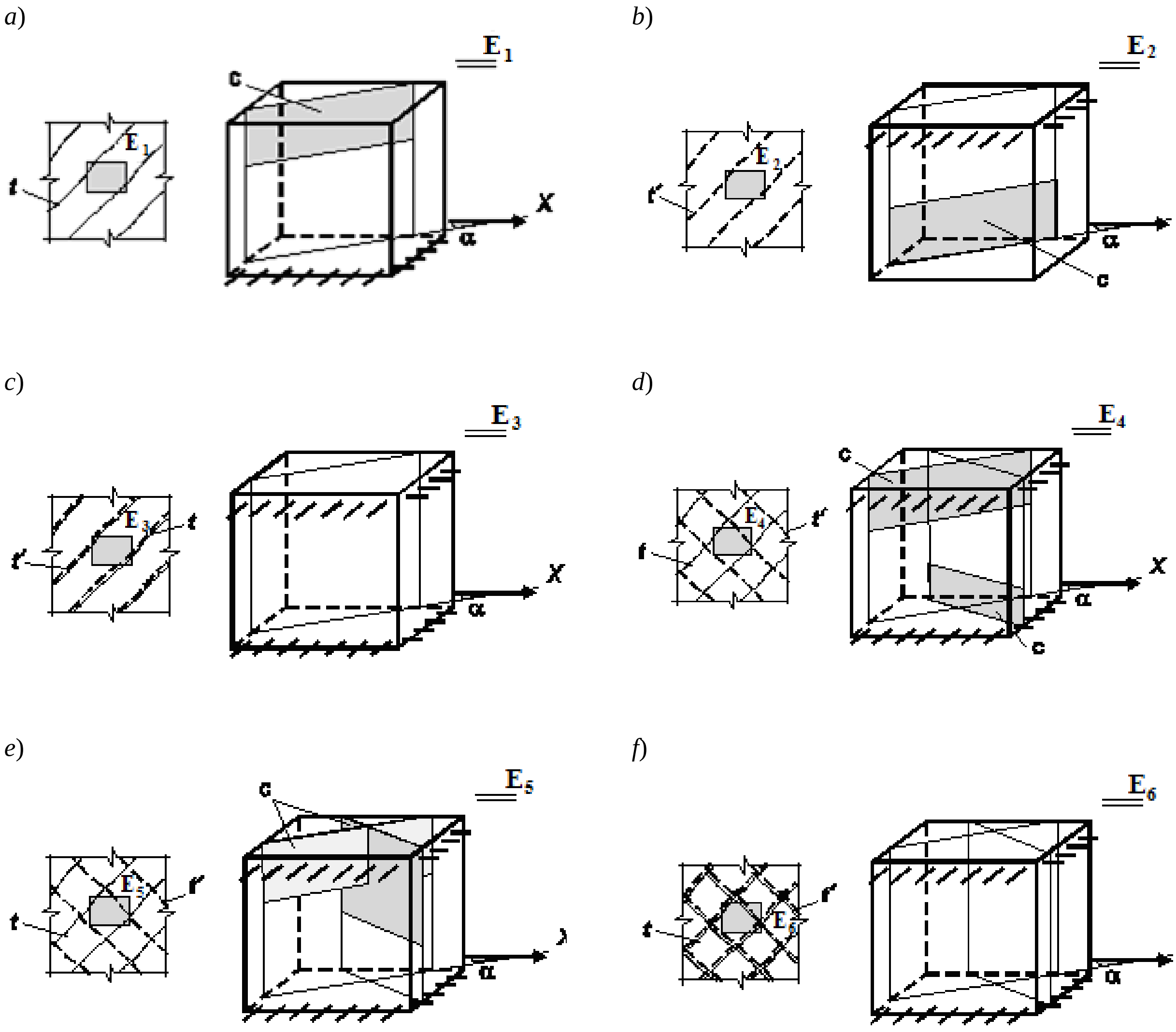
Let us consider another example that shows significant difference between k_{def} and k_{force} when the normal section strength requirements of SP are satisfied. Considering an element under axial tension with A500 reinforcement again, we assume that the longitudinal force is such that the stress of 0.99R_s occurs in the reinforcement. It is obvious that in this case k_{force}=0.99. And the reinforcement strain is equal to 0.99ε_{s0}, where (according to Sec. 6.2.11 of SP 63.13330) $\varepsilon_{s0}=R_s/E_s$=0.002175. That is $k_{def}=0.99\varepsilon_{s0}/\varepsilon_{s,ult}$=0.8613.

A similar situation can arise when comparing other factors.

7.9.3 Basic Principles of the Selection of Reinforcement in the Finite Elements of a Slab and Shell

Selection of the Type of Reinforcement

Following [14], first of all consider the conditions of the possible formation of cracks in the slab, which the need for reinforcement. The analysis is performed on an example of studying the behavior of some small in plan characteristic elements with cracks extracted from the structure. Since there are no fundamental differences between the performance of a small element of the slab and that of the shell (the curvature of the shell has no effect within the small area), we will consider flat plates taking both bending and membrane groups of forces, i.e. the combined action of all three moments (bending and torque) and three forces (normal and shear) on the elements is considered.



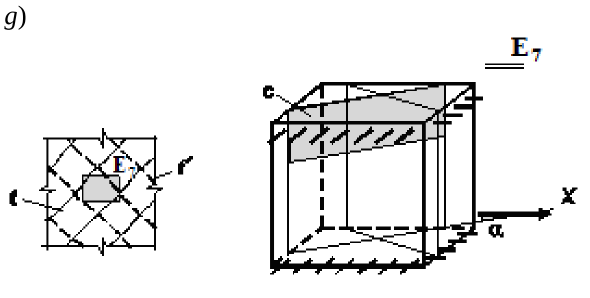


Figure 7.9.3-1. *Slabs a, b, c, d, e, f, g with cracks:*
t — crack on the lower surface;
t' — crack on the upper surface;
c —compressed area;
E_1, E_2, E_3, E_4, E_5, E_6, E_7 — elements of the slab with cracks of the respective type

The considered elements are assumed to be such that we can neglect the effect of shear forces on their deformability and strength both before and after the appearance of cracks. Due to this hypothesis the considered slabs belong to the category of thin slabs. If it has been somehow established that it is possible to neglect the action of shear forces, the obtained conclusions can be applied to plates of medium thickness as well.

The character of deformation of reinforced concrete spatial structures in areas with cracks depends on the arrangement of cracks, i.e. orientation of cracks with respect to the directions of reinforcement, mutual intersection of cracks, formation of cracks on one or both surfaces of the element, whether the cracks are through or non-through. Through cracks occur in a membrane stress state (in elements in the plane stress-strain state such as deep beams) or under the additional action of small moments. It is assumed that cracks are normal to the middle surface of elements.

The variety of cracks in elements can be reduced to seven arrangements (Fig. 7.9.3-1):

1. Nonintersecting cracks are located on one of the surfaces: lower (Fig. 7.9.3-1, *a*) or upper (Fig. 7.9.3-1, *b*); the elements have a tension area with cracks and a compressed area without cracks (elements E_1 and E_2).
2. The element is laced with nonintersecting through cracks (Fig. 7.9.3-1, *c*).
3. There are cracks in different directions both on the lower and on the upper surfaces of the element (Fig. 7.9.3-1, *d*). They often meet in the corner areas of slabs with large torques. The upper and lower areas have concrete strips between the cracks taking the compression forces (they are a kind of “compressed areas” laced with cracks).
4. One of the surfaces of the element — lower or upper (Fig. 7.9.3-1, *e*) has intersecting cracks in two directions.
5. The element is laced with intersecting through cracks (Fig. 7.9.3-1, *f*).
6. One of the surfaces has nonintersecting cracks, and the other one has intersecting cracks (Fig. 7.9.3-1, *g*).

All arrangements can be divided in two large groups: group *N* — nonintersecting cracks (see Fig. 7.9.3-1) (arrangements *a*, *b*, *c*), and group *I* — intersecting cracks (arrangements *d*, *e*, *f*, *g*).

Conditions of crack formation for elements subjected to the combined action of moments and normal forces can be formulated as a generalization of the theory of core moments of the crack formation of bar systems. The suggestion comes down to applying the theory of core moments of Gvozdev A.A. and Dmitriev S.A. [1] to more complex concrete structures of slabs and shells.

Two surfaces in the element at the distance of $\pm r_я$ from the middle surface are called the upper and lower surfaces of core points respectively. The value $r_я = h/6$ is determined as for a strip of slab by the formula of the strength of materials.

Let’s introduce two new tensors of core moments: one with component M_{cx}, M_{cy} and M_{cxy}, where

$$M_{cx} = M_x + N_x r_c; \quad M_{cy} = M_y + N_y r_c; \quad M_{cxy} = M_{xy} + N_{xy} r_c, \qquad (7.9.3.1)$$

and the other with components

$$M'_{cx} = - M_x + N_x r_c; \quad M'_{cy} = - M_y + N_y r_c; \quad M'_{cxy} = - M_{xy} + N_{xy} r_c. \qquad (7.9.3.2)$$

From the very definition of core sizes which are calculated based on the condition of zero stresses on the edge of a cross-section [14], it follows that the appearance of the positive core moment means that there is tension on the respective edge. If we assume that tension has to be taken by reinforcement, the decision on whether the reinforcement is necessary is made based on the analysis of the following inequalities:

for the lower area of the element:

$$M_{c.max} \le 0; \qquad M_{c.min} \le 0, \qquad (7.9.3.3)$$

for the upper area of the element:

$$M'_{c.max} \le 0; \qquad M'_{c.min} \le 0. \qquad (7.9.3.4)$$

If the first condition is violated (7.9.3.3), a crack is formed according to the arrangement *a* (Fig. 7.9.3-1) on the area subjected to the main core moment $M_{c.max}$. The angle α between this area (crack) and the X axis at the moment of violation of the first inequality (7.9.3.3) can be determined using the following expression

$$\mathrm{tg}\alpha = (M_{c.max} - M_{cy}) / M_{cxy}. \qquad (7.9.3.5)$$

When the first inequality is violated (7.9.3.4), the cracks are formed according to the arrangement *b* (Fig. 7.9.3-1). The angle between the upper crack and the X axis can be determined using the following expression

$$tg\alpha = (M'_{c.max} - M'_{cy}) / M'_{cxy}. \quad (7.9.3.6)$$

Simultaneous violation of the first condition (7.9.3.3) or the first condition (7.9.3.4) and of the additional inequality $N_{cl} \le 0{,}75 R_p h$ indicates that cracks are formed according to the arrangement *c* (Fig. 7.9.3-1). Additional inequality is established on the basis of the assumption that the through cracks are formed at the moment when the cross section has an unambiguous trapezoidal diagram of tensile normal stresses equal on one of the extreme and middle surface to R_p, and on the other extreme surface — to zero.

Simultaneous violation of the first inequality (7.9.3.3) and of the first inequality (7.9.3.4) indicates that cracks are formed according to the arrangement *d* (Fig. 7.9.3-1).

Violation of two inequalities (7.9.3.3) indicates that cracks are formed in the lower area according to the arrangement *e* (Fig. 7.9.3-1), and violation of two inequalities (7.9.3.4) indicates that cracks are formed in the upper area according to the arrangement *e*.

It is assumed that simultaneous violation of conditions (7.9.3.3) and of inequalities $N_{cl} \le 0{,}75 R_p h$ and $N_{c2} \le 0{,}75 R_p h$ or (7.9.3.4) and inequalities $N'_{cl} \le 0{,}75 R_p h$ and $N'_{c2} \le 0{,}75 R_p h$ leads to the formation of cracks according to the arrangement *f* (Fig. 7.9.3-1).

Here N_{cl}, N_{c2}, N'_{cl}, N'_{c2} are the values of normal stresses on the respective areas subjected to the core moments.

Simultaneous violation of the condition (7.9.3.3) and of only one of the inequalities $N_{cl} \le 0{,}75 R_p h$ and $N_{c2} \le 0{,}75 R_p h$ or of the conditions (7.9.3.4) and of only one of the inequalities $N'_{cl} \le 0{,}75 R_p h$ and $N'_{c2} \le 0{,}75 R_p h$ indicates that cracks are formed according to the arrangement *g* (Fig. 7.9.3-1).

Strength Check

It is known that the angle of principal planes may not coincide with the direction of crack propagation [14]. Therefore, when checking the reinforcement **ARBAT** also performs the search of areas where the conditions of strength may be violated the most. Suppose, for example, this area is located at an angle α to the X axis of reinforcement and it is assumed that the reinforcement is parallel to the X and Y axes of the Cartesian coordinate system. We will consider only the case when there is a non-through crack in the concrete and only the lower reinforcement is required (Fig. 7.9.3-2).

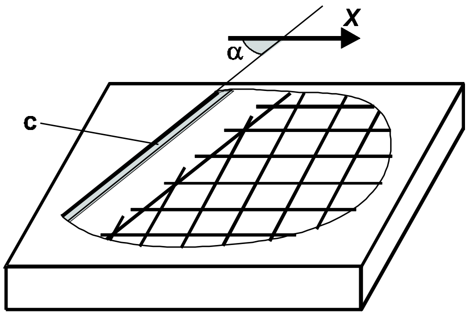


Figure 7.9.3-2. *To the strength analysis*

If we extract an elementary triangular prism (Fig. 7.9.3-3) and assume that the reinforcement transferring the forces in a crack takes only the longitudinal forces (neglect the dowel action), then when writing the equilibrium conditions with respect to axes 1 and 2 passing through the center of the compressed area of concrete parallel to axes X and Y, we will obtain:

$$R_a F_{a,x} Z \sin\alpha = (M_x + N_x Z_b) \sin\alpha + (M_{yx} + N_{yx} Z_b) \cos\alpha;$$
$$R_a F_{a,y} Z \cos\alpha = (M_y + N_y Z_b) \cos\alpha + (M_{xy} + N_{xy} Z_b) \sin\alpha, \quad (7.9.3.7)$$

where Z is an arm of the tensile reinforcement with respect to the center of gravity of the compressed area of concrete, $F_{a,x}$ and $F_{a,y}$ are reinforcement areas per running meter in the X and Y direction respectively. Since the prism has a unit length of the oblique plane and all internal forces are given by their values per unit length, the equations of equilibrium will include the lengths of planes 1×sinα and 1×cosα the total force is taken from.

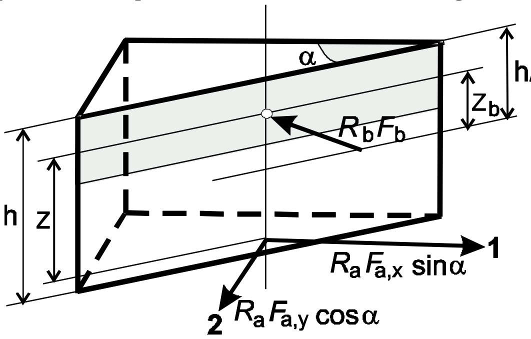


Figure 7.9.3-3. *To the determination of stresses in reinforcement and concrete in the area of an oblique crack:*

arrangement of forces in the area of the crack

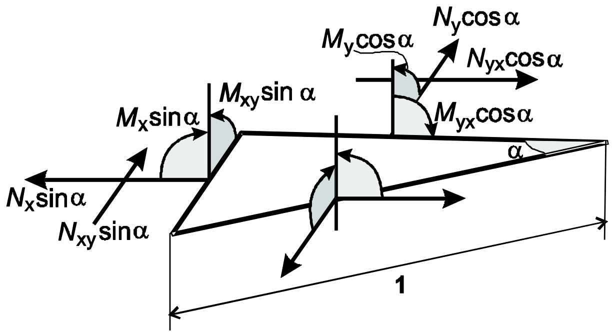


Figure 7.9.3-4. *To the determination of stresses in reinforcement and concrete in the area of an oblique crack:*

arrangement of moments and longitudinal forces applied to the sides of the elementary triangular prism

It is not difficult to adjust these equations for the case of compressed reinforcement. Then for example for the X axis we will obtain the following formula

$$M_{x1} + \sigma' A'_{sx1} Z' - M_{bx1} \leq 0, \qquad (7.9.3.8)$$

where M_{x1} is the projection of moments of all forces on the plane perpendicular to the X axis;

$$M_{x1} = (M_x + N_x Z_b) + (M_{yx} + N_{yx} Z_b)\text{ctg}\alpha\ , \qquad (7.9.3.9)$$

$\sigma' A'_{sx1} Z'$ is the moment of forces in reinforcement located in the compressed area of concrete and working in compression together with concrete with respect to the axis of the tensile reinforcement; M_{bx1} is the moment of forces in the compressed area of concrete with respect to the axis of the tensile reinforcement. Naturally, if there is compressed reinforcement, the value of the compressed area of concrete is clarified (this is done in an iterative cycle) and the value of Z is adjusted accordingly.

The condition of strength for the Y axis is determined in a similar way (by a simple replacement of values with the subscript *x* with the corresponding values with the subscript *y*).

For other cases of crack formation and reinforcement arrangements all considerations are performed similarly to those given above. They are described in detail in [14] and are used in **ARBAT** when checking the elements of reinforced concrete structures and selecting reinforcement.

The aforementioned selection of the location of areas where the conditions of strength can be violated the most is implemented by the direct enumeration of angles α with a step of 7.5°. The performed comparative calculations have shown that the use of a smaller step of scanning does not lead to noticeable changes in the results, but significantly slows down the computation.

Analysis for Lateral Forces

When plates are calculated for the action of lateral forces, **ARBAT** performs two separate calculations – for the action of Qx and for the action of Qy. In each case a strip of unit width is “cut out” from the plate in the respective direction. Then this strip is calculated for the action of the lateral force as a flexural element according to the recommendations of the codes for bar elements.

If we analyze the method of calculating plates for the action of shear forces, it can be seen that the strength is determined by a single value of the intensity $W = \frac{a_w}{s_x s_y}$ (a_w is the cross-sectional area of one transverse rebar), i.e. the amount of transverse reinforcement per unit area.

The values of the diameter of reinforcement and the spacing do not matter. It is only important that the value of W is greater than a certain value that ensures that the inequality of the following type is satisfied $\frac{Q}{0.5 \cdot R_{bt} \cdot h_0 + W \cdot R_{sw} \cdot h_0} \leq 1.$

When determining the required transverse *assortment* reinforcement, the software selects the diameter (based on the set of diameters of the given class of reinforcement and the limitations on the maximum diameter) and two spacings (from the lists of allowable spacing), so that the value of W is greater than the *design* value and gives the minimum intensity (minimizes the steel consumption). The intensities per running meter in the directions W_x,W_y can be either less than the design values, or considerably greater.

Determination of the Crack Opening Width

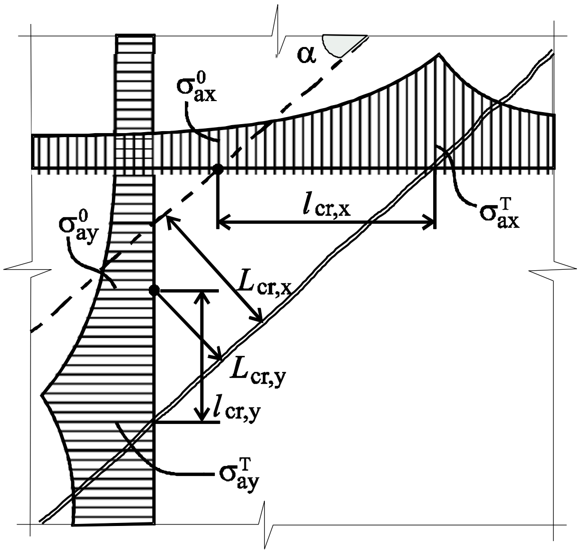


Figure 7.9.3-5. *Fragment of the surface of a plate with a crack*.
Diagrams of the variation of tensile stresses in reinforcement from the values σ^T_{ax} and σ^T_{ay} near the crack to the values σ^0_{ax} and σ^0_{ay} at the place of formation of a new crack

Crack opening calculation is performed in the same way as the strength analysis for one of the variants of their formation.

Two assumptions are used:

- the change in the stress state in the area between adjacent cracks can be neglected;
- adjacent cracks can be considered as parallel.

In accordance with the guidelines of V.I.Murashev [15], which are used as the basis for the current SNiP, it is assumed that the adjacent crack is formed at such a distance l_{cr}, that the stress in reinforcement due to the adhesion forces of concrete decrease from the value σ^T_a to σ^0_a, and the forces in concrete increase from zero near the crack to the values causing the formation of a new crack. The following values are predetermined:

$$L_{crc,x} = (bh^2/3,5)(d_x/4)\eta\ \varphi_l\ /\ (A_{sx}Z)\sin\alpha; \quad (7.9.3.10)$$

$$L_{crc,y} = (bh^2/3{,}5)(d_y/4)\eta\ \varphi_l / (A_{sy}Z)\cos\alpha\ ,$$

where d_x and d_y are rebar diameters;

η is the coefficient accounting for the profile of reinforcement;

φ_l is the coefficient accounting for the duration of the load action for different types of concrete.

The average distance between cracks L_{crc} is determined as the maximal of the two values (7.9.3.10), since this value defines the area of the plate where the adhesion between concrete and reinforcement is fully restored in two directions.

A value for the lower or upper reinforcement (depending on the situation) in the respective direction is used as the area of reinforcement per unit length A_{sx} and A_{sy}.

Stresses in reinforcement are determined by the formula:

$$\sigma_{sx} = M_x / (A_{sx}Z);\ \ \sigma_{sy} = M_y / (A_{sy}Z), \qquad (7.9.3.11)$$

where M_i is the projection of moments of all forces acting on the section with respect to the axis of the respective tensile reinforcement (i = x, y);

Z is an arm of the tensile reinforcement with respect to the center of gravity of the compressed area of concrete;

A_{sx} and A_{sy} is the area of tensile reinforcement per unit length.

The crack opening width is determined by the formula:

$$a_{crc} = (\varepsilon_{sx} + \varepsilon_{sy})L_{crc}\ , \qquad (7.9.3.12)$$

where $\varepsilon_{sx} = (\sigma_{sx} * \psi_s) / E_s$ and $\varepsilon_{sy} = (\sigma_{sy} * \psi_s) / E_s$. Here ψ_s is the coefficient accounting for the behavior of concrete in tension within the area with cracks and determined according to the recommendations of Sec. 4.29 SNiP.

Thus the crack opening width is determined by the average distance between them, and the stresses in reinforcement are determined without using the semi-empirical relationship given in SNiP for bar elements.

General Algorithm

In the general case it is necessary to determine the values for six types of reinforcement (Fig. 7.9.3-6).

A set of unfavorable combinations of loadings is predetermined. This set includes internal forces creating extreme values of normal stresses on the inner and outer surfaces, and also extreme values of shear stresses in the direction of the plate thickness. The values relating to the midpoints of finite elements are used.

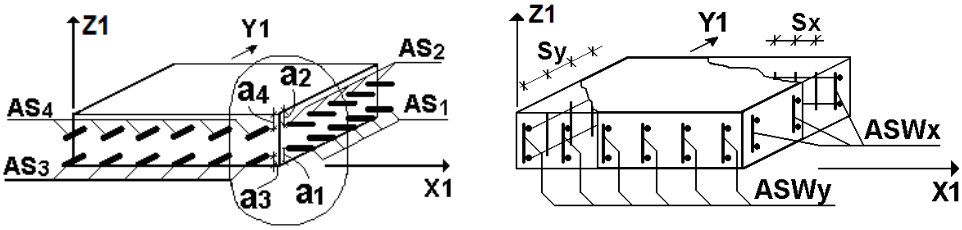


Figure 7.9.3-6. *Reinforcement layout in sections of a finite element of a slab and shell*

The strength of the plate is checked by the enumeration of variants of combinations for each finite element (core moments, angles α, the possible arrangement of cracks etc. are determined) and if the conditions of strength are not satisfied, the value of the intensity of reinforcement is increased. This is done until the conditions of strength are satisfied or it turns out that it is impossible to provide the conditions of strength without exceeding the specified maximum percentage of reinforcement, and the respective message is generated in this case.

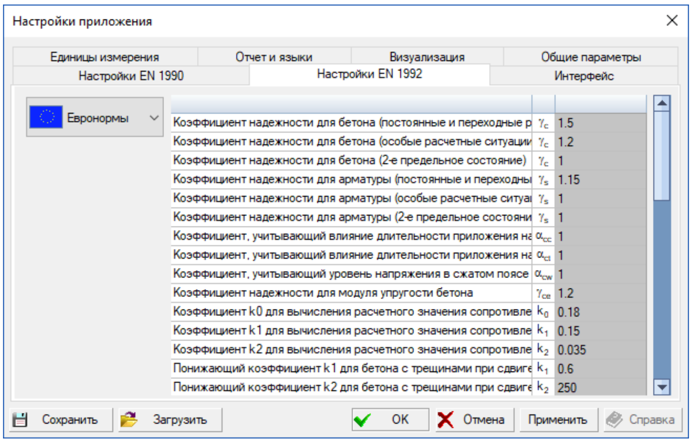
Then the crack opening width is checked for the selected reinforcement (if necessary). If the crack opening width exceeds the allowable value, the area of reinforcement is increased until the crack opening width becomes satisfactory. When selecting the longitudinal and transverse reinforcement from the action of each subsequent combination the reinforcement selected from the action of previous combinations is taken into account.

7.9.4 Peculiarities of the Analysis according to EN 1992-1-1

If the analysis is performed according to EN 1992-1-1, you have to specify the following data on concrete:

- Concrete class;
- Concrete aggregate;
- Age of concrete;
- Strength class of cement;
- Time of creep development;
- Temperature T during the creep development;
- Number of days where a temperature T prevails;
- Relative humidity;
- Additional service factor.

Various factors and partial safety factors are taken from the respective National Annexes to Eurocode, and can be selected in the **EN 1992 Settings** tab of the multi-tab **Settings** dialog box, where you can also modify some of these parameters.



Additional service factors for concrete and reinforcement enable to modify the design strength of materials and are used together with the factors provided in EN 1992-1-1, such as γ_s, α_{CC}, … . Design resistances are *divided* by these additional factors.

When the analysis is performed according to EN 1992-1-1, you do not have to specify the data on the random eccentricities and static definability. Geometric imperfections are taken into account according to Sec. 5.2(9). Second order effects are taken into account by the method based on nominal stiffness (Sec. 5.8.7). Formula (5.29) conservatively uses the value c_0=8, which is applicable both in the case of a uniaxial stress state and biaxial bending (see Note to Sec. 5.8.7.3(3)).

Creep is taken into account based on the recommendations of Annex B.

Strength analysis of normal sections is based on the concrete and reinforcement strain diagrams (Sec. 6.1). Bi-linear Prandtl diagrams are used (see Fig. 3.4, 3.8 of EN 1992-1-1).

Unlike SNiP, the user has to specify only one value in order to perform the crack resistance analysis — maximum crack width (Sec. 7.3.1(5)), which can be specified using, for example, Table 7.1N of EN 1992-1-1. Calculation of crack widths is provided in the case of single bending only, because EN 1992 considers only the case of a rectangular tension zones. For the same reason the calculation of the crack widths for circular sections is not implemented.

The analysis of plate members is based on the Karpenko theory using the properties of materials (concrete and reinforcement) provided in EN 1992-1-1. Shear analysis of slabs is performed according to the recommendations of EN 1992-1-1.

The cross-sections of bar structural members are checked for compliance with requirements of EN 1992-1-1 for the following factors:

Check	**Reference to EN 1992-1-1**
Ultimate moment strength	
Compressive strain in concrete	Sec. 6.1(3)P
Tensile strains in the reinforcing steel	Sec. 6.1(3)P
Axial force including second order effects	Sec. 5.8.7.3
Cracking stress	Sec. 7.1(2)
Crack width	Sec. 7.3.4
Shear resistance to Vz without shear reinforcement	Sec. 6.2.2
Shear resistance to Vy without shear reinforcement	Sec. 6.2.2
Shear resistance to Vz with shear reinforcement	Sec. 6.2.3
Shear resistance to Vy with shear reinforcement	Sec. 6.2.3
Design torsional resistance of a cross-section	Sec. 6.3.2 (4)
Torsional resistance of longitudinal reinforcement	Sec. 6.3.2 (4)
Torsional resistance of shear reinforcement	Sec. 6.3.2 (4)
Resistance of a member subjected to torsion and shear	Sec. 6.3.2(4)
Stress level in concrete	Sec. 7.2
Stress level in reinforcement	Sec. 7.2

Remark.The *ultimate moment strength* is a number which has to be divided by N, M_y, M_z, in order to obtain the maximum allowable values of the deformations of concrete or reinforcement.

The plate structural members are checked for the following factors:

Check	**Reference to EN 1992-1-1**
Strength of the slab section	
Shear resistance to Vx without shear reinforcement	Sec. 6.2.2
Shear resistance to Vy without shear reinforcement	Sec. 6.2.2
Shear resistance to Vx with shear reinforcement	Sec. 6.2.3
Shear resistance to Vy with shear reinforcement	Sec. 6.2.3
Possibility of cracking	Sec. 7.1(2)
Crack width	

7.9.5 Peculiarities of the Analysis according to DBN V.2.6-98:2009

If the analysis is performed according to DBN V.2.6-98:2009 (DSTU B V.2.6-156:2010), it should be noted that these design codes contain a huge number of errors and they are not consistent with other design codes of Ukraine, in particular, with DBN V.1.2-2:2006.

Since DBN V.2.6-98:2009 and DSTU B V.2.6-156:2010 are based on EN 1992-1-1, in case of conflict SCAD uses the corresponding provisions of EN 1992-1-1.

7.9.6 Analysis of Round Sections

Technically, the rules for the analysis of bar elements in shear provided in the design codes do not contain any restrictions on the shape of the cross section. However, using parameters *b* and h_0 for the analysis of round sections makes no sense.

The following technique is used to get around this problem. A round section with radius *R* is replaced by a square with an area equal to that of a circle with radius *R-a*, where *a* is the distance to the center of gravity of the reinforcement. The analysis of ring sections is performed in a similar way.

It should be noted that similar recommendations can be found in Sec. 3.3 of *Guidelines for the Design and Analysis of Bored Pile Retaining Walls*.

7.9.7 Design Codes the Requirements of Which are Implemented in Arbat

Mode	References to sections of codes and standards		
	SNiP 2.03.01-84*, KMK 2.03.01-96	**SNiP 52-01-2003 SP 52-101-2003**	**SP 63.13330**
Concrete Class	Table 12, 13 of SNiP 2.03.01-84*	Sec. 5.1.1–5.1.5, Table 5.1–5.3 of SP 51-101-2003 Sec. 5.1.1–5.1.3 of SNiP 52-01-2003	Table 6.7, 6.8
Concrete Grade	Table 11, 13 of SNiP II-21-75	Table 11, 13 of SNiP II-21-75	Table 11, 13 of SNiP II-21-75
Reinforce-ment	Table 19, 22 of SNiP 2.03.01-84*; GOST 5781-82	Sec. 5.3.1-5.3.3 of SNiP 52-01-2003 Sec. 5.2.1–5.2.3, Table 5.7, 5.8 of SP 51-101-2003	Table 6.13, 6.14, 6.15
Service Factor	Table 15–17 of SNiP 2.03.01-84*	Sec. 5.2.3, 5.4.3 of SNiP 52-01-2003 Sec. 5.1.9, 5.1.10, 5.2.6, 5.2.7 of SP 51-101-2003	Sec. 6.1.12, 6.2.9
Deflection Limits	Table 19, 21, 22 of SNiP 2.01.07-85*	Table 19, 21, 22 of SNiP 2.01.07-85*	Table F.1, F.3 of SP 20.13330.2011
Strength of RC 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, 5.3, 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, 8.2.2 of SP 52-101-2003; 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	Sec. 5.2.1, 5.2.2, 5.2.7-5.2.14, 6.1.1, 6.1.4, 6.1.10, 6.1.11, 6.1.15, 6.1.19-6.1.26, 6.2.8-6.2.15, 8.1.1-8.1.10, 8.1.14-8.1.30, 8.1.32-8.1.42,8.2.1-8.2.18, 10.2.2

Mode	References to sections of codes and standards		
	SNiP 2.03.01-84*, KMK 2.03.01-96	**SNiP 52-01-2003 SP 52-101-2003**	**SP 63.13330**
Strength of Concrete Sections	Sec. 1.21, 3.1–3.5, 3.6, 3.8, 3.30, 3.32 of SNiP 2.03.01-84*; Sec. 3.54, 4.4 of the Guide to SNiP 2.03.01-84* [7].	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, 8.2.2 of SP 52-101-2003; 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	Sec. 5.2.1-5.2.6, 7.1.1-7.1.12, 10.2.2
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 52-101-2003; 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	Sec. 8.2.21-8.2.32
Beam Check	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. 6.2.1, 6.2.2-6.2.14, 6.2.21–6.2.35, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.15 of SP 52-101-2003; 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	Sec. 8.1.8-8.1.13, 8.1.20-8.1.35

Mode	References to sections of codes and standards		
	SNiP 2.03.01-84*, KMK 2.03.01-96	**SNiP 52-01-2003 SP 52-101-2003**	**SP 63.13330**
Column Check	Sec. 3.10–3.12, 3.19–3.20, 3.24, 3.26, 3.28, 3.30, 3.32 of 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, 5.3, 4.11 of the Guide to SNiP 2.03.01-84 [7]	Sec. 4.2.6, 6.2.1–6.2.3, 6.2.15-6.2.20, 6.2.21–6.2.34, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.15, 8.2.2 of SP 52-101-2003; 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	Sec. 8.1.1-8.1.19, 8.1.20-8.1.35, 8.1.41
Slab Check	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 52-101-2003; Sec. 18, 19 of the Instruction Manual [42]	Sec. 8.1.53-8.1.59
Selection of Beam Reinforcement	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. 6.2.1, 6.2.2-6.2.14, 6.2.21 – 6.2.35, 7.1.2 – 7.1.3, 7.2.2 – 7.2.5, 7.2.11-7.2.15 of SP 52-101-2003; 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	Sec. 8.1.8-8.1.13, 8.1.20-8.1.35
Selection of Column Reinforcement	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*[7]	Sec. 4.2.6, 6.2.1–6.2.3, 6.2.15-6.2.20, 6.2.21–6.2.34, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.15, 8.2.2 of SP 52-101-2003; 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	Sec. 8.1.1-8.1.19, 8.1.20-8.1.35, 8.1.41

Mode	References to sections of codes and standards		
	SNiP 2.03.01-84*, KMK 2.03.01-96	**SNiP 52-01-2003 SP 52-101-2003**	**SP 63.13330**
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*[7]	Sec. 6.2.43–6.2.45 of SP 52-101-2003	Sec. 8.1.43-8.1.45
Punching	Sec. 3.42, 5.29 of SNiP 2.03.01-84*; Sec. 3.98 of the Manual [12]	Sec. 6.2.46–6.2.52 of SP 52-101-2003	Sec. 8.1.46-8.1.51
Tearing	Sec. 3.43 of SNiP 2.03.01-84*; Sec. 3.97 of the Guide to SNiP 2.03.01-84* [55]; Sec. 3.121 of the Guide to SNiP 2.03.01-84* [7]		
Inserts	Sec. 3.44–3.46 of SNiP 2.03.01-84*		Sec. 10.3.24 SP, Sec. 4.2-4.12 Recomendations
Short Cantilevers	Sec. 3.34 of SNiP 2.03.01-84*; Sec. 3.99 of the Guide to SNiP 2.03.01-84* [7]		Annex G
Keys	Sec. 3.115 of the Guide to SNiP 2.03.01-84* [7]		Annex F

7.10 References

[1] Gvozdev A.A., Analysis of bearing capacity of structures by limit state method – M.: Gosstroyizdat, 1949.

[2] Dubinsky A.M. Analysis of bearing capacity of reinforced concrete slabs – K.: Gosstroyizdat of the Ukrainian SSR, 1976 – 160 p.

[3] Kuznetsov V.S., Talyzova Yu.A. Analysis of Strength of Monolithic Beamless Floors Using the Limit Equilibrium Method // Vestnik MGSU, 2013, No.7 – p. 51-57.

[4] Rzhanitsyn A.R., Limit Equilibrium of Plates and Shells – M.: Science, 1983 – 288 p.

[5] Varvak P.M., Ryabov A.F. (1971) Handbook on the Theory of Elasticity (for structural engineers). Budivelnik. Kyiv – 418 p.

[6] Instruction manual for the analysis of statically indeterminate reinforced concrete structures with the consideration of the redistribution of forces / Res. Inst. for Reinforced Concrete.— Moscow, "Stroyizdat", 1961. 112 p.

[7] Guide on designing of concrete and reinforced concrete structures made of heavy-weight or lightweight concrete (no prestressing) (to SNiP 2.03.01-84*) / Central Res. Inst. of Industrial Buildings and Res. Inst. for Reinforced Concrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1986, 188 p.

[8] Guide on designing of residential buildings.— Issue 3. Designs of residential buildings (to SNiP 2.08.01-85) / Centr. Resrch. Inst. for Resident. Eng., State Committee for Architecture. Moscow, "Stroyizdat", 1989. 304 p.

[9] Recommended Practice for the Design of Steel Embedded Fittings for Reinforced Concrete Structures, Research Institute of Concrete and Reinforced Concrete, USSR State Committee for Construction.— Moscow, "Stroyizdat", 1984.— 87 p.

[10] Recommendations for the Calculation, Design, Manufacture and Installation of End-plate Joints of Steel Structures. – TsBNTI Minmontazhspetsstroy USSR, 1989. – 54 p.

[11] Manual on designing of concrete and reinforced concrete structures 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 Reinforced Concrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1978, 175 p.
[12] Manual on designing of concrete and reinforced concrete structures made of heavy-weight concrete (no prestressing) / Central Res. Inst. of Industrial Buildings and Res. Inst. for Reinforced Concrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1978, 321 p.
[13] Guide to design of masonry and reinforcement masonry structures (supplement to SNiP II-V.2-71). V.Kucherenko Centr. Res. Inst. for Structural Constructions, USSR State Comm. for Construction, Moscow, "Stroyizdat" Publ., 1974. — 183 p.
[14] N.I. Karpenko, The theory of deformation of reinforced concrete with cracks. — M.: Stroyizdat, 1976.
[15] V.I. Murashev, Crack resistance, rigidity and strength of reinforced concrete. — M.: Mashstroyizdat, 1950.

8. KAMIN

KAMIN enables you to perform structural analysis and various checks of members of masonry and reinforced masonry structures for compliance with requirements of SNiP II-22-81, KMK 2.03.07-98 or SP 15.13330.2012. The design forces are assumed to comply with loads defined by SNiP 2.01.07-85*. Rules for choosing design combinations of forces implemented in the software comply with the requirements of the same document as well. All modes of the application assume that the ***design values of loads*** are specified.

KAMIN is also based on documents related to SNiP II-22-81 and a previous edition of the design code, namely: Reference manual on design of masonry and reinforced masonry structures (supplement to SNiP II-22-81) [2], Guide to design of masonry and reinforced masonry structures (supplement to SNiP II-V.2-71) [4], Recommendations on reinforcing of masonry structural components [3], Designer’s reference manual on masonry and reinforced masonry structures [5], a book by P.F.Vakhnenko [1], SP 427.1325800.2018.

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

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

The check is performed both for undamaged structural members and for those with cracks in masonry and fire damages caused by high temperature.

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

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

In addition to these functions, **KAMIN** to a certain extent plays the role of a manual which can be used to check some factual data on the materials used, recommendations of SNiP II-22-81 and SP 15.13330, and estimates of the values and nature of the structural damages. Special reference modes are used for this purpose (see below). Some dialog boxes have the button clicking which displays additional reference information.

8.1 General

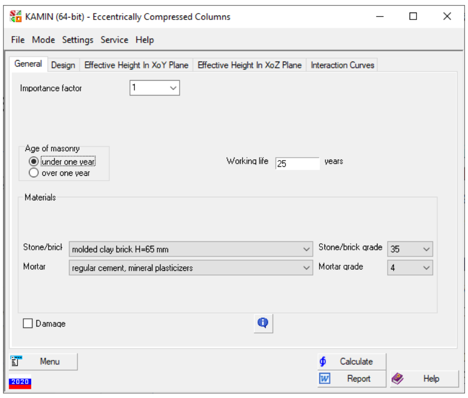


Figure 8.1-1. *The* **General** *tab*

All the modes of the application (except for the reference ones) require specifying a material the structural member is made of. This can be done by selecting a stone/brick type and grade, together with a mortar type and grade from the drop-down lists on the **General** tab (Fig. 8.1-1).

This tab is also used to specify the importance factor, working life of the structure, age of masonry (under one year or over one year), and, in some modes, the construction season (summer or winter).

When the analysis of masonry from hollow bricks is performed according to SP 15.13330.2012 you can additionally specify the void ratio according to Sec. 6.1 of the given design code.

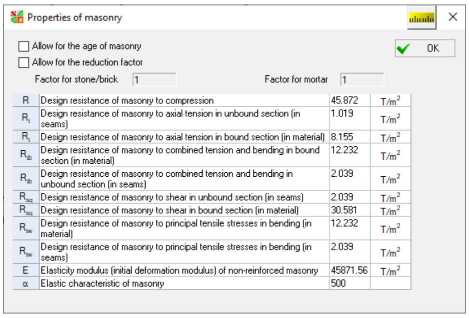


Figure 8.1-2. *The* **Design properties of masonry** *dialog box*

The button invokes a dialog box containing information on the design properties of the masonry used (Fig. 8.1-2).

This window enables to obtain the information on the service factors of masonry and to specify conditions for the calculation of the design strength using the respective checkboxes.

You should keep in mind that service factors are the reduction factors for the design strength of masonry.

The title bar of the dialog contains a button, , clicking which will open a dialog box for the customization of the units of measurement.

If the considered structure has damages, you have to check the respective checkbox on the **General** tab (see Fig. 8.1-1) to open the **Damage** tab where you can specify the relevant information.

8.2 Limitations

Each group of modes and individual modes have their limitations which are included in their descriptions. The following limitations are common for all modes:

- only solid masonry is considered;
- only finished masonry can be analyzed (fresh masonry or masonry that has not gained strength yet are not considered);
- top quality masonry (using leveling boards), for the design strength of which the reduction factors are not introduced, are not considered.

Limitations of bricks/stones — bricks/stones of the following types are not considered:

- stones/bricks with courses from 200 to 300 mm high, and between 150 and 200 mm;
- earth stones;
- adobe bricks.

Technology limitations — the following masonry is not considered:

- vibrated brickwork;
- frozen brickwork;
- rubble concrete.

There are mortar limitations: only heavy-weight mortars are used (cement, stiff, with organic plasticizers, with potash admix).

8.3 Materials

The application considers structural members made from the following materials:

SNiP, KMK, SP	**EN, DBN**
Stone/brick	
• molded clay brick, H=65 mm; • solid calcium silicate brick, H=88 mm; • solid calcium silicate brick, H=65 mm; • hollow calcium silicate brick, H = 88 mm; • calcium silicate block, H = 138 mm;	• Clay units • Calcium silicate units • Aggregate concrete units (Dense and light weight aggregates) • Autoclaved aerated concrete units

• ceramic block, $H \leq 150$ mm; • concrete brick, 200 mm $\leq H \leq$ 300 mm; • concrete bricks from pervious concrete and cellular concrete according to GOST 25485-82, 200 mm $\leq H \leq$ 300 mm; • concrete bricks from B type cellular concrete according to SNiP 2.03.01-84*, 200 mm $\leq H \leq$ 300 mm; • solid gypsum blocks, 200 mm $\leq H \leq$ 300 mm; • hollow gypsum blocks, 200 mm ≤ H ≤ 300 mm; • solid cinder blocks, 200 mm ≤ H ≤ 300 mm; • solid coal cinder blocks, 200 mm ≤ H ≤ 300 mm; • hollow cinder blocks, 200 mm ≤ H ≤ 300 mm; • hollow coal cinder blocks, 200 mm ≤ H ≤ 300 mm; • low-strength natural stones (sawn and finely cut), H ≤ 150 mm; • low-strength natural stones (semi-finely cut), H ≤ 150 mm; • low-strength natural stones (roughly cut), H ≤ 150 mm; • normal-strength natural stones (sawn and finely cut), 200 mm ≤ H ≤ 300 mm; • normal-strength natural stones (semi-finely cut), 200 mm ≤ H ≤ 300 mm; • normal-strength natural stones (roughly cut), 200 mm ≤ H ≤ 300 mm; • low-strength natural stones (sawn and finely cut), 200 mm ≤ H ≤ 300 mm; • low-strength natural stones (semi-finely cut), 200 mm ≤ H ≤ 300 mm; • low-strength natural stones (roughly cut), 200 mm ≤ H ≤ 300 mm; • flat rubble; • random rubble; • large concrete blocks, 500 mm ≤ H ≤ 1000 mm.	• Manufactured stone units • Natural stone units
Mortar	
• regular cement, mineral plasticizers; • stiff cement; • cement, organic plasticizers; • cement, potash added.	• General purpose mortar • Lightweight mortar of density from 600 to 800 kg/m3 • Lightweight mortar of density from 800 to 1300 kg/m3 • Thin layer mortar (bed joint from 0,5 to 3 mm)

8.4 Damage

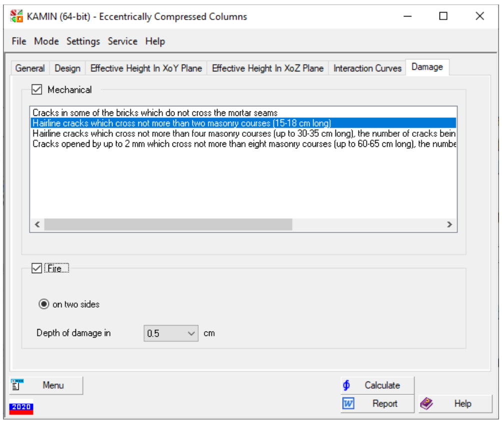


Figure 8.4-1. *The* **Damage** *tab*

Mechanical and fire damages can be taken into account when checking masonry and reinforced masonry structures in **KAMIN**. The damages are classified and specified in compliance with the Recommendations [3] and SP 427.1325800.2018. Information on the damages is specified in the respective tab (Fig. 8.4-1).

If there are mechanical damages, check the respective checkbox and select a damage type from the list. The following damage types are available:

- cracks in some of the bricks which do not cross the mortar seams;
- hairline cracks which cross not more than two masonry courses (15–18 cm long);
- hairline cracks which cross not more than four masonry courses (up to 30–35 cm), the number of cracks being not more than four per 1 m of width (thickness);
- cracks opened by up to 2 mm which cross not more than eight masonry courses (up to 60–65 cm long), the number of cracks being not more than four per 1 m of width (thickness);
- cracks opened by up to 2 mm which cross more than eight masonry courses (over 65 cm long);
- local (edge) damage of masonry up to 2 cm deep (fine cracks, spot peeling) and formation of vertical cracks at the ends of supports of beams, trusses, and lintels which cross not more than two courses of masonry (up to 15-18 cm long);
- local (edge) damage of masonry up to 2 cm deep (fine cracks, spot peeling) and formation of vertical cracks at the ends of supports of beams, trusses, and lintels which cross not more than four courses of masonry (up to 30-35 cm long);
- edge damage of masonry over 2 cm deep and formation of vertical and oblique cracks at the ends and under supports of beams and trusses which cross more than four courses of masonry (over 30 cm long).

If there are fire damages, check the respective checkbox and select a type and depth of the fire damage of the masonry.

The presence of the damage is accounted for by changing the geometric sizes of a structural member (in case of fire damages) and reducing the design strength of the masonry.

8.5 Eccentricity Calculation

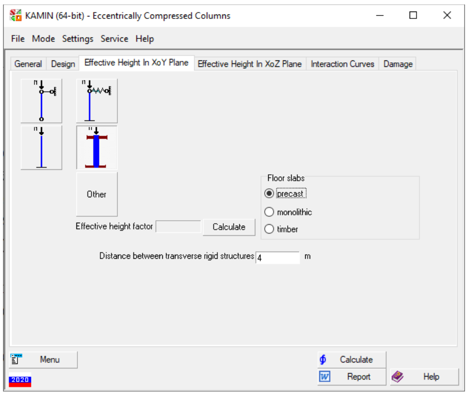


Figure 8.5-1. *Calculation of eccentrically compressed columns*

The calculation of eccentrically compressed elements of masonry structures is performed according to formula (7.4) of SP 15.13330.2020, which includes the eccentricity e_0 of the design force N with respect to the center of gravity of the section. This eccentricity is defined by the diagram of moments which appear in the masonry unit from the action of a concentrated moment M, applied in the upper section.

The program provides four variants of the design model (Fig. 8.5-1). It is necessary to know the effective height of the wall or the column l_0 (the effective length factor μ) and the moment diagram for each design model.

In three cases the design model is specified explicitly and it is very easy to determine the moment diagram. The respective solutions are shown in Fig. 8.5-2.

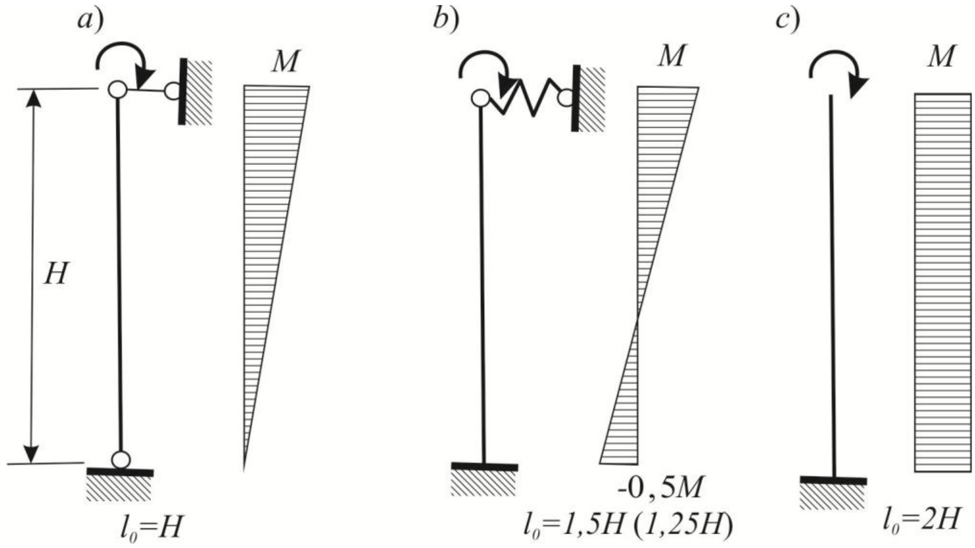


Figure 8.5-2

In the case "*b*" the effective length factor is taken as μ=1,5 for single-span buildings and as μ=1,25 for multi-span ones.

If the "Other" option is selected and the user has specified a certain value μ of the free length factor, the degree of restraint in the supporting sections has to be coordinated with the specified value μ. In order to do this let's consider a design model of the bar the ends of which are elastically clamped by springs with stiffness k (Fig. 8.5-3, *a*). The effective length factor of this bar is determined by the following formula (see SP 16.13330, p. 146 or DBN B.2.6-163: 2000, p. 182):

$$\mu = 0{,}5\sqrt{\frac{(\alpha+4{,}8)(\alpha+4{,}8)}{(\alpha+2{,}4)(\alpha+2{,}4)}}, \quad \alpha = \frac{kH}{EJ}.$$

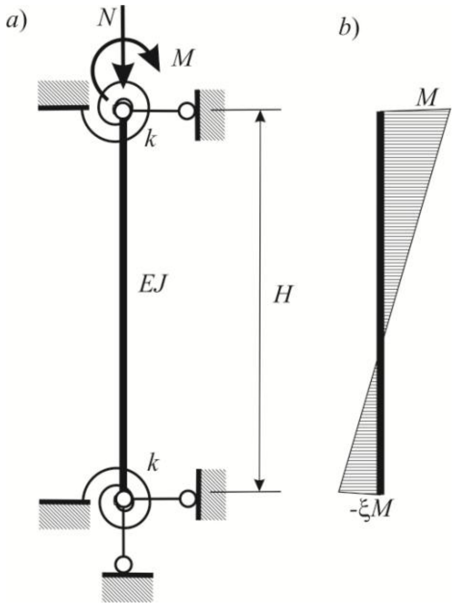


Figure 8.5-3

Therefore, if the value of the factor μ is specified, the clamping stiffness *k* is determined as:

$$2\mu = \frac{(\alpha + 4{,}8)}{(\alpha + 2{,}4)}; \quad 2\mu\alpha + 4{,}8\mu = \alpha + 4{,}8;$$

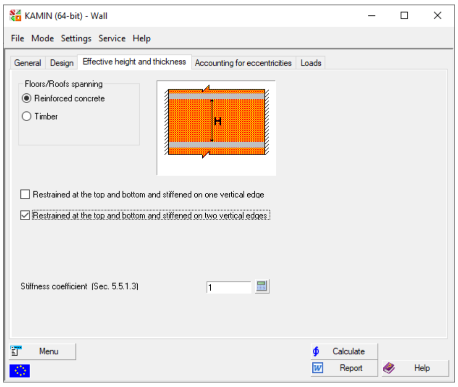
$$\alpha = \frac{4{,}8(1-\mu)}{2\mu - 1}; \quad k = \alpha\left(EJ/H\right) = \frac{4{,}8EJ\left(1-\mu\right)}{H\left(2\mu-1\right)}.$$

The moment diagram for this bar is characterized by the coefficient ξ (see Fig. 8.5-3.*b*) which is determined by the following formula:

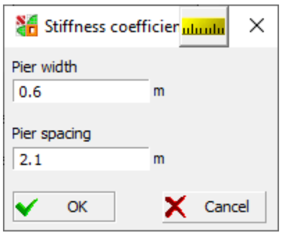
$$\xi = \frac{2EJ / H}{k + 4EJ / H} = \frac{2}{\alpha + 4} = \frac{(2\mu - 1)}{2{,}4(1-\mu) + 2(2\mu - 1)} = \frac{2\mu - 1}{1{,}6\mu + 0{,}4}.$$

8.6 Effective Height and Thickness (EN 1996, DBN)

When the analysis is performed according to EN 1996 or DBN, the effective height and thickness are specified in a separate tab (Fig. 8.6-1). Here you need to specify the type of floors/roofs (reinforced concrete or timber) and constraints along the vertical edges using the respective checkboxes.



Fugure 8.6-1. *Tab*
Effective Height and Thickness



Fugure. 8.6-2. Stiffness coefficient

It should be noted that the recommendations of EN 1996-1-1 do not cover all possible designs. In these cases, the calculation assumes an effective height factor of 1.0.

You also need to specify the stiffness coefficient (see Sec. 5.5.1.3 of EN 1996-1-1) in this tab, which is used in the calculation of the effective thickness of walls stiffened by piers. By default, this factor is 1.0 (which assumes no piers). Using the button , you can invoke the dialog box (Fig. 8.6-2) and specify the thickness of piers and the distance between them, and the program will calculate the stiffness coefficient based on Table 5.1 of EN 1996-1-1.

8.7 Accounting for Eccentricities (EN 1996, DBN)

Data on the floors and other walls adjacent to the considered wall are specified in a separate tab (Fig. **Error! Reference source not found.**-1).

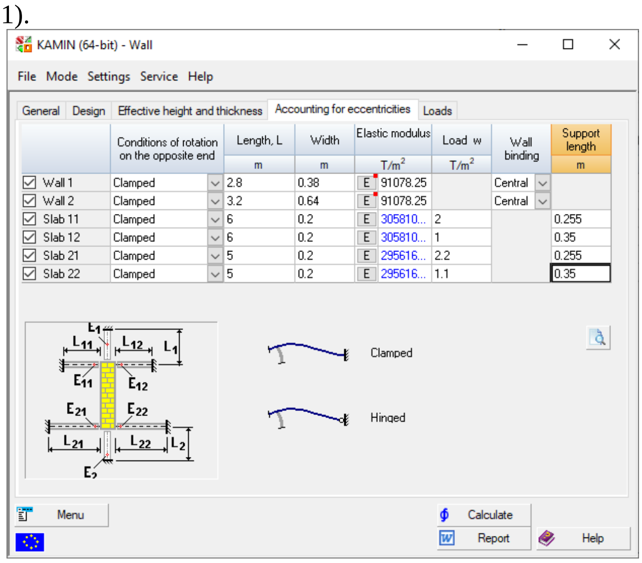


Figure. **Error! Reference source not found.**-1. 8.7 **Accounting for Eccentricities** *Tab*

In the general case there can be a junction of two walls (top and bottom) and four floors (right and left, top and bottom). The presence or absence of each of these elements is specified using the respective checkbox.

You have to specify the modulus of elasticity, length and thickness of the walls and floors. Clicking the button **E** in the table invokes a menu where you can select the concrete class for floors and the program automatically enters the value of the modulus of elasticity (information on the modulus of elasticity is not required for timber floors). For the upper or lower wall, when you click this button, the elastic modulus of the calculated wall will be assigned.

Moreover, you need to specify the type of the wall-to-wall junction (central, along the left edge, or along the right edge). The bearing length and the value of a uniformly distributed load are also specified for floors.

The program will automatically calculate the corresponding moments acting at the top and bottom of the analyzed wall and the eccentricity of the applied forces based on Annex C of EN 1996-1-1.

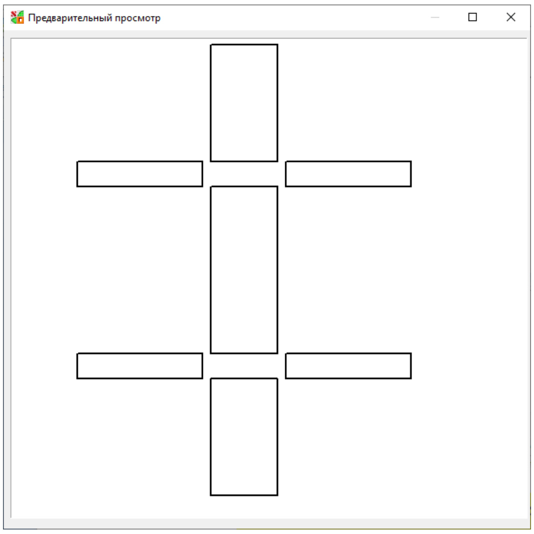
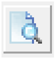


Figure. **Error! Reference source not found.**-2. Drawing of the structure

Clicking the **Preview** button  invokes a dialog box with a drawing of the structure and the values of the calculated moments (Fig. **Error! Reference source not found.**-2).

8.8 Main Window

Figure 8.8-1. *The main window*
a) EN 1996-1-1; *b)* DBN

When the application is started, the first thing that appears on the screen is its main window (Fig. 8.8-1 *a)* EN 1996-1-1*; b)*DBN) with a set of buttons for selecting a working mode. The modes can be classified into five groups four of which are used to perform various checks (**Masonry Structures**, **Reinforced Masonry Structures**, **Structures Under Reconstruction, Support Joints**), and the fifth one provides reference information (**Reference Information**). A detailed description of each mode is given below. Their brief characteristic is given here.

8.8.1 Masonry Structures

Modes of this group enable to perform checks of the following individual structural members of masonry structures:

- axially compressed columns;

- eccentrically compressed columns;
- external walls;
- basement walls;
- lintels.

Moreover, the checks of the local strength at the bearing areas of beams etc. are performed as well.

Each of the listed structural members is considered within one storey of the building.

The **Axially Compressed Columns** and **Eccentrically Compressed Columns** modes are used to check the strength and stability of separate or built-in columns compressed either axially or eccentrically.

The **External Wall** mode is used to check the strength and stability of building external walls, including those with openings.

The **Basement Wall** mode is used to check the strength and stability of a building basement wall.

The **Lintels** mode is used to check the strength and stability of lintels, including those with ties.

The **Local Strength** mode is used to check the local strength at the areas of bearing of beams, purlins, and other elements on walls and columns.

8.8.2 Reinforced Masonry Structures

Modes of this group enable to perform checks of the following individual structural members of reinforced masonry structures:

- axially compressed reinforced columns;
- eccentrically compressed reinforced columns;
- reinforced external walls of a building;
- reinforced basement walls.

The local strength of reinforced structural members can be checked as well.

Each of the listed structural members is considered within one storey of the building.

The **Axially Compressed Reinforced Columns** and **Eccentrically Compressed Reinforced Columns** modes are used to check the strength and stability of separate or built-in reinforced masonry columns compressed either axially or eccentrically.

The **Reinforced External Wall** mode is used to check the strength and stability of building reinforced masonry external walls, including those with openings.

The **Reinforced Basement Wall** mode is used to check the strength and stability of a building reinforced masonry basement wall.

The **Local Strength of Reinforced Structures** mode is used to check the local strength at the areas of bearing of beams, purlins, and other elements on reinforced masonry walls and columns.

8.8.3 Structures under Reconstruction

Modes of this group enable to perform checks of the following individual structural members of masonry structures including the damaged ones and reinforced by steel battens:

- axially compressed columns reinforced by battens;
- eccentrically compressed columns reinforced by battens;
- building walls reinforced by battens;
- openings in a wall.

The **Axially Compressed Columns Reinforced by Battens** and **Eccentrically Compressed Columns Reinforced by Battens** modes are used to check the strength and stability of separate or built-in axially or eccentrically compressed columns reinforced by steel battens.

The **Building Wall Reinforced by Battens** mode is used to check the strength and stability of a fragment of a masonry wall without openings reinforced by steel battens within the storey height.

The **Opening in a Wall** mode is used to check the strength and stability of masonry and (or) a steel lintel over an opening in the existing solid wall.

8.8.4 Support Joints

Modes of this group enable to perform a detailed check of the areas of beams, slabs, and trusses bearing on masonry (and reinforced masonry in some modes) walls and columns. Moreover, the strength of hanging walls is checked in the areas of the foundation beams bearing on fixed supports.

These modes are essentially used to check the local strength of the support joints, taking into account peculiarities of their designs and a static model of the structure.

The following members can be checked:

- hanging walls;
- bearing of beams and slabs on a wall;
- bearing of beams and/or trusses on piers and columns.

The **Hanging Walls** mode is used to check the local strength of a hanging wall bearing on a foundation beam at the areas adjacent to the supports of the foundation beam.

The **Bearing of Beams and Slabs on a Wall** mode is used to check the local strength of masonry at the areas of beams and slabs bearing on walls.

The **Bearing of Beams and/or Trusses on Piers and Columns** mode is used to check the local strength of masonry or reinforced masonry at the areas of beams and/or trusses bearing on piers and columns. The supported structure is a single beam/truss with a rectangular support base.

Reference Information contains **specific weights of masonry** of brick, natural or artificial stone with heavy-weight mortars, and a **damage classification** for masonry according to the Recommendations [3] and SP 427.1325800.2018.

8.9 Masonry Structures

8.9.1 Axially Compressed Columns

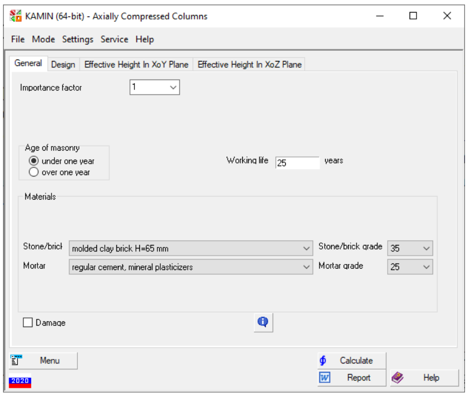


Figure 8.9.1-1. *The* **General** *tab*

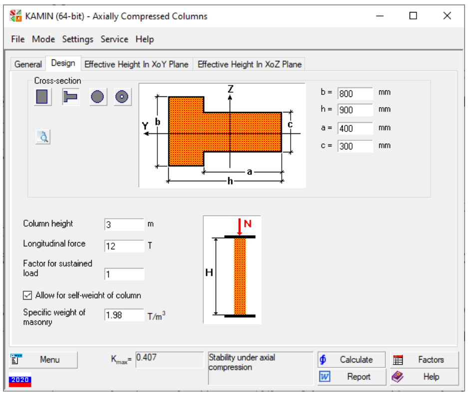


Figure 8.9.1-2. *The* **Design** *tab*

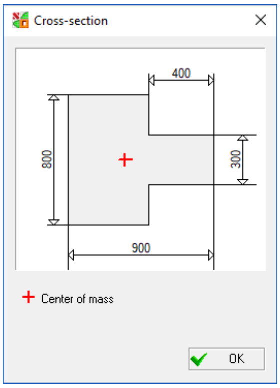


Figure 8.9.1-3. Cross-section

The mode is used to check the strength and stability of separate or built-in axially compressed columns. It implements requirements of Sec. 4.1-4.4 of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.1-7.4 SP 15.13330). The load-bearing capacity is checked in the most dangerous lower section. The columns are assumed to have a constant cross-section throughout their height. Rectangular, T-shaped, round, and ring cross-sections are available.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Effective Height In XoY/XoZ Plane**;
- **Damage**.

The **General** tab (Fig. 8.9.1-1) contains general data on the structure: the importance factor, age of masonry, working life, presence of damage, and data on the material — the types and grades of the stones/bricks and mortars used. All these data are described in the **Materials** section.

The **Design** tab (Fig. 8.9.1-2) contains data on the cross-sections of a column. Select one of the following cross-section types using the respective buttons: rectangular, T-section, round, and ring, and enter the sizes for the cross-section. You also have to specify the height of the column, the longitudinal force (applied in the upper section of the column) and the factor for sustained load.

To verify the sizes, click the button located under the cross-section selection buttons. The **Cross-section** dialog box (Fig. 8.9.1-3) will display the current cross-section with its sizes and the position of the center of mass.

It should be noted that the load is defined by the design value of the combination of loads and the factor for the sustained load in compliance with SNiP 2.01.07-85*.

It is possible to include the self-weight of the column in the check. To do it, check the respective checkbox and specify the specific weight of masonry. In this case the weight of the column will be added to the specified load.

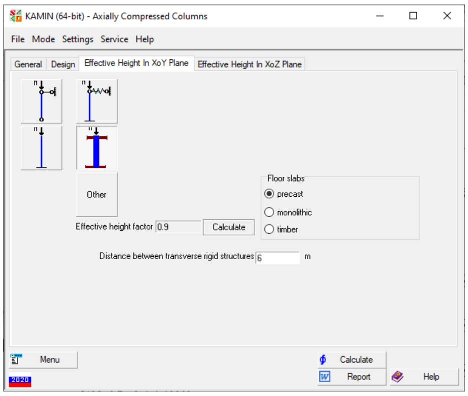


Figure 8.9.1-4. *The* **Effective Height In XoY Plane** *tab*

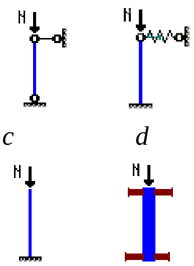


Figure 8.9.1-5. Boundary conditions of columns

The **Effective Height In XoY Plane** and **Effective Height In XoZ Plane** tabs are used to determine the effective height factors in both principal planes, XoY and XoZ (see Fig. 8.9.1-4).

The actual geometric height of a column should be multiplied by these factors calculated according to Sec. 4.3, 4.4, 6.7 of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.3, 7.4, 9.11 of SP 15.13330.2020). You can also enter the values of factors different from the design ones.

Different boundary conditions for columns are given in Fig. 8.9.1-5:

a) according to Sec. 4.3, *a* of SNiP II-22-81 (Sec. 7.3, *a* of SP 15.13330);

b) according to Sec. 4.3, *b* of SNiP II-22-81 (Sec. 7.3, *b* of SP 15.13330);

c) according to Sec. 4.3, *c* of SNiP II-22-81 (Sec. 7.3, *c* of SP 15.13330);

d) according to Sec. 4.3, *d* of SNiP II-22-81 (Sec. 7.3, *d* of SP 15.13330) and is used when an element is clamped in the floor slabs.

The cases *b*) and *d*) require the following additional information:

b) – type of the building (single-span or multi-span);

d) – type of the floor slabs (precast, monolithic, or timber) and the distance between transverse rigid structures.

Clicking the **Calculate** button will display values of the effective height factors.

The user can also enter his/her values for the effective height factor in the result fields by clicking the **Other** button.

The **Damage** tab provides data on fire and mechanical damages which can be taken into account when checking a structural member. The possible structural damages are described in Sec. 8.4.

The fire damages are taken into account along the whole perimeter of a column cross-section and are assumed to be uniform throughout the height of the column.

Mechanical damages of the masonry are also assumed uniform throughout the column height.

8.9.2 Eccentrically Compressed Columns

The mode is used to check the strength and stability of separate or built-in eccentrically compressed columns. The eccentric application of a load is assumed only in one principal plane of the column cross-section. The mode implements requirements of Sec. 4.7-4.9, 4.11, 5.3 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.7-7.9, 7.11, 8.3 of SP 15.13330). The columns are assumed to have a constant cross-section throughout their height. Rectangular, T-shaped, round, and ring cross-sections are available.

The column is checked for stability both in the moment (eccentricity) plane and out of the moment plane. The number of performed checks in the moment plane corresponds to the number of checked sections throughout the bar height where the moment diagram has the same sign. The check out of the moment plane is performed in the same way as for an axially compressed bar.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;

- **Effective Height In XoY/XoZ Plane**;
- **Interaction Curves**;
- **Damage.**

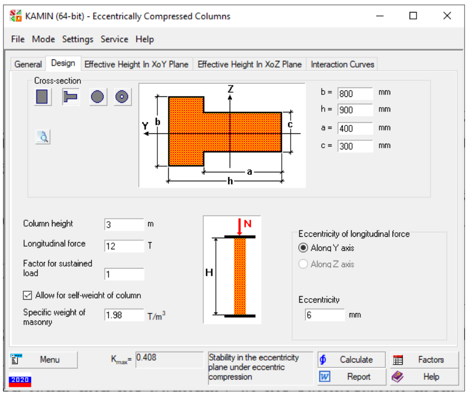


Figure 8.9.2-1. *The* **Design** *tab*

The **General** tab contains general data on the structure and is identical to the respective tab of the **Axially Compressed Columns** mode.

The **Design** tab (Fig. 8.9.2-1) is much similar to the respective tab of the **Axially Compressed Columns** mode. Additionally, you have to specify an axis along which the eccentricity is introduced and the value of the eccentricity.

The **Cross-section** dialog box (Fig. 8.9.2-2), which can be invoked by the button , displays the information on the location of the center of mass of the cross-section, the eccentricity of the applied load, and the boundary of the compression area.

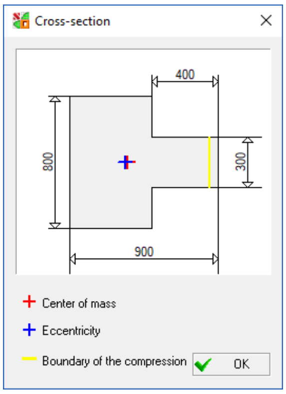


Figure 8.9.2-2. *Cross-section*

The **Effective Height In XoY Plane**, **Effective Height In XoZ Plane**, and **Damage** tabs are identical to the respective tabs of the **Axially Compressed Columns** mode.

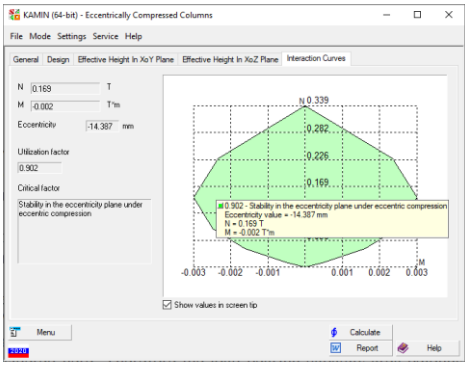


Figure 8.9.2-3. *The* **Interaction Curves** *tab*

The **Interaction Curves** tab (Fig. 8.9.2-3) enables to plot the curves enclosing an area of load-bearing capacity of the eccentrically compressed column under a longitudinal force and a moment.

If the **Allow for self-weight of column** checkbox is checked in the **General** tab, the self-weight will be allowed for in every point of the plotted load-bearing capacity area.

The load-bearing capacity area is plotted in *M-N* coordinates. Every point of this plane has a corresponding eccentricity equal to *M/N*. Unlike **Kristall** and **ARBAT** where the load-bearing capacity area surrounds the coordinate origin, only compressed columns are considered here. Therefore the load-bearing capacity area belongs to the upper half-plane ($N > 0$).

The curves (see Fig. 8.9.2-3) enclose an area inside which there are points with acceptable pairs of forces. It should be noted that a pair of forces is deemed acceptable when $K_{max} \leq 1$.

Using your mouse pointer, you can explore the area of the forces variation shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the longitudinal force and the moment (eccentricity); their values are displayed in the respective fields.

There are regions outside the load-bearing capacity area where the values of the factor K_{max} cannot be calculated. This is due to the fact that in those regions the eccentricity value is such that there is no compression area in the column cross-section.

The dialog also displays the maximum value of the utilization factor of restrictions that corresponds to these forces and the type of check in which it takes place. When the pointer is placed over a point where $K_{max} > 1$, a warning sign is displayed .

Clicking the right mouse button will display the full list of performed checks and values of the factors for the set of forces corresponding to the position of the pointer.

The **Damage** tab is identical to the respective tab of the **Axially Compressed Columns** mode. Only the internal side of the wall may have fire damages.

8.9.3 External Wall

This mode is used to check a longitudinal external wall of a building within a storey height. The wall may have openings. A wall fragment should be specified along its length in such a way so as to define the openings and piers. The length of the fragment (and further the length of the checked element) depends on the presence of openings. If there are none, one meter of the wall length is considered. The openings (as well as the piers) are assumed to be equal throughout the wall length. If there are openings, a fragment equal to the pier width is considered.

The check of stability under eccentric compression out of the wall plane is the main one. The accompanying checks are performed as well (tension and, if necessary, shear). The mode implements requirements of Sec. 4.7-4.9, 4.11, 5.3 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.7-7.9, 7.11, 8.3 of SP 15.13330). The wall within one storey is considered as a span of a continuous beam. The stability in the wall plane is assumed to be provided (even if there are openings), and is not checked. The piers are additionally checked for stability in the wall plane (according to Sec. 4.5 of SNiP II-22-81, Sec. 7.5 of SP 15.13330) as being axially compressed.

The check is performed for the following cross-sections:

- in the upper part of the wall directly under the floor slab;
- in the middle part of the wall;
- in the lower part of the wall, when it lies on a lower floor slab or on a foundation.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Loads**;
- **Damage** (for SNiP)**;**
- **Effective Height and Thickness** (for EN 1996-1-1 and DBN)**;**
- **Accounting for Eccentricities** (for EN 1996-1-1 and DBN).

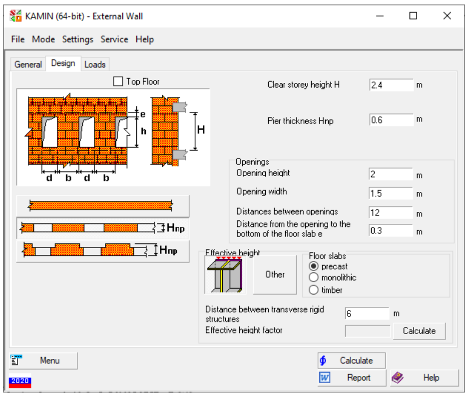
The **General** tab contains general data on the structure and is identical to the respective tab of the **Axially Compressed Columns** mode.

The **Design** tab (Fig. 8.9.3-1, *a*) contains information on the wall, its dimensions, and the dimensions of openings and cross-sections. Select one of the following cross-section types using the respective buttons: rectangular or T-section (tees are available only if there are openings).

The effective height (Fig. 8.9.3-1, *b*) is assigned in compliance with Sec. 4.3, *d* and Note 1 of SNiP II-22-81 (Sec. 7.3,*d* of SP 15.13330). The supporting sections of the wall are clamped.

The obtained stress reduction factors are assumed to be constant throughout the wall height.

a



b

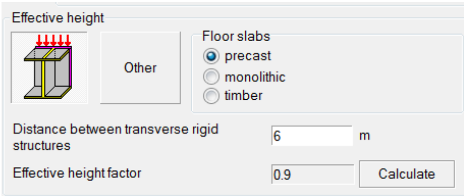


Figure 8.9.3-1. *The* **Design** *tab*

If the **Top Floor** checkbox is selected, then the calculation model of the wall is used in the form of a single-span beam, in which the lower support is rigid supported, and the upper one has a hinge, otherwise rigid support is assumed at both ends.

For EN 1996-1-1 and DBN data concerning effective height and effective thickness have been defined on the separate tab.

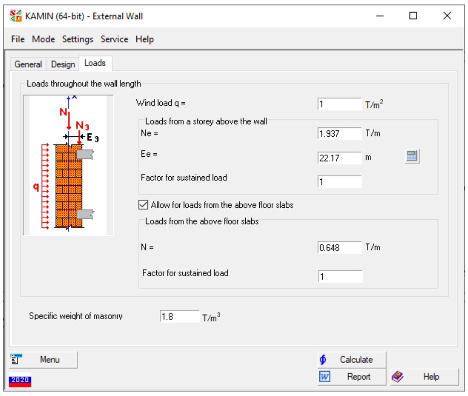


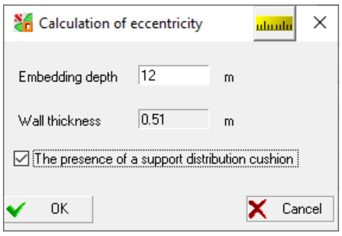
Figure 8.9.3-2. *The* **Loads** *tab*

The **Loads** tab. A wall is checked for the action of the following loads (Fig. 8.9.3-2):

- wind, normal to the wall surface;
- loads from the above stories axially applied to the top of the wall or of the pier;
- loads from a floor slab bearing directly on a wall applied eccentrically in the general case;
- dead weight of the wall.

All loads are assumed to be defined by their design values. The full values of loads and factors of sustained loads are specified for loads from the floor slabs.

The button  invokes a dialog box



where you can specify the slab embedment depth and whether there is a support distribution pad. The eccentricity of the load from the floor above the wall will be automatically calculated after clicking the **OK** button based on the recommendations of Sec. 6.10 of SNiP II-22-81 or Sec. B.1 of Annex B of SP 15.13330.2020.

The checks are performed for one principal combination of loads, the wind load being taken with the factor of 0.9 and other loads – with the factor of 1.

The dead weight of the wall is always taken into account.

The **Damage** tab is identical to the respective tab of the **Axially Compressed Columns** mode. Only the internal side of the wall may have fire damages.

8.9.4 Basement Wall

This mode is used to check a basement wall of a building within the height of the basement storey. It is assumed that the basement wall has no openings, is simply supported on a foundation or on a floor slab above the basement, and has a rectangular cross-section.

The check of stability under eccentric compression out of the wall plane is the main one. The accompanying checks are performed as well (tension and, if necessary, shear). The mode implements requirements of Sec. 4.1, 4.2, 4.7-4.9, 4.11, 5.3 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.1, 7.2, 7.7-7.9, 7.11, 8.3 of SP 15.13330). The stability in the wall plane is assumed to be provided and is not checked. A random eccentricity according to Sec. 6.65 of SNiP II-22-81 (Sec. 9.71 of SP 15.13330.2020) is taken into account only for loads from the upper wall.

The check is performed for the following cross-sections:

- in the upper part of the wall directly under the floor slab, treated as an eccentrically compressed element;
- in the middle part of the wall;
- in the lower part of the wall, when it lies on a foundation, treated as an axially compressed element.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design and Loads**;
- **Effective Height and Thickness** (for EN 1996-1-1 and DBN)**;**
- **Accounting for Eccentricities** (for EN 1996-1-1 and DBN);
- **Damage** (for SNiP).

The **General** tab contains general data on the structure and is identical to the respective tab of the **Axially Compressed Columns** mode.

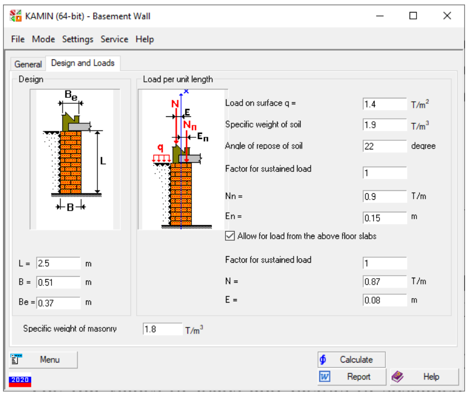


Figure 8.9.4-1. *The* **Design and Loads** *tab*

The **Design and Loads** tab (Fig. 8.9.4-1) contains information on the wall, its dimensions, and the dimensions of its cross-section. Segments of the wall are assumed to have the same footing depth.

The following loads are taken into account:

- the weight of the basement wall;
- loads from the above stories;
- loads from the floor slab over the basement;
- lateral earth pressure;
- loads on the soil near the basement (the design elevation is assumed to coincide with the top of the basement wall). The value of the load on the soil should be not less than 1000 kg/m^2.

All loads are assumed to be defined by their design values. The full values of loads and factors of sustained loads are specified for loads from the floor slab over the basement and from the above stories.

The **Damage** tab is identical to the respective tab of the **Axially Compressed Columns** mode. Only the internal side of the wall may have fire damages.

8.9.5 Lintels

This mode is used to perform the analysis of masonry (coursed, Dutch, arched) and reinforced concrete lintels. All lintels are assumed to have a rectangular cross-section. A lintel can cover a middle or an end span. Each masonry lintel can have a tie.

The check of stability under eccentric compression in the wall plane is the main one. The lintel is assumed to be partially clamped in the piers, and the effective length factor is taken as 2/3. The stability of the lintel out of the wall plane is assumed to be provided. The mode implements requirements of Sec. 4.7, 4.8, 6.47 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.7, 7.8, 9.53 of SP 15.13330.2020). In cases when a lintel covers an end opening and there is no tie, the end pier is additionally checked for shear (in mortar and in stone) caused by the thrust in the lintel (Sec. 4.20 of SNiP II-22-81, Sec. 7.20 of SP 15.13330.2012). Both the lintel and the pier are assumed to be made of masonry with the same properties. If there is a tie, its strength is checked according to Sec. 5.1 of SNiP II-23-81* (Sec. 7.1.1 of SP 16.13330).

The strength checks of a reinforced concrete lintel are performed according to the requirements of SNiP 2.03.01-84* or SP 63.13330.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Loads**;
- **Reinforcement Arrangement**.

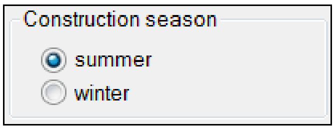


Figure 8.9.5-1. *The* **General** *tab* (a fragment)

The **General** tab contains general data on the structure and is almost identical to the respective tab of the **Axially Compressed Columns** mode.

There are the following differences (for SNiP):

- the user should specify the season of construction (Fig. 8.9.5-1);
- the list of bricks/stones does not contain low-strength natural stones and rubble;
- mortars of grade 25 and higher are used.

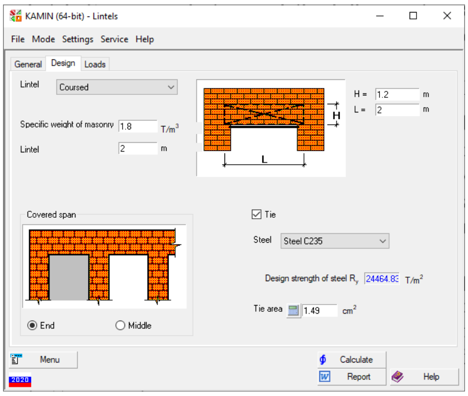


Figure 8.9.5-2. *The* **Design** *tab*

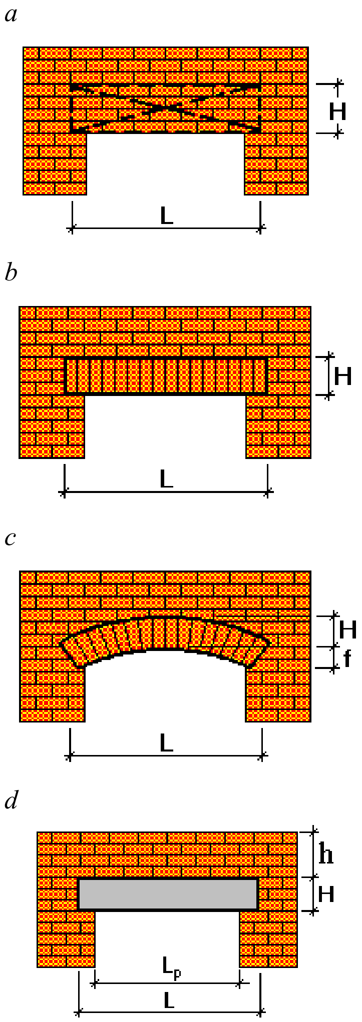


Figure 8.9.5-3. Lintel types

The **Design** tab (Fig. 8.9.5-2) is used to specify a design of the lintel. You can select one of the following types of lintels: coursed (Fig. 8.9.5-3, *a*), Dutch (Fig. 8.9.5-3, *b*), arched (Fig. 8.9.5-3, *c*), and reinforced concrete (Fig. 8.9.5-3, *d*) lintels.

Every masonry lintel may have a tie for which its steel grade and its cross-section area should be specified. The button enables to obtain the area of the tie in cases when the latter is made of rebars or rolled profiles. Clicking this button opens a dialog box shown in Fig. 8.9.5-4, where you have to select a rebar diameter or a rolled steel profile and their number. Clicking the **OK** button will transfer the value of the area to the respective tie area text field.

In the case when the lintel covers the end span and there is no tie, it is necessary to check the strength of the pier. Therefore, you have to specify the area of the pier and the value of the additional load which does not depend on the load transferred by the lintel (Fig. 8.9.5-2).

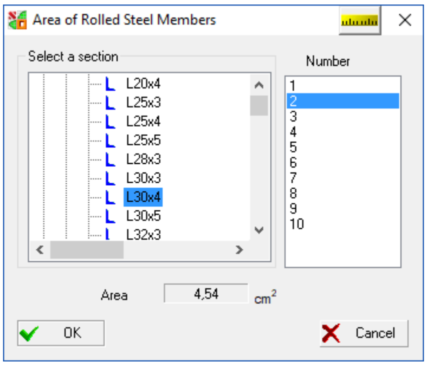


Figure 8.9.5-4. The area of the tie

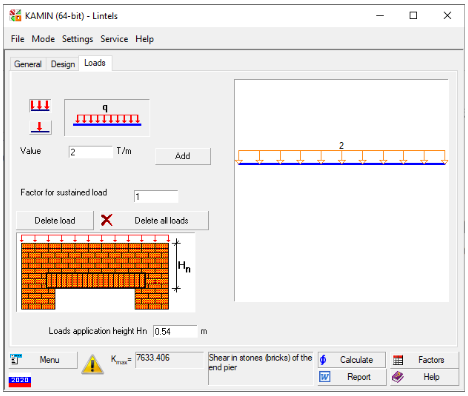


Figure 8.9.5-5. *The* **Loads** *tab*

The **Loads** tab is used to specify loads on the lintel transferred from the floor slab of the above storey (Fig. 8.9.5-5), the load application height, and the factor for sustained loading.

Loads of the following two types can be applied:

- uniformly distributed throughout the lintel span;
- a system of vertical concentrated forces, each one defined by its value, direction, and a coordinate with respect to the left support.

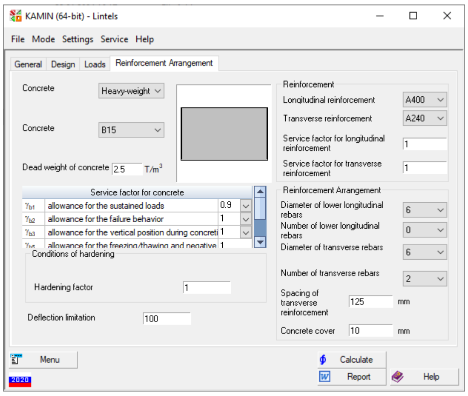


Figure 8.9.5-6. *The* **Reinforcement Arrangement** *tab*

If a reinforced concrete lintel has been selected, an additional tab, **Reinforcement Arrangement** (see Fig. 8.9.5-6), appears, where you have to select a type and grade of concrete from the respective lists. You also have to assign the service factors for concrete and the hardening factor.

The service factor for the concrete, γ_{b2}, allows for the duration of load, and is taken as 1 (by default) or 0.9 (item 2*a* in Table 15 of SNiP 2.03.01-84*, Sec. 6.1.12,*a* of SP 63.13330).

The service factor for concrete, γ_b, is a product of all service factors for concrete from Table 15 of SNiP 2.03.01-84* (Sec. 6.1.12 of SP 63.13330) except for γ_{b2}; and is taken as 1 by default.

If the value of the initial elasticity modulus of concrete is different from its tabular value, then a hardening factor should be specified, which will be used to adjust the modulus value (it should be specified only in case of the natural concrete hardening).

You also have to specify the following information on the reinforcement arrangement:

- the thickness of the concrete cover;
- classes of longitudinal and transverse reinforcement;
- service factors for the reinforcement;
- a diameter and number of the lower longitudinal rebars arranged in one row;
- a diameter and number of transverse rebars, and their spacing.

If the number of rebars or the spacing of the transverse reinforcement are set to zero, it is assumed that there is no transverse reinforcement, and the respective checks will not be performed.

The reinforced concrete lintels do not require the upper reinforcement, therefore the data on the upper reinforcement is not specified and the check of the upper reinforcement is not performed.

8.9.6 Local Strength

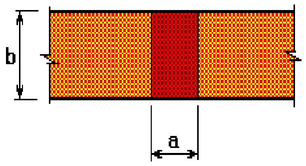
The **Local Strength** mode is used to perform checks of the local strength in places where concentrated loads are transferred (from the bearing of beams, purlins, and other members) onto walls and columns. The mode implements requirements of Sec. 4.13, 4.14, 4.16 of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.13, 7.14, 7.16 of SP 15.13330.2012). The simultaneous action of the local and main loads (Sec. 4.15 of SNiP II-22-81, Sec. 7.15 of SP 15.13330.2012) is not checked. The distribution of pressure in places where the loads are transferred is assumed to be uniform throughout the whole transfer area.

The dialog box of this mode contains the following tabs:

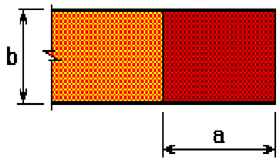
- **General**;
- **Load Arrangement**;
- **Damage** (for SNiP).

The **General** tab contains general data on the structure and is almost identical to the respective tab of the **Axially Compressed Columns** mode.

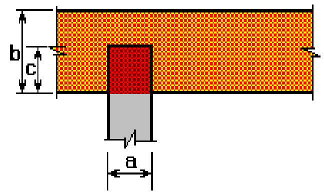
The **Load Arrangement** tab is used to specify the value of the local load and to select its arrangement. Fig. 8.9.6-1 shows the available load arrangements and indicates (in brackets) the respective designs from Fig. 9 of SNiP II-22-81 (SP 15.13330) (an area where the load is applied is highlighted).



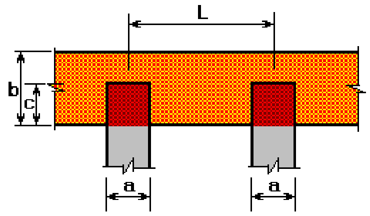
a) Local load throughout the width of an element (9, *a*)



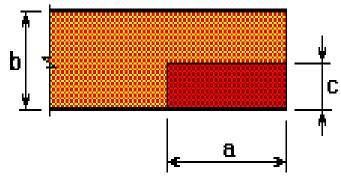
b) Local edge load throughout the width of an element (9, *b*)



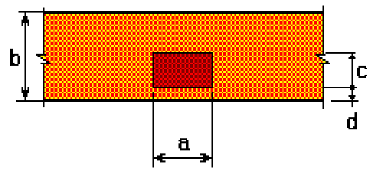
c) Local load at the end supports of beams and purlins (9, *c*)



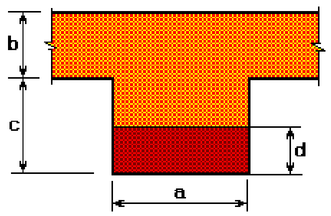
d) Local load at the end supports of beams and purlins (9, c_1)



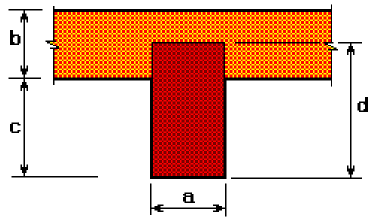
e) Local load on the corner of an element (9, *d*)



f) Local load applied to a part of the length and width of an element (9, *e*)



g) Local load within the protruding part of the wall (9, *f*)



h) Local load within a wall and its protruding part (9, *g*)

Figure 8.9.6-1. Local load arrangement

The **Damage** tab provides data on the mechanical damages which can be taken into account when checking the local strength. Mechanical damages of the masonry are assumed uniform throughout the whole volume of the checked fragment.

8.10 Reinforced Masonry Structures

The information given below includes general limitations of the current version of **KAMIN** and concerns all modes of the **Reinforced Masonry Structures** group.

Only elements with the transverse mesh reinforcement are considered. The reinforcement meshes are either rectangular or of the “zigzag” type placed in two adjacent seams of masonry. The reinforcement of the following classes is used: A-I, Bp-I according to SNiP 2.03.01-84* (A240, Bp500 of SP 63.13330).

Limitations on the rebar diameters:

- in the rectangular meshes — 3 to 6 mm;
- in the zigzag meshes — 3 to 8 mm.

Limitations on the rebar spacing in meshes:

- in the rectangular meshes — 3 to 12 cm;
- in the zigzag meshes — 3 to 12 cm;
- the rebar spacing is the same in both directions and in all meshes.

The diameters of the rebars in both directions are the same in the rectangular meshes; both zigzag meshes are also identical.

The cross-sections can be filled with reinforcement within the following limits:

- the minimum reinforcement percentage is 0.1;
- the maximum reinforcement percentage is 1.

In the case when the reinforcement percentage is less than the minimum value, the analysis cannot be performed. In the case when the reinforcement percentage is greater than the maximum one, the check will be performed for the maximum value of the reinforcement percentage. A respective warning message is generated in both cases.

Limitations on the materials used:

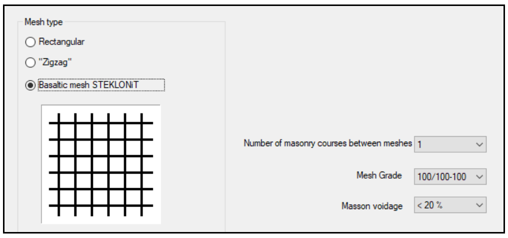
- stones over 150 mm high and rubble are not used;
- only mortars of grade 25 and higher are used.

The strength of the reinforced masonry in the course of its erection is not checked (i.e. formula 28 of SNiP II-22-81, formula (7.24) of SP 15.13330.2020 is not implemented).

Moreover, the check of an eccentrically compressed reinforced masonry structure with a large eccentricity will *not* be performed in cases when the eccentricity exceeds half of the distance between the cross-section mass center and the external edge of the compression area; the respective warning message will be generated.

See additional limitations in the description of the **Local Strength of Reinforced Structures** mode.

Besides the masonry with reinforcement meshes, the application enables to analyze the masonry with basalt meshes, STEKLONiT (basalt meshes should be used when the grades of mortar in the seams are not lower than M75). In this case, the user has to specify the number of courses between the meshes, the mesh grade and the data on the stone voidage.



The analysis is performed in accordance with the manufacturer's data on strength characteristics of the mesh and “Recommendations for the Design of Masonry Structures Reinforced with Basalt Meshes Produced by JSC STEKLONiT” developed by V. Kucherenko Central Research Institute for Structural Constructions.

EN 1996-1-1 does not provide any recommendations for the calculation of masonry design resistance for reinforced masonry structures with horizontal reinforcement. Therefore, the program uses the relevant recommendations of SNiP.

Moreover, for EN 1995-1-1 and DBN it is necessary to set information about masson's height.

8.10.1 Axially Compressed Reinforced Columns

The mode is used to check the strength and stability of separate or built-in axially compressed reinforced columns. It implements requirements of Sec. 4.30 and those related to this section of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.31 of SP 15.13330.2020). The load-bearing capacity is checked in the most dangerous lower section. The columns are assumed to have a constant cross-section throughout their height. Rectangular, T-shaped, round, and ring cross-sections are available.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Effective Height In XoY/XoZ Plane**;
- **Damage**;
- **Reinforcement Data**.

The **General**, **Design**, **Effective Height In XoY Plane**, **Effective Height In XoZ Plane**, and **Damage** tabs are almost identical to the respective tabs of the **Axially Compressed Columns** mode.

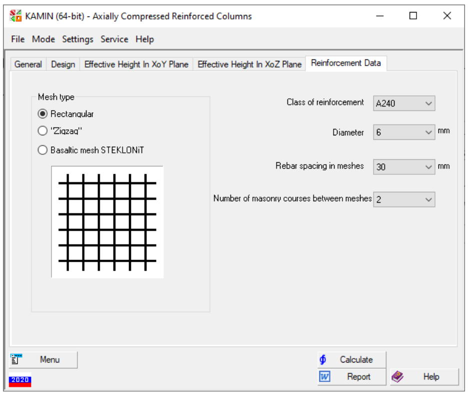


Figure 8.10.1-1. *The* **Reinforcement Data** *tab*

The **Reinforcement Data** tab (Fig 8.10.1-1) is used to select the following data from the respective lists: class of reinforcement, rebar diameter, rebar spacing in meshes, and number of masonry courses between the meshes. Use the respective radio buttons to select a mesh type (either rectangular or "zigzag").

8.10.2 Eccentrically Compressed Reinforced Columns

The mode is used to check the strength and stability of separate or built-in eccentrically compressed reinforced columns. The eccentric application of a load is assumed only in one principal plane of the column cross-section. The mode implements requirements of Sec. 4.31 and those related to this section of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.32 of SP 15.13330.2020). The columns are assumed to have a constant cross-section throughout their height. Rectangular, T-shaped, round, and ring cross-sections are available.

There are the following differences:

- stones over 150 mm high and rubble are not used;
- only mortars of grade 25 and higher are used.

The column is checked for stability both in the moment (eccentricity) plane and out of the moment plane. The peculiarities of the in the moment plane check are similar to the respective calculation in the **Eccentrically Compressed Columns** mode. The check out of the moment plane is performed in the same way as for an axially compressed bar.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Effective Height In XoY/XoZ Plane**;
- **Interaction Curves**;
- **Damage**;
- **Reinforcement Data**.

The structure and usage of this mode are very similar to the modes described above, **Axially Compressed Reinforced Columns** and **Eccentrically Compressed Columns**. The **General**, **Design**, **Damage**, **Interaction Curves**, **Effective Height In XoY Plane**, **Effective Height In XoZ Plane**, and **Reinforcement Data** tabs are identical to the respective tabs of other modes.

8.10.3 Reinforced External Wall

This mode is used to check a longitudinal reinforced external wall of a building within a storey height. The wall may have openings. A wall fragment should be specified along its length in such a way so as to define the openings and piers. The length of the fragment (and further the length of the checked element) depends on the presence of openings. If there are none, one meter of the wall length is considered. The openings (as well as the piers) are assumed to be equal throughout the wall length. If there are openings, a fragment equal to the pier width is considered.

The check of stability under eccentric compression out of the wall plane is the main one. The mode implements requirements of Sec. 4.31 and those related to this section of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.32 of SP 15.13330.2020). The wall within one storey is considered as a span of a continuous beam. The stability in the wall plane is assumed to be provided (even if there are openings), and is not checked. The piers are additionally checked for stability in the wall plane (according to Sec. 4.5, 4.30 of SNiP II-22-81, KMK 2.03.07-98, Sec. 7.5, 7.31 of SP 15.13330.2020) as being axially compressed.

The check is performed for the following cross-sections:

- in the upper part of the wall directly under the floor slab;
- in the middle part of the wall;
- in the lower part of the wall, when it lies on a lower floor slab or on a foundation.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Loads**;
- **Effective Height and Thickness** (for EN 1996-1-1 and DBN);
- **Accounting for Eccentricities** (for EN 1996-1-1 and DBN)**;**
- **Damage** (for SNiP);
- **Reinforcement Data**.

The **Design** and **Loads** tabs are identical to the respective tabs of the **External Wall** mode. The other tabs are identical to those of the **Axially Compressed Reinforced Columns** mode.

8.10.4 Reinforced Basement Wall

This mode is used to check a reinforced basement wall of a building within the height of the basement storey. The basement wall has no openings. One meter of the wall length is considered; the cross-section is rectangular.

The check of stability under eccentric compression out of the wall plane is the main one. The accompanying checks are performed as well (tension and, if necessary, shear). The mode implements requirements of Sec. 4.30, 4.31 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.31, 7.32 of SP 15.13330.2020). The basement wall is assumed to be simply supported on the foundation and on the floor slab over the basement. The stability in the wall plane is assumed to be provided and is not checked. A random eccentricity according to Sec. 6.65 of SNiP II-22-81 (Sec. 9.71 of SP 15.13330.2020) is taken into account only for loads from the upper wall.

The check is performed for the following cross-sections:

- in the upper part of the wall directly under the floor slab, treated as an eccentrically compressed element;
- in the middle part of the wall;
- in the lower part of the wall, when it lies on a foundation, treated as an axially compressed element.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design and Loads**;
- **Damage** (for SNiP);
- **Effective Height and Thickness** (for EN 1996-1-1 and DBN)**;**
- **Accounting for Eccentricities** (for EN 1996-1-1 and DBN);
- **Reinforcement Data**.

The **General** tab contains general data on the structure and is identical to the respective tab of the **Axially Compressed Columns** mode.

The **Design and Loads** tab is identical to that of the **Basement Wall** mode. The other tabs are identical to the respective tabs of the **Axially Compressed Reinforced Columns** mode.

8.10.5 Local Strength of Reinforced Structures

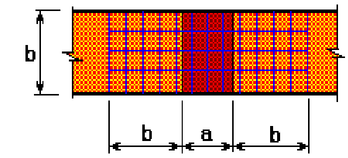
This mode is used to perform checks of the local strength in places where concentrated loads are transferred (from the bearing of beams, purlins, and other members) onto reinforced walls and columns. The mode implements requirements of Sec. 4.13, 4.14, 4.16 of SNiP II-22-81 (Sec. 7.13, 7.14, 7.16 of SP 15.13330). The simultaneous action of the local and main loads (Sec. 4.15 of SNiP II-22-81, Sec. 7.15 of SP 15.13330) is not checked. The distribution of pressure in places where the loads are transferred is assumed to be uniform throughout the whole transfer area.

The dialog box of this mode contains the following tabs:

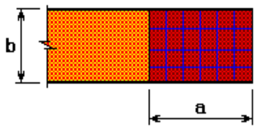
- **General**;
- **Load Arrangement**;
- **Damage** (for SNiP);
- **Reinforcement Data**.

The **General** and **Damage** tabs are identical to the respective tabs of the **Local Strength** mode. The **Reinforcement Data** tab is identical to the respective tab of the **Axially Compressed Reinforced Columns** mode.

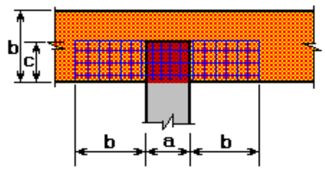
The **Load Arrangement** tab is used to specify the value of the local load and to select its arrangement. Fig. 8.10.5-1 shows the available load arrangements and indicates (in brackets) the respective designs from Fig. 9 of SNiP II-22-81 (Fig.7.6 SP 15.13330.2020) (an area where the load is applied is highlighted). The reinforcement is shown only where it is minimally necessary — in an effective area of the cross-section determined according to Sec. 4.16 of SNiP II-22-81 (Sec. 7.16 of SP 15.13330). Only rectangular meshes with the spacing not more than 60×60 mm are used for the reinforcement.



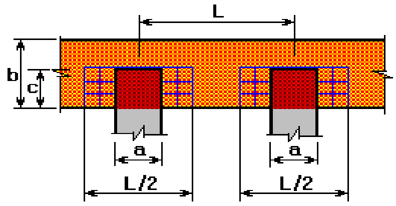
a) Local load throughout the width of an element (9, *a*)



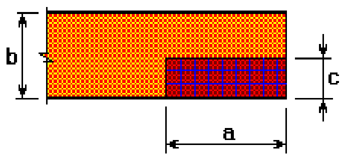
b) Local edge load throughout the width of an element (9, *b*)



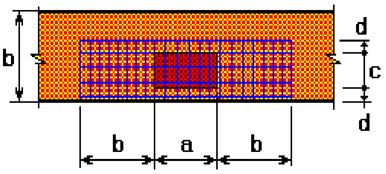
c) Local load at the end supports of beams and purlins (9, *c*)



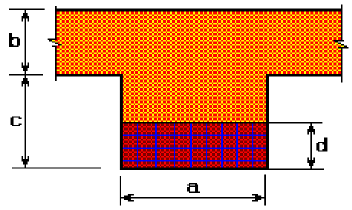
d) Local load at the end supports of beams and purlins (9, c_1)



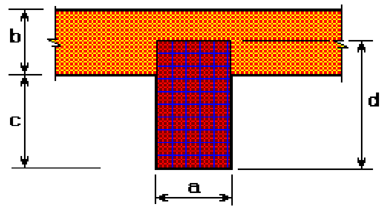
e) Local load on the corner of an element (9, *d*)



f) Local load applied to a part of the length and width of an element (9, e)



g) Local load within the protruding part of the wall (9,*f*)



h) Local load within a wall and its protruding part (9,*g*)

Figure 8.10.5-1. *Local load arrangement*

8.11 Structures under Reconstruction

The modes of this group are used to check various structural members of masonry structures, including the damaged ones reinforced by steel battens.

The damages of masonry structures can be of mechanical or fire origin. Damages are classified according to Table 1 (mechanical ones in walls, piers, and columns), Table 2 (mechanical damages of masonry supports of beams, trusses, purlins and lintels), Table 3 (fire damages of the masonry of walls and columns) from Recommendations [3] (see also tables 3-6 SP 427.1325800.2018). The damages are taken into account by reducing the design strength of masonry and reducing the cross-sectional dimensions of elements. The mechanical and fire damages can take place simultaneously in a single structural element (except for the checks of local strength where only the mechanical damages are considered). The fire damages are taken into account only if the considered wall is at least 38 cm thick, or if the cross-sectional dimension of the considered column is at least 38 cm. In cases when the damages cannot be taken into account, the respective message is generated. The overall maximal reduction of the design strength caused by both mechanical and fire damages must not exceed 50%. The columns and wall fragments are reinforced according to the recommendations of the Manual [2], Guide [4], and SP 427.1325800.2018.

8.11.1 Axially Compressed Columns Reinforced by Battens

The mode is used to check the strength and stability of separate or built-in axially compressed columns reinforced by battens. The columns are assumed to have a constant cross-section throughout their height.

Rectangular and T-shaped cross-sections are available. The reinforcement is performed from vertical equal angles placed at the corners of a column and horizontal battens connecting these angles. The horizontal battens are not pre-stressed. The forces acting on a member are not transferred to the reinforcing elements (battens). The vertical distances between the transverse reinforcing elements are assumed to be not more than the least size of the reinforced member or 50 cm, whichever is less, and the distance between the vertical elements in plan is assumed to be not more than 100 cm or two thicknesses of the member. Recommendations from Sec. 5.34, 5.35, 5.38, 5.40 of the Manual [2], Sec. 5.42, 5.45, 5.46 of the Guide [4], and Sec. 8.21-8.27 of SP 427.1325800.2018 are taken into account.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Effective Height In XoY/XoZ Plane**;
- **Damage**;
- **Reinforcing**.

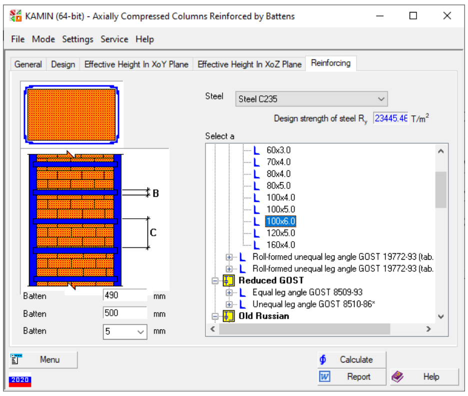


Figure 8.11.1-1. *The* **Reinforcing** *tab*

The **General**, **Design**, **Effective Height In XoY Plane**, **Effective Height In XoZ Plane**, and **Damage** tabs are identical to the respective tabs of the **Axially Compressed Columns** mode.

The **Reinforcing** tab (Fig. 8.11.1-1) contains data on the design of the reinforcing, steel class, and vertical angles from the assortments.

The vertical and horizontal reinforcing elements are assumed to be made of steel of the same class.

8.11.2 Eccentrically Compressed Columns Reinforced by Battens

The mode is used to check the strength and stability of separate or built-in eccentrically compressed columns reinforced by battens. The columns are assumed to have a constant cross-section throughout their height. Rectangular and T-shaped cross-sections are available. The reinforcement is performed from vertical equal angles placed at the corners of a column and horizontal battens connecting these angles. The horizontal battens are not pre-stressed. The forces acting on a member are not transferred to the reinforcing elements (battens). The vertical distances between the transverse reinforcing elements are assumed to be not more than the least size of the reinforced member or 50 cm, whichever is less, and the distance between the vertical elements in plan is assumed to be not more than 100 cm or two thicknesses of the member. Recommendations from Sec. 5.34, 5.35, 5.38, 5.40 of the Manual [2], Sec. 5.42, 5.45, 5.46 of the Guide [4] , and Sec. 8.21-8.27 of SP 427.1325800.2018 are taken into account.

The column is checked for stability both in the moment (eccentricity) plane and out of the moment plane. The peculiarities of the in the moment plane check are similar to the respective calculation in the **Eccentrically Compressed Columns** mode. The check out of the moment plane is performed in the same way as for an axially compressed bar.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Effective Height In XoY/XoZ Plane**;

- **Interaction Curves**;
- **Damage**;
- **Reinforcing**.

The **General**, **Design**, **Interaction Curves**, **Effective Height In XoY Plane**, **Effective Height In XoZ Plane**, and **Damage** tabs are identical to the respective tabs of the **Eccentrically Compressed Columns** mode.

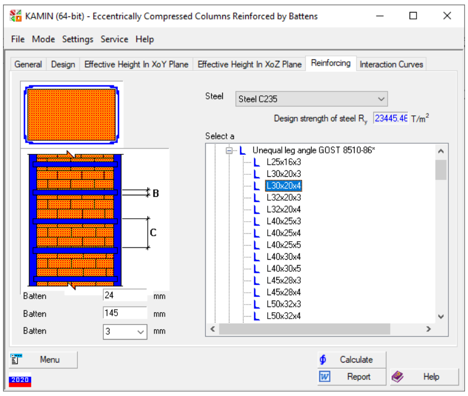


Figure 8.11.2-1. *The* **Reinforcing** *tab*

The **Reinforcing** tab (Fig. 8.11.2-1) contains data on the design of the reinforcing, steel class, and vertical angles from the assortments.

The vertical and horizontal reinforcing elements are assumed to be made of steel of the same class.

8.11.3 Building Wall Reinforced by Battens

This mode is used to check a longitudinal building wall without openings within a storey height. The length of a wall segment reinforced by battens should be specified; the wall is assumed to have a rectangular cross-section. The vertical battens are installed on the edges of the reinforced wall segment and, if necessary, evenly along the reinforced segment. Additional vertical battens are installed on the condition that the distance between the vertical reinforcing elements must not exceed one meter or two thicknesses of the wall. The reinforcing elements are not pre-stressed. The forces acting on the wall are not transferred to the battens. The distance between the transverse battens must not exceed 50 cm or the thickness of the wall. The longitudinal and transverse reinforcing elements must be connected at their intersections. Moreover, the corresponding joints on the opposite sides of the wall are connected to each other by horizontal round structural elements going through the wall. These structural elements are not considered in the analysis.

The reinforced wall segment is checked for stability out of the wall plane under eccentric compression. The mode implements requirements of Sec. 4.7–4.9, 4.11 of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.7-7.9, 7.11 of SP 15.13330) taking into account the recommendations of Sec. 5.34, 5.35, 5.38, 5.40 of the Manual [2], Sec. 5.42, 5.45, 5.46 of the Guide [4], and Sec. 8.21-8.27 of SP 427.1325800.2018.

The wall within one storey is considered as a span of a continuous beam. The stability of a reinforced segment in the wall plane is assumed to be provided, and is not checked.

The check is performed for the following cross-sections:

- in the upper part of the wall directly under the floor slab;
- in the middle part of the wall;
- in the lower part of the wall, when it lies on a lower floor slab or on a foundation.

Only one stability check of the reinforced wall segment is performed in each cross-section (the accompanying checks, shear and tension, are not performed).

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Loads**;
- **Damage**;
- **Reinforcing**.

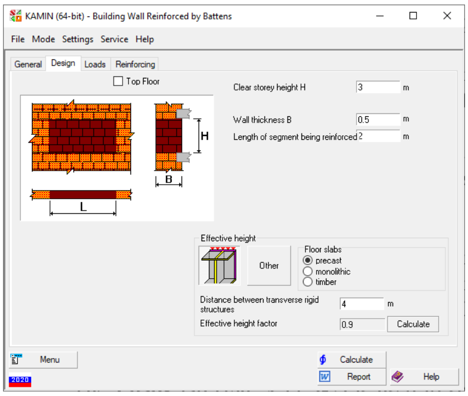


Figure 8.11.3-1. *The* **Design** *tab*

The **General** tab is identical to the respective tab of the **Axially Compressed Columns** mode.

The **Design** tab (Fig. 8.11.3-1) contains information on the **reinforced** wall segment (highlighted in the figure) and on its dimensions.

The effective height is assigned in the same way as in the **External Wall** mode. The supporting sections of the wall are clamped. The obtained stress reduction factors are assumed to be constant throughout the wall height.

If the **Top Floor** checkbox is selected, then the calculation model of the wall is used in the form of a single-span beam, in which the lower support is rigid supported, and the upper one has a hinge, otherwise rigid support is assumed at both ends.

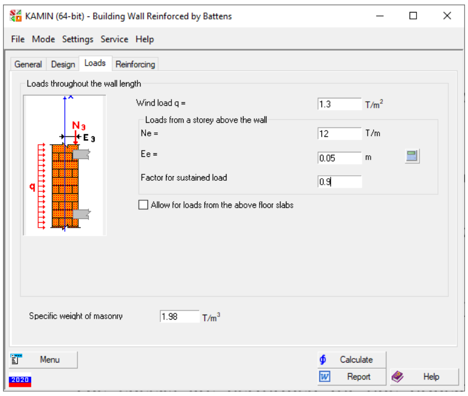


Figure 8.11.3-2. *The* **Loads** *tab*

The **Loads** tab (Fig. 8.11.3-2) is used to specify the following loads:

- wind, normal to the wall surface;
- loads from the above stories applied to the top of the wall or of the pier eccentrically in the general case;
- loads from a floor slab bearing directly on a wall applied axially;
- dead weight of the wall.

All loads are assumed to be defined by their design values. The full values of loads and factors of sustained loads are specified for loads from the floor slabs.

The checks are performed for one principal combination of loads, the wind load being taken with the factor of 0.9 and the other loads – with the factor of 1.

The dead weight of the wall is always taken into account.

The **Damage** tab is identical to the respective tab of the **Axially Compressed Columns** mode. The wall may have fire damages on its one side, or on two sides.

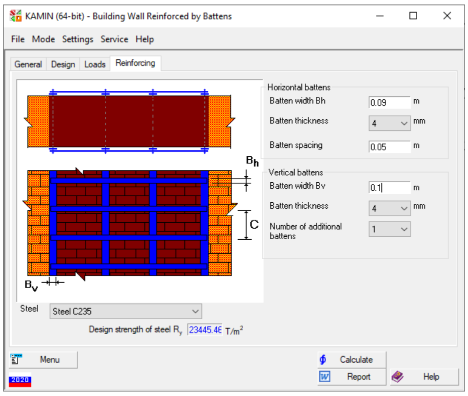


Figure 8.11.3-3. *The* **Reinforcing** *tab*

The **Reinforcing** tab (Fig. 8.11.3-3) contains data on the design of the reinforcing and its steel class.

The vertical and horizontal reinforcing elements are assumed to be made of steel of the same class.

8.11.4 Opening in a Wall

This mode considers an opening in an existing solid brick wall with the following features:

- the openings do not affect the loads on the wall;
- the lower part of the opening is not defined, i.e. it is not specified whether it is a door or a window opening;
- it is assumed that there is a floor slab above the considered opening (in a particular case, the opening can be made directly below the floor slab);
- the wall with the opening can be either exterior or interior, and it must not have protruding parts;
- the wall may have mechanical and fire damages;
- low-strength natural stones and rubble are not used;
- mortars of grade 25 and higher are used.

The opening is bounded by a steel lintel formed by double angles (equal or unequal), double channels, or one I-beam with a horizontal web.

Masonry above the opening is considered as a coursed lintel. If the lintel has sufficient height, its strength is checked as for a coursed lintel.

The stability of the lintel out of the wall plane is assumed to be provided. The mode implements requirements of Sec. 4.7, 4.8, 6.47 and those related to these sections of SNiP II-22-81, KMK 2.03.07-98 (Sec. 7.7, 7.8, 9.53 of SP 15.13330.2020). If the masonry lintel has insufficient height or insufficient strength, the bending steel lintel is checked for strength. The requirements of Sec. 5.12 of SNiP II-23-81* (Sec. 8.2.1 of SP 16.13330) are implemented. The combined behavior of masonry and the steel lintel is not taken into account. The local strength of masonry under the steel lintel is always checked. The main check for the masonry lintels is the stability check at the eccentric compression in the plane of the wall which is performed in the most dangerous section of the lintel defined by the moment diagram.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Loads**;
- **Damage**.

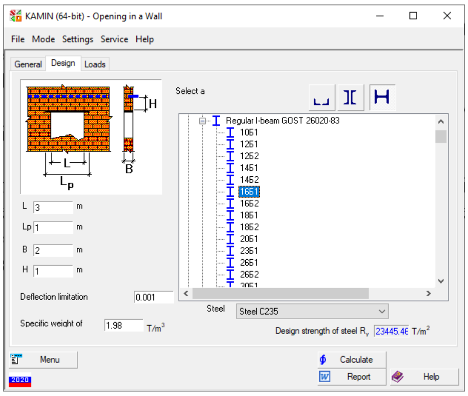


Figure 8.11.4-1. *The* **Design** *tab*

The **General** and **Damage** tabs are almost identical to the respective tabs of the **Axially Compressed Columns** mode. The **Loads** tab is identical to the respective tab of the **Lintels** mode.

The **Design** tab (Fig. 8.11.4-1) is used to specify data on the opening and the steel lintel.

The lintel can be made from angles, channels, or I-beams. A section can be selected from the respective list. The design strength of steel is assigned automatically once its grade is selected from the respective list; it can be also specified in the respective text field if you select **Other** in the list.

8.12 Support Joints

8.12.1 Hanging Walls

The mode is used to check the local strength of a hanging wall bearing on a foundation beam in the areas adjacent to the supports of the foundation beam.

The height of the wall must not be less than half of the beam span for the assumptions, which the analysis is based on, to be true.

The wall is axially supported on the beam.

The local strength is checked only above the foundation beam.

The foundation beams can be either single- or multi-span. The single-span beams are simply supported, and the end supports of the multiple-span beams are simply supported as well.

The reactions in a multi-span beam are determined as for a three-span continuous beam with equal spans, loaded by a uniform load from the wall weight in all its spans.

The cross-sections of the foundation beams are constant throughout their length. All the supports have the same width. The supported part of a beam in each span is half the width of the support in all cases.

The foundation beam can be made either of reinforced concrete or steel.

There may be one window or door opening within the wall span. It is assumed that there are no openings in the adjacent spans (in the case of a continuous beam).

The strength of a single beam span (a middle or an end one in continuous beams) with or without an opening is always checked.

The load decreased by the presence of an opening is corrected according to a statically determinate model within the span.

The dialog box of this mode contains the following tabs:

- **General**;
- **Foundation Beam;**
- **Design**.

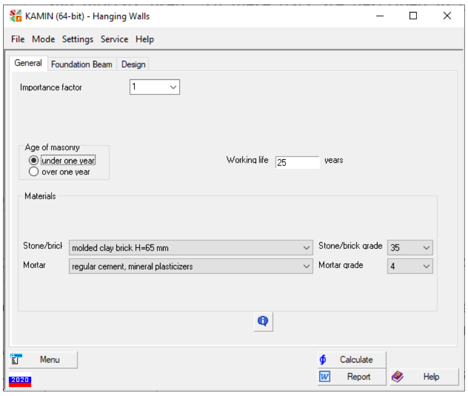


Figure 8.12.1-1. *The* **General** *tab*

The **General** tab (Fig. 8.12.1-1) contains general data on the structure: the importance factor, age of masonry, working life, and data on the material — the types and grades of the stones/bricks and mortars used. All these data are described in the **Materials** section.

The **Foundation Beam** tab contains data on the design of a foundation beam (single- or multi-span, middle or end span, the length of the considered span, and the presence of openings within the checked span).

For steel beams (Fig. 8.12.1-2,*a*), its cross-section type and dimensions should be specified. In cases when the cross-section is made from rolled profiles, the respective profile is selected from the assortment. For a box cross-section, its transverse dimensions and thickness are specified by the user.

For reinforced concrete beams (Fig. 8.12.1-2,*b*), its cross-section type and dimensions should be specified. The user also has to specify all the required information on concrete.

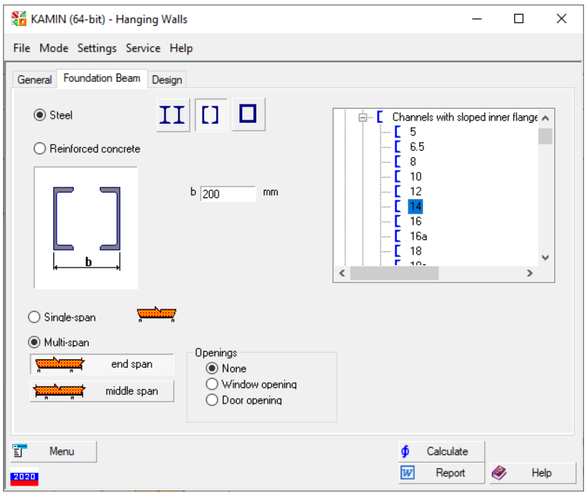


Figure 8.12.1-2, *a*. *The* **Foundation Beam** *tab* (a steel beam)

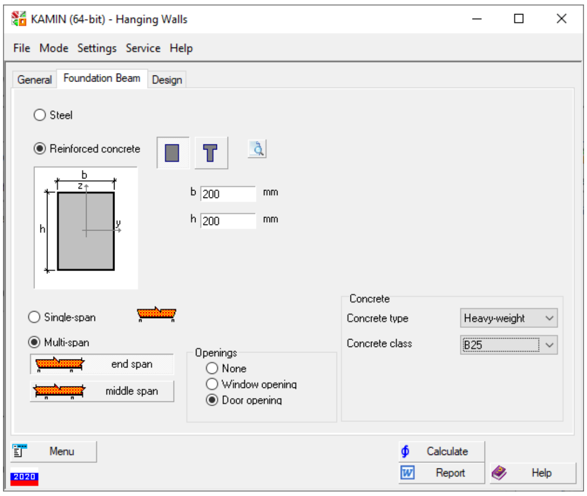


Figure 8.12.1-2, *b*. *The* **Foundation Beam** *tab* (a reinforced concrete beam)

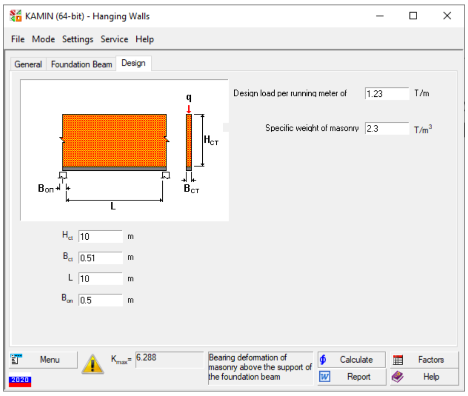


Figure 8.12.1-3, *a*. *The* **Design** *tab* (a multi-span beam, end span, no openings)

The **Design** tab (Fig. 8.12.1-3, *a-c*) contains data on the design of the wall, specific weight of masonry, the length of the considered span, the dimensions of supports and openings.

This tab is also used to specify the design load transferred onto the wall from the above structures.

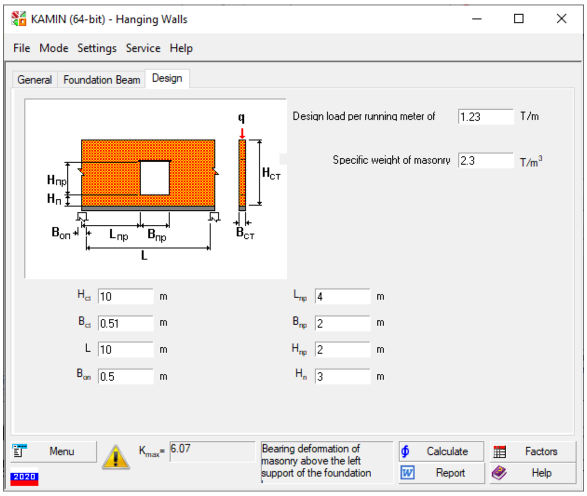


Figure 8.12.1-3, *b*. *The* **Design** *tab* (a single-span beam, a window opening)

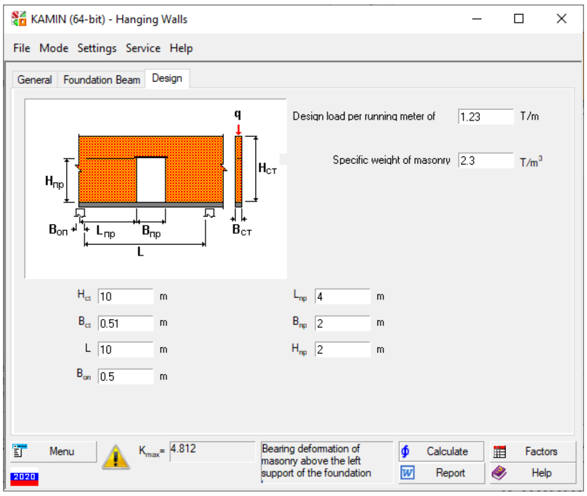


Figure 8.12.1-3, *c*. *The* **Design** *tab* (a multi-span beam, a middle span, a door opening)

Limitations

- the strength under the supports of the foundation beams is not checked (the requirement of Sec. 6.51, paragraph 2 of SNiP II-22-81, Sec. 9.57, paragraph 2 of SP 15.13330.2020 is not implemented);
- the strength of the spandrel beam is not checked as well (the requirement of Sec. 6.53 of SNiP II-22-81, Sec. 9.59 of SP 15.13330.2020 is not implemented);
- when determining the stiffness of reinforced concrete beams, the initial elasticity modulus, E_b, is taken for concrete, and it is not modified afterwards;
- the requirement of Sec. 6.51 of SNiP II-22-81, third paragraph (Sec. 9.57 of SP 15.13330.2020) concerning the check of cross-sections above the foundation beam is not implemented. In our version this is a section at the level of the bottom of the window opening;
- when the masonry above intermediate supports of multi-span beams is checked for strength, and the door openings are within the action of local stresses, a fullness factor of the stress diagram ψ at the cut-off part of the diagram is increased proportionally to a ratio of the full base of the stress diagram to the truncated one;
- the width of the beam top must be not less than the wall thickness.

8.12.2 Bearing of Beams and Slabs on a Wall

The mode is used to check the local strength of masonry at the areas of the beams and slabs bearing on walls. The mode implements requirements of Sec. 6.46 in SNiP II-22-81, KMK 2.03.07-98 (Sec. 9.52 of SP 15.13330.2020) and Sec. 7.3 of Designer's manual [5].

The mode enables to perform a check of bearing of solid reinforced concrete slabs, steel or reinforced concrete beams. Only single beams and slabs are considered. A mutual effect of nearby structures is not taken into account.

All supported structures can be simply supported on both sides, clamped on both sides, or cantilever.

You have to specify the design value of load on the supported structures uniformly distributed over a unit area for slabs, and unit length for beams. The self-weight of the supported structure is taken with the importance factor of 1,05 for steel beams and 1,1 – for reinforced concrete beams and slabs.

You can select one of the following structural designs of the beam/slab bearing on masonry:

- simply supported beams/slabs;
- beams/slabs clamped in masonry;
- beams clamped in masonry with a distribution packing plate below;
- beams clamped in masonry with a distribution packing plate below and above.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**.

The **General** tab (Fig. 8.12.2-1) contains general data on the structure: the importance factor, age of masonry, working life, and data on the material — the types and grades of the stones/bricks and mortars used. All these data are described in the **Materials** section.

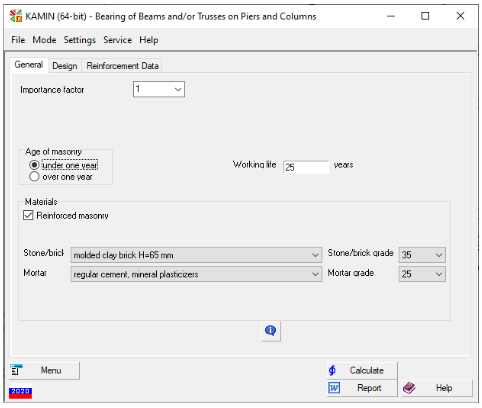


Figure 8.12.2-1. *The* **General** *tab*

The **Design** tab contains data on the thickness of the wall, material and design of a beam/slab, support conditions, the length of the considered span, and the dimensions of supports.

This tab is also used to specify the design load transferred onto the beam/slab from the above structures.

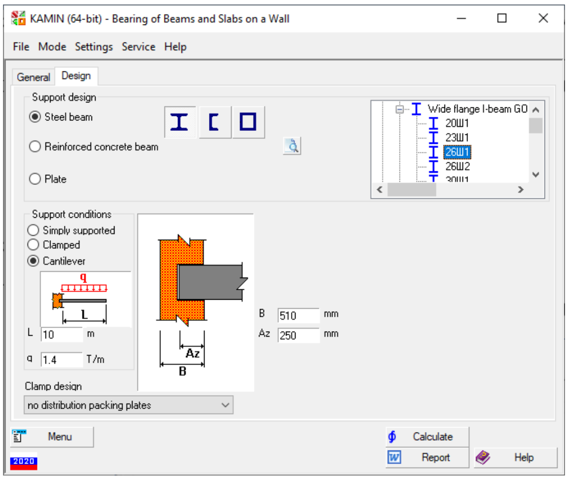


Figure 8.12.2-2, *a*. *The* **Design** *tab* (bearing of a cantilever steel beam)

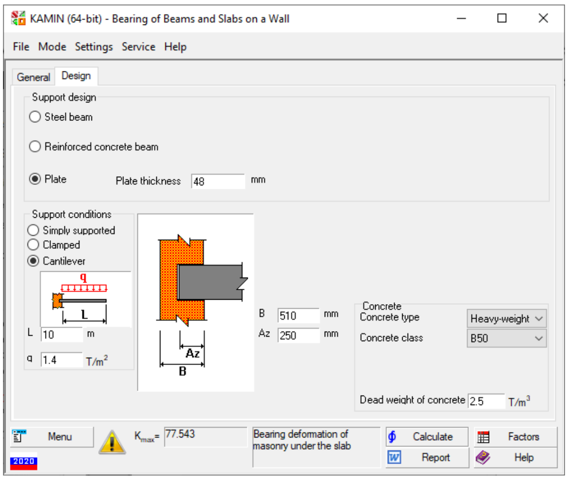


Figure 8.12.2-2, *b*. *The* **Design** *tab* (bearing of a cantilever reinforced concrete slab)

Limitations

- the strength of the steel packing plates is assumed to be provided and is not checked;
- the lower and upper packing plates are assumed to have the same width (in the direction perpendicular to the wall plane);
- the length of the upper packing plate must not exceed that of the lower one;
- the width of the packing plate should be not less than 1/5 of the embedment depth. In cases when the width of the packing plate does not meet this requirement, the user is recommended to increase its width up to the required value;
- the effective width of the lower packing plate is determined from the condition (7.21) of Designer's manual [5]. In cases when the effective width of the packing plate is less than that specified by the user, the check cannot be performed;
- the embedment depth is not determined by the formula (54) of SNiP; the embedment is checked according to the formula (53) of SNiP.

8.12.3 Bearing of Beams and/or Trusses on Piers and Columns

This mode is used to check the local strength of masonry and reinforced masonry at the areas of the beams and/or trusses bearing on piers and columns. The supported structure is a single beam/truss with a rectangular support base. The size of the support base is limited by the bearing part of the structure or by the size of a steel packing plate at the support.

The check implements Sec. 6.44 of SNiP II-22-81, KMK 2.03.07-98 (Sec. 9.50 of SP 15.13330.2020) and recommendations of § 5.6 of the book by P.F. Vakhnenko [1].

The supporting structure is a pier of a wall or a column.

The peculiarity of the supporting structure is that the beam/truss is supported on a part of the masonry. There is no masonry above the support. A transverse reinforcement of the masonry with meshes can be made under the bearing part. The meshes are always distributed throughout the bearing area, the width of the pier, and the thickness of the wall.

You can select one of the following structural designs of the bearing area:

- directly on masonry;
- on a reinforced concrete cushion placed over the whole width of the pier;
- on a reinforced concrete cushion placed over the whole width of the pier through an aligning steel packing plate. The aligning packing plate occupies the whole width of the supported structure and can stick out.

The design load from the supported structure is specified as a concentrated force.

The check for local bearing, when the structure is supported directly on masonry, is performed for the masonry cross-section under the beam/truss; and if it is supported through a reinforced concrete cushion, the masonry cross-section under the cushion is used.

The dialog box of this mode contains the following tabs:

- **General**;
- **Design**;
- **Reinforcement Data**.

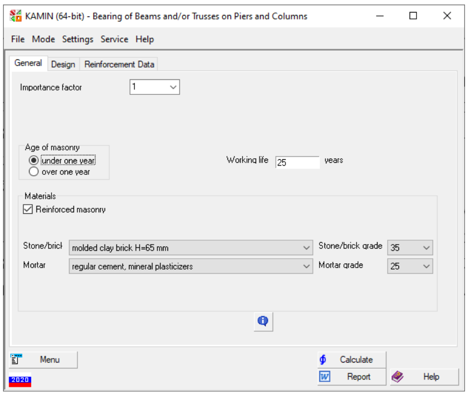


Figure 8.12.3-1. *The* **General** *tab*

The **General** tab (Fig. 8.12.3-1) contains general data on the structure: the importance factor, age of masonry, working life, and data on the material — the types and grades of the stones/bricks and mortars used. All these data are described in the **Materials** section.

This tab is also used to specify whether the masonry below the support is reinforced or not reinforced. Depending on this, the user has options of bricks/stones for either not reinforced or reinforced masonry according to the general rules assumed in **KAMIN**.

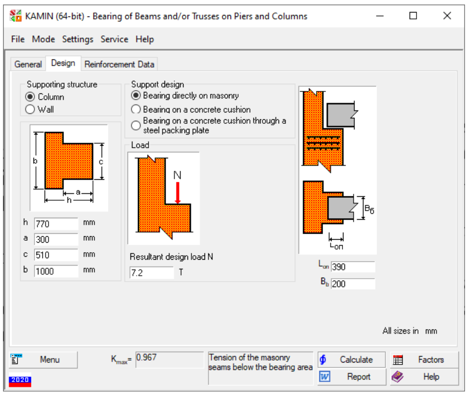


Figure 8.12.3-2, *a*. *The* **Design** *tab* (bearing directly on a column, reinforced bearing area)

The **Design** tab (Fig. 8.12.3-2, *a-c*) contains data on the column/wall, the load, the dimensions of the bearing area, the dimensions of the reinforced concrete cushion and the steel packing plate.

If there is a reinforced concrete cushion, you also have to specify data on the concrete.

The material and the thickness of the steel packing plate are not specified; the strength of the packing plate is assumed to be provided and is not checked.

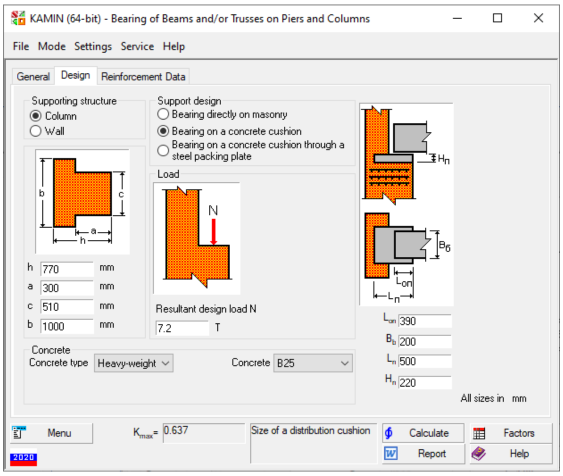


Figure 8.12.3-2, *b*. *The* **Design** *tab* (bearing on a wall through a reinforced concrete cushion)

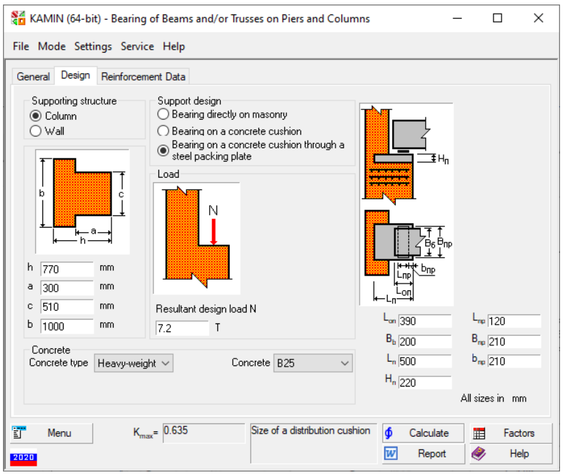


Figure 8.12.3-2, *c*. *The* **Design** *tab* (bearing on a wall through a reinforced concrete cushion and an aligning packing plate, reinforced bearing area)

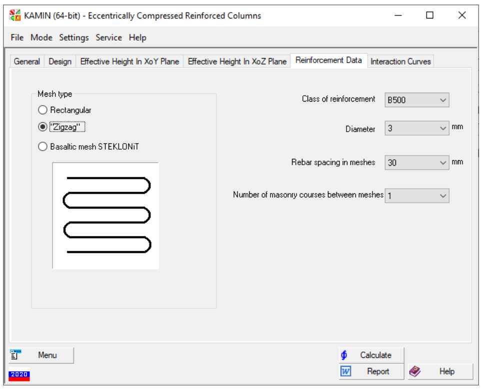


Figure 8.12.3-3. *The* **Reinforcement Data** *tab*

The **Reinforcement Data** tab (Fig. 8.12.3-3) contains information on the reinforcement of the wall/column part below the bearing place.

This tab is identical to the respective one of the **Axially Compressed Reinforced Columns** mode.

Limitations

- if the structure is bearing on a wall, the check for axial compression below the bearing area uses a conventional cross-section of a column with greater side equal to a doubled pier width;
- in the check of masonry cross-section under the cushion, the resultant load is assumed to be applied at the distance of 1/3 from the interior edge of the cushion if there is no distribution packing plate, or from the interior edge of the packing plate if there is one;
- the check for tension of the masonry seams and of the reinforcement (if the masonry is reinforced below the bearing area) is performed only if the structure is supported directly on the masonry. The check is based on a principle given in § 5.6 of the book by P.F. Vakhnenko [1]. Vertical seams of the masonry are always checked for tension. The obtained result is final for not reinforced masonry. The reinforcement of the reinforced masonry is checked for tension if the tensile strength of the masonry itself is insufficient.

8.13 Reference Information

8.13.1 Specific Weights

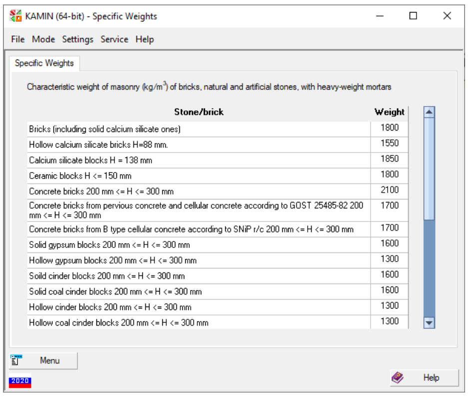


Figure 8.13.1-1. *The* **Specific Weights** *tab*

This mode (Fig. 8.13.1-1) provides a table with reference information on the specific weights of masonry with heavy-weight mortars for all types of stones/bricks used in the application.

8.13.2 Damage Classification

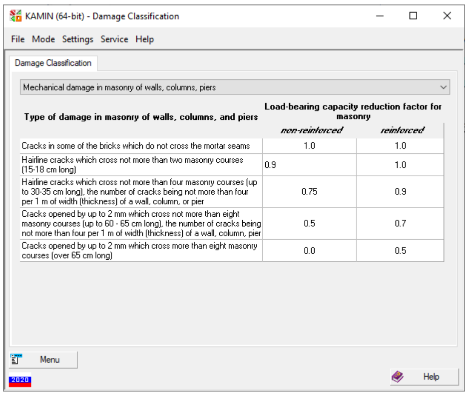


Figure 8.13.2-1. *The* **Damage Classification** *tab*

This mode (Fig. 8.13.2-1) provides information on the load-bearing capacity reduction factors for masonry and reinforced masonry structures with various mechanical and fire damages, and recommendations on temporary reinforcing in compliance with the "Recommendations" [3] and SP 427.1325800.2018.

To obtain the necessary information, select the respective item from the drop-down list.

8.14 Design Codes Implemented by KAMIN

Mode	Factors checked	Sections of design codes					
		SNiP II-22-81	SP 15.13330.2012	SP 15.13330.2020	EN 1996-1-1	DBN B.1.2.2-2006	KMK 2.03.07-98
MASONRY STRUCTURES							
Axially Compressed Columns	Stability under axial compression	Sec. 4.1 of SNiP II-22-81	Sec. 7.1 of SP 15.13330.2012	Sec. 7.1 of SP 15.13330.2020			Sec. 4.1 of KMK 2.03.07-98
Eccentrically Compressed Columns	Stability in the eccentricity plane under eccentric compression	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability out of the eccentricity plane under axial compression	Sec. 4.11 of SNiP II-22-81	Sec. 7.11 of SP 15.13330.2012	Sec. 7.11 of SP 15.13330.2020			Sec. 4.11 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
External Wall	Stability under eccentric compression of the cross-section under a floor slab	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability under eccentric compression of the middle cross-section	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability under eccentric compression of the lower cross-section	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability of a pier in the wall plane	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98

	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
	Massonry strength at top floor				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength at mid height of wall				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength at underside floor				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength for shear load at top floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at mid height of wall				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at underside floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
Basement Wall	Stability under eccentric compression of a cross-section under the floor slab above the basement	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability under eccentric compression of the middle cross-section	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Stability under axial compression of the lower cross-section	Sec. 4.1 of SNiP II-22-81	Sec. 7.1 of SP 15.13330.2012	Sec. 7.1 of SP 15.13330.2020			Sec. 4.1 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98

	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
	Massonry strength at top floor				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength at mid height of wall				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength at underside floor				Sec.6.1.2. 1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2. 1.1 formula (11.1)	
	Massonry strength for shear load at top floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at mid height of wall				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at underside floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
Lintels	Stability of a lintel	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98
	Shear in seams of the end pier	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks) of the end pier	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Tie strength	Sec. 5.1 of SNiP II-23-81*.	Sec. 8.1 of SP 15.13330.2012	Sec. 8.1 of SP 15.13330.2020			Sec. 7.1 of KMK 2.03.05-97
	Ultimate moment strength of the cross-section	Sec. 3.15-3.17, 3.26 of SNiP 2.03.01-84*					Sec. 3.15-3.17, 3.26 of KMK 2.03.01-96
	Strength accounting for the resistance of concrete in the tension area	Sec. 3.8 of SNiP 2.03.01-84*	Sec. 6.8 of SP 15.13330.2012	Sec. 6.11 of SP 15.13330.2020			Sec. 3.8 of KMK 2.03.01-96
	Strains in compressed concrete		Sec. 6.2.21-6.2.31 SP 63.13330	Sec. 6.2.21-6.2.31 SP 63.13330			

	Strains in tensioned reinforcement		Sec. 6.2.21-6.2.31 SP 63.13330	Sec. 6.2.21-6.2.31 SP 63.13330			
	Strength in an oblique strip between oblique cracks	Sec. 3.30 of SNiP 2.03.01-84*	Sec. 8.1.33 SP 63.13330	Sec. 8.1.33 SP 63.13330			Sec. 3.30 KMK 2.03.01-96
	Strength in an oblique strip without transverse reinforcement	Sec. 3.32 of SNiP 2.03.01-84*	Sec. 8.1.33 SP 63.13330	Sec. 8.1.33 SP 63.13330			Sec. 3.32 KMK 2.03.01-96
	Strength in an oblique crack	Sec. 3.31 of SNiP 2.03.01-84*	Sec. 8.1.33 SP 63.13330	Sec. 8.1.33 SP 63.13330			Sec. 3.31 KMK 2.03.01-96
	Local strength under the support of a reinforced concrete lintel	Sec. 4.13 of SNiP II-22-81	Sec. 7.13 of SP 15.13330.2012	Sec. 7.13 of SP 15.13330.2020			Sec. 4.13 of KMK 2.03.07-98
	Deflection of a reinforced concrete lintel	Sec. 10.1 of SNiP 2.01.07-85*	Sec. 15 of SP 20.13330	Sec. 15 of SP 20.13330			Sec. 10.1 KMK 2.03.01-96
Local Strength	Bearing deformation under the local load	Sec. 4.13 of SNiP II-22-81	Sec. 7.13 of SP 15.13330.2012	Sec. 7.13 of SP 15.13330.2020	Sec. 6.1.3(2) EN 1996-1-1	Sec. 11.1.3.2	Sec. 4.13, 6.46 of KMK 2.03.07-98

REINFORCED MASONRY STRUCTURES							
Mode	**Factors checked**	**Sections of design codes**					
		SNiP II-22-81	**SP 15.13330.2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
Axially Compressed Reinforced Columns	Stability under axial compression	Sec. 4.30 of SNiP II-22-81	Sec. 7.30 of SP 15.13330.2012	Sec. 7.31 of SP 15.13330.2020			Sec. 4.30 of KMK 2.03.07-98
Eccentrically Compressed Reinforced Columns	Stability in the eccentricity plane under eccentric compression	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Stability out of the eccentricity plane under axial compression	Sec. 4.30 of SNiP II-22-81	Sec. 7.30 of SP 15.13330.2012	Sec. 7.31 of SP 15.13330.2020			Sec. 4.30 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98

REINFORCED MASONRY STRUCTURES							
Mode	**Factors checked**	**Sections of design codes**					
		SNiP II-22-81	**SP 15.13330.2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.30 of KMK 2.03.07-98
	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
	Stability under eccentric compression of the cross-section under a floor slab	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Stability under eccentric compression of the middle cross-section	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
Reinforced External Wall	Stability under eccentric compression of the lower cross-section	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Stability of a pier in the wall plane	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
	Massonry strength at top floor				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	
	Massonry strength at mid height of wall				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	

REINFORCED MASONRY STRUCTURES							
Mode	**Factors checked**	**Sections of design codes**					
		SNiP II-22-81	**SP 15.13330.2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
	Massonry strength at underside floor				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	
	Massonry strength for shear load at top floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at mid height of wall				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at underside floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
Reinforced Basement Wall	Stability under eccentric compression of a cross-section under the floor slab above the basement	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Stability under eccentric compression of the middle cross-section	Sec. 4.31 of SNiP II-22-81	Sec. 7.31 of SP 15.13330.2012	Sec. 7.32 of SP 15.13330.2020			Sec. 4.31 of KMK 2.03.07-98
	Stability under axial compression of the lower cross-section	Sec. 4.30 of SNiP II-22-81	Sec. 7.30 of SP 15.13330.2012	Sec. 7.31 of SP 15.13330.2020			Sec. 4.30 of KMK 2.03.07-98
	Shear in seams	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98
	Shear in stones (bricks)	Sec. 4.20 of SNiP II-22-81	Sec. 7.20 of SP 15.13330.2012	Sec. 7.20 of SP 15.13330.2020			Sec. 4.20 of KMK 2.03.07-98

REINFORCED MASONRY STRUCTURES							
Mode	**Factors checked**	**Sections of design codes**					
		SNiP II-22-81	**SP 15.13330.2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
	Masonry seam opening	Sec. 5.3 of SNiP II-22-81	Sec. 8.3 of SP 15.13330.2012	Sec. 8.3 of SP 15.13330.2020			Sec. 5.3 of KMK 2.03.07-98
	Massonry strength at top floor				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	
	Massonry strength at mid height of wall				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	
	Massonry strength at underside floor				Sec.6.1.2.1(1) formula (6.1) EN 1996-1-1	Sec.11.1.2.1.1 formula (11.1)	
	Massonry strength for shear load at top floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at mid height of wall				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
	Massonry strength for shear load at underside floor				Sec. 6.2(1) formula (6.12) EN 1996-1-1	Sec. 11.2.1 formula (11.12)	
Local Strength of Reinforced Structures	Bearing deformation under the local load	Sec. 4.13 of SNiP II-22-81	Sec. 7.13 of SP 15.13330.2012	Sec. 7.13 of SP 15.13330.2020	Sec. 6.1.3(2) EN 1996-1-1	Sec. 11.1.3.2	Sec. 4.13 of KMK 2.03.07-98

STRUCTURES UNDER RECONSTRUCTION							
Mode	Factors checked	Sections of design codes					
		SNiP II-22-81	SP 15.13330 .2012	SP 15.13330.2020	EN 1996-1-1	DBN B.1.2.2-2006	KMK 2.03.07-98
Axially Compressed Columns Reinforced by Battens	Stability under axial compression	Sec. 5.38 of Manual [54], Sec. 5.45 of Guide [67]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
Eccentrically Compressed Columns Reinforced by Battens	Stability in the eccentricity plane under eccentric compression	Sec. 5.38 of Manual [54], Sec. 5.45 of Guide [67]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
	Stability out of the eccentricity plane under axial compression	Sec. 5.38 of Manual [2], Sec. 5.45 of Guide [4]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
Building Wall Reinforced by Battens	Stability under eccentric compression of the cross-section under a floor slab	Sec. 5.38 of Manual [2], Sec. 5.45 of Guide [4]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
	Stability under eccentric compression of the middle cross-section	Sec. 5.38 of Manual [2], Sec. 5.45 of Guide [4]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
	Stability under eccentric compression of the lower cross-section	Sec. 5.38 of Manual [2], Sec. 5.45 of Guide [4]	Sec. 8.25 of SP 427.1325800.2018	Sec. 8.25 of SP 427.1325800.2018			
Opening in a Wall	Stability of a lintel	Sec. 4.7 of SNiP II-22-81	Sec. 7.7 of SP 15.13330.2012	Sec. 7.7 of SP 15.13330.2020			Sec. 4.7 of KMK 2.03.07-98

STRUCTURES UNDER RECONSTRUCTION								
Mode	**Factors checked**	**Sections of design codes**						
		SNiP II-22-81	**SP 15.13330 .2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
	Normal stresses in a steel lintel	Sec. 5.12 of SNiP II-23-81*	Sec. 8.2.1 of SP 16.13330	Sec. 8.2.1 of SP 16.13330			Sec. 7.12 of KMK 2.03.05-97
	Tangential stresses in a steel lintel	Sec. 5.12 of SNiP II-23-81*	Sec. 8.2.1 of SP 16.13330	Sec. 8.2.1 of SP 16.13330			Sec. 7.12 of KMK 2.03.05-97
	Deflection of a steel lintel	Sec. 10.1 of SNiP 2.01.07-85*	Sec. 15.1 of SP 20.13330	Sec. 15.1 of SP 20.13330			Sec. 10.1 of KMK 2.01.07-96
	Local strength under the support of a steel lintel	Sec. 4.13 of SNiP II-22-81	Sec. 7.13 of SP 15.13330. 2012	Sec. 7.13 of SP 15.13330.2020			Sec. 4.13 of KMK 2.03.07-98

SUPPORT JOINTS							
Mode	**Factors checked**	**Sections of design codes**					
		SNiP II-22-81	**SP 15.13330.2012**	**SP 15.13330.2020**	**EN 1996-1-1**	**DBN B.1.2.2-2006**	**KMK 2.03.07-98**
Hanging Walls	Bearing deformation of masonry above the support of the foundation beam	Sec. 4.13, 6.48, 6.51 of SNiP II-22-81 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.48, 9.51 of SP 15.13330.2012 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.54, 9.57 of SP 15.13330.2020 Sec. 14.2 of Designer’s Manual [5]			Sec. 4.13, 6.48, 6.51 of KMK 2.03.07-98
	Bearing deformation of masonry above the left support of the foundation beam	Sec. 4.13, 6.48, 6.51 of SNiP II-22-81 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.48, 9.51 of SP 15.13330.2012 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.54, 9.57 of SP 15.13330.2020 Sec. 14.2 of Designer’s Manual [5]			Sec. 4.13, 6.48, 6.51 of KMK 2.03.07-98
	Bearing deformation of masonry above the right support of the foundation beam	Sec. 4.13, 6.48, 6.51 of SNiP II-22-81 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.48, 9.51 of SP 15.13330.2012 Sec. 14.2 of Designer’s Manual [5]	Sec. 9.54, 9.57 of SP 15.13330.2012 Sec. 14.2 of Designer’s Manual [5]			Sec. 4.13, 6.48, 6.51 of KMK 2.03.07-98

Bearing of Beams and Slabs on a Wall	Bearing deformation of masonry under the slab support Bearing deformation of masonry above the slab Bearing deformation of masonry under the beam support Bearing deformation of masonry above the beam Bearing deformation of masonry under the lower packing plate Bearing deformation of masonry above the upper packing plate	Sec. 4.13 of SNiP II-22-81 Sec. 7.3 of Designer's Manual [5]	Sec. 9.46 of SP 15.13330.2012 п Sec. 7.3 of Designer's Manual [5]	Sec. 9.46 of SP 15.13330.2020 п Sec. 7.3 of Designer's Manual [5]			Sec. 4.13, 6.46 of KMK 2.03.07-98
Bearing of Beams and/or Trusses on Piers and Columns	Size of a distribution cushion	P.F. Vakhnenko § 5.6. [1]	P.F. Vakhnenko § 5.6. [1]	P.F. Vakhnenko § 5.6. [1]			
	Axial compression of a cross-section below the bearing area	Sec. 6.44 of SNiP II-22-81	Sec. 9.44 of SP 15.13330.2012	Sec. 9.44 of SP 15.13330.2020			Sec. 6.44 of KMK 2.03.07-98
	Tension of the masonry seams below the bearing area	P.F. Vakhnenko § 5.6. [1]	P.F. Vakhnenko § 5.6. [1]	P.F. Vakhnenko § 5.6. [1]			
	Tension of the reinforcement below the bearing area						
	Bearing deformation of masonry under the supported structure	Sec. 6.44 of SNiP II-22-81	Sec. 9.44 of SP 15.13330.2012	Sec. 9.44 of SP 15.13330.2020			Sec. 6.44 of KMK 2.03.07-98

8.15 References

[1] P.F.Vakhnenko, Masonry and reinforced masonry structures. — Kiev, "Budivelnik" Publ., 1990. — 184 p.

[2] Reference manual on design of masonry and reinforcement masonry structures (supplement to SNiP II-22-81). V.Kucherenko Centr. Res. Inst. for Structural Constructions, USSR State Comm. for Construction, Moscow, "Stroyizdat" Publ., 1989. — 185 p.

[3] Recommendations on reinforcing of masonry structural components. V.Kucherenko Centr. Res. Inst. for Structural Constructions, USSR State Comm. for Construction, Moscow, "Stroyizdat" Publ, 1984. — 37 p.

[4] Guide to design of masonry and reinforcement masonry structures (supplement to SNiP II-V.2-71). V.Kucherenko Centr. Res. Inst. for Structural Constructions, USSR State Comm. for Construction, Moscow, "Stroyizdat" Publ., 1974. — 183 p.

[5] Designer's manual. Masonry and reinforced masonry structures. — Moscow, "Stroyizdat" Publ., 1968. — 175 p.

9. COMET

9.1 General Information

COMET is used to check and design the most common types of joints of steel bar structures used in civil and industrial engineering. The application enables to perform checks for compliance with requirements of one of the following design codes:

- SNiP II-23-81*,
- ShNK 2.03.05-13,
- SP 53-102-2004,
- SP 16.13330.2011 (revised and updated edition of SNiP II-23-81*),
- SP 16.13330.2017 (revised and updated edition of SNiP II-23-81*),
- DBN B.2.6-163:2010,
- DBN B.2.6-198:2014,
- EN 1993-1-1:2005 and EN 1993-1-8: 2005,
- SNiP RK 5.04-23-2002.

and to design a steel structural joint based on a particular prototype.

Unlike invention, the prototype-based engineering involves using available designs. This approach is implemented in **COMET**, and is based on selecting from a set of parameterized standard structural designs of joints (prototypes). Parameters of a prototype depend on the specified design conditions (material, internal forces etc.), and cannot be determined independently because certain relationships usually exist between them.

COMET uses the above approach and thus enables the engineer to improve the efficiency of his work by providing him with a wide range of prototypes. In this way the highly qualified personnel does not have to do the routine technical work of checking and correcting a multitude of parameters to comply with building codes and design specifications.

Once a design is selected for the joint, the application enables to determine all its parameters, which must comply with the standard requirements, a number of structural constraints, and the assortment of steel members. Both the standard requirements and structural constraints provided in the codes are obligatory, and their violation is not allowed. However, there are also certain design constraints violating which would cause only a warning, and the application can generate a solution with such violations.

The initial data for computer-aided design of steel structural joints include a configuration or type of the joint, type and sizes of cross-sections of bearing members connected in this joint, and internal forces acting in these members for an arbitrary number of design combinations of loadings.

On the **Design** stage (after clicking the respective button) the software performs the selection of a rational design of the joint which satisfies all the requirements and a set of structural and assortment limitations. The user can either accept the suggested solution or modify it according to his preferences, in order to take into account:

- the technology used to manufacture the structure;
- requirements of unification of the design within the framework of a project or other accepted limits of application (design organization, manufacturing plant etc.);
- the usage of standard designs commonly applied in the enterprise;
- the quality control system, the accepted marking system etc.

On the **Calculate** stage (after clicking the respective button) a check of the design of the joint is performed and a drawing (a sketch of the joint design with all its parameters close to the MS (metal structures) stage) is generated. In order to be able to modify the design thus generated, or to alter the format of representation of drawings (e.g., dimensioning, legends etc.), the graphical results can be exported as a DXF (AutoCAD) file.

All design modes of the application (except for the **Beam-To-Column Joints** mode) assume that all members connected in the joint and all auxiliary members of the joint (gussets, stiffeners, angle cleats etc.) are made of the same steel.

Check and design of joints are usually performed for the action of several load cases or their combinations, which can be specified by the user. It should be noted that the sequence of specification of the load combinations can in some cases affect the result of the design.

According to their structure, prerequisites, checks, structural limitations and recommendations SNiP II-23-81*, ShNK 2.03.05-13, SP 53-102-2004, SP 16.13330.2011, SP 16.13330.2017, DBN B.2.6-163:2010 and DBN B.2.6-198:2014 are quite close documents. Since the set of problems which can be solved in **COMET** is the same for these documents, all the general references to SNiP II-23-81* found in the text should be treated as similar references to ShNK 2.03.05-13, SP 53-102-2004, SP 16.13330, DBN B.2.6-163:2010 or DBN B.2.6-198:2014.

A separate chapter of this book (see Section 9) is devoted to the detailed description of the **COMET** branch which implements the requirements of EN 1993-1-1:2005 and EN 1993-1-8: 2005.

9.2 Decision-Making Algorithm

The software implementation of the automatic selection of the unknown parameters of the given type of the joint structure has been reduced to the problem of **making a decision** on the basis of the analysis of a mathematical model of the joint structure. Below you will find some additional commentaries given in order to define a number of general principles for decision making, which are implemented in the software and thus have to be taken into account by the users.

A final combination of *internal* (integrated) parameters was determined for each group of joints (column bases, beam splices, beam-to-column joints) $\vec{P} = \{P_n\}$, $n = \overline{1, N_P}$. A set of *controlled* parameters $\vec{X} = \{X_i\} \subset \vec{P},\ i = \overline{1, N_X}$, and a set of *subordinated* parameters $\vec{Y} = \{Y_j\} \subset \vec{P},\ j = \overline{1, N_Y}$, $\vec{P} = \vec{X} \cup \vec{Y}$ were distinguished among them. A specific feature of the design object (structural joint) is that its controlled parameters can be both independent and dependent on each other (for example, the end-plate thickness and the diameter of bolts in end-plate joints), and it is difficult to represent this dependence analytically, while the subordinated parameters can always be determined (calculated) unambiguously depending on the values of the controlled parameters, $\vec{Y} = \mathbf{\Theta}\{\vec{X}\}$. For example, if we consider an erection beam splice with end-plates, the end-plate thickness, the diameter of bolts and the number of bolt rows will be the controlled parameters, and the other parameters of the joint (width and height of the end-plate, sizes of the stiffeners etc.) will be the subordinated ones.

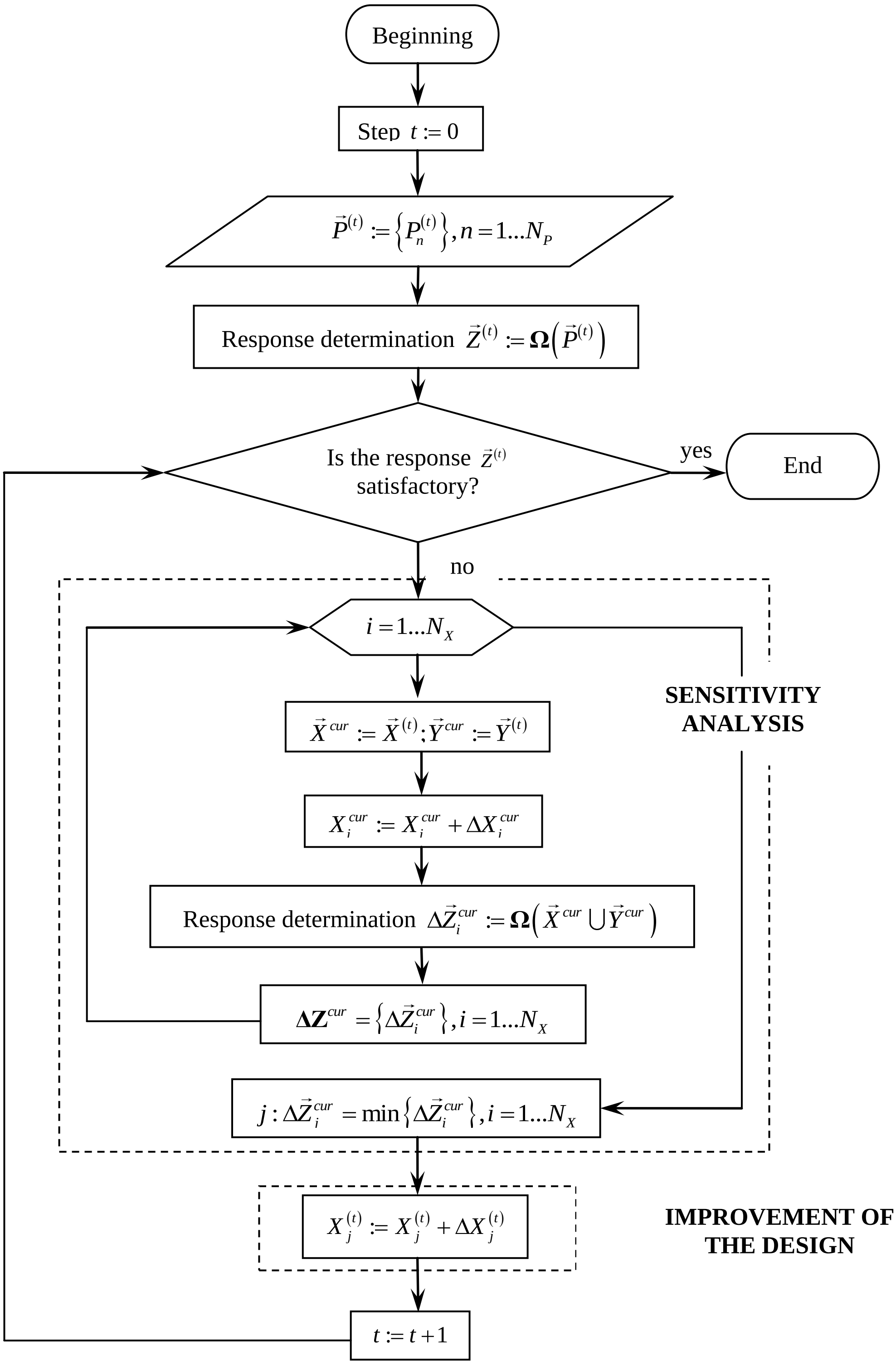


Figure 9.2-1. *A flowchart of the decision making algorithm when selecting the parameters of the structural joint*

The *parameters of state* (output) have been also included in the mathematical model of the structural joint $\vec{Z} = \{Z_k\}, k = \overline{1, N_Z}$, $k = \overline{1, N_Z}$ – values which characterize the integral properties of the design object. The parameters of state can only be calculated: $\vec{Z} := \mathbf{\Omega}(\vec{P})$, but they cannot be directly varied. They are unambiguously dependent on the values of the internal parameters of the design object (structural joint). The **utilization factors of restrictions** of the load-bearing capacity of the structural elements of the joint defined by the standard requirements were considered as the parameters of state.

The selection of a certain design of the joint is the determination of the certain values of the whole set of its internal parameters.

When the decision was made, the values of the internal parameters varied within certain limits defined by the system of inequalities:

$$\begin{cases} \boldsymbol{\varphi}_1(\vec{P}) \le 0,\ \boldsymbol{\varphi}_1 = \{\psi_\kappa,\ \phi_\eta\}; \\ \boldsymbol{\varphi}_2(\vec{Z}(\vec{P})) \le 0,\ \boldsymbol{\varphi}_2 = \{\psi_\kappa,\ \phi_\eta\}; \end{cases} \quad \kappa = \overline{1, N_{EC}},\ \eta = \overline{1, N_{IC}} \tag{8.2-1}$$

which included:

- *constraints by bearing capacity* of the members included in the joint (strictly speaking the load-bearing capacity of bar members connected in the joint should be ensured before designing or checking the joint. The checks performed here are just additional main checks which ensure only the strength of members in the elastic stage of their behavior); these constraints are defined by the standard requirements;
- *constraints by assortments* of rolled structural steels and plates;
- *structural constraints*, which reflect the conditions of manufacturing of joint members, constraints imposed on the mutual arrangement of members by the possibility of making welded and bolted connections, conditions of weldability of members of different thickness, and others.

The automatic determination of the unknown values of the internal parameters of the joint design is implemented as a targeted iterative improvement of a certain initial joint design toward the fulfillment of a set of constraints by bearing capacity taking into account the structural constraints and assortment constraints. The iterative improvement of the design is performed on the basis of the **analysis of its sensitivity** with respect to the variation of the controlled parameters in the structure of the joint. The user can fix some internal parameters of the joint (both controlled and subordinated ones). In this case the selection of the design of the joint will be performed for the fixed values of the user-defined parameters of the joint.

The response of the system (values of the utilization factors of restrictions of the load-bearing capacity) is evaluated at each variation of a certain controlled parameter. Finally, the decision to increase the controlled parameter the variation of which has provided the “best” (in terms of satisfying the restrictions of the bearing capacity) design of the joint structure is made.

Assortment constraints are taken into account both on the stage of determining the starting values of the internal parameters of the joint structure and on the stage of their variation (increasing) which is provided strictly according to the assortments of rolled structural steels and plates.

Structural constraints (which include the conditions of manufacturing structural members, constraints imposed on the mutual arrangement of members by the possibility of making welded and bolted connections, conditions of weldability of members of different thickness, and others) were described by the functional relationships which connected the subordinated parameters of the joint structure with the controlled ones.

A flowchart of the decision making algorithm is given in Fig. 9.2-1.

In cases when it is impossible to obtain a joint design for the initial data specified by the user, which would comply with building codes, the application performs an analysis of the load-bearing capacity of the joint design, outputs the results of the analysis, and gives recommendations on how to improve its load-bearing capacity.

9.3 Main Window

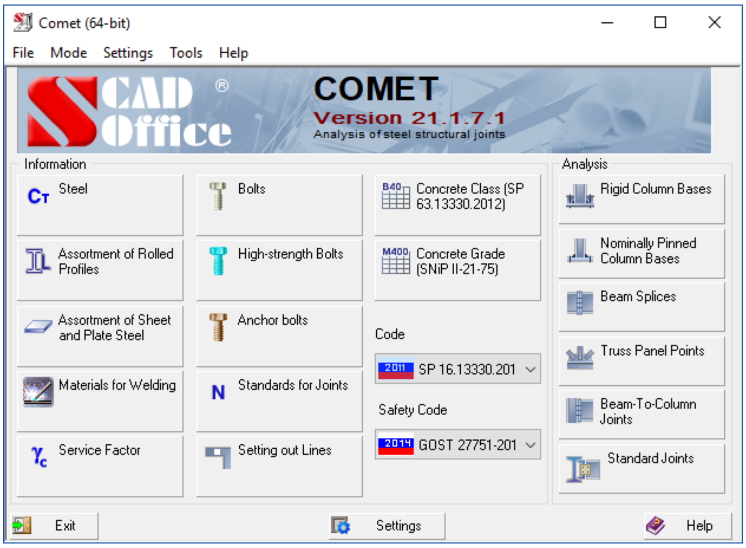


Figure 9.3-1. *The main window*

When the application is started, the first thing that appears on the screen is its main window (Fig. 9.3-1) with a set of buttons for selecting a working mode. These modes can be subdivided into two groups:

- *reference modes* which perform reference and auxiliary operations and are united in the **Information** group;
- *design modes* which enable to perform checks and design of steel structural joints and are united in the **Analysis** group.

A detailed description of each mode is given below. Their brief characteristic is given here.

The reference modes include:

- **Steel —** implements recommendations of building codes on selection of steel grades (Table 50* of SNiP II-23-81* and Annex C of SP 53-102-2004, SP 16.13330, and Annex F of DBN B.2.6-163:2010, and Annex D of DBN B.2.6-198:2014). The mode also gives information on the relation between steel grades defined by SNiP II-23-81* (GOST 27772-88*) to steel grades defined by GOST or standard specifications (Table 51, b of SNiP II-23-81*), and provides reference data on the design resistance according to the tables of the respective code;
- **Assortment of Rolled Profiles** – enables to browse through assortments of structural steel;
- **Assortment of Sheet and Plate Steel —** provides information on assortments of sheet and plate steel;
- **Materials for Welding** — implements requirements of the sections of implemented codes for selection of materials for welded joints and for selection of the design strength values for welded joints;
- **Service Factor** — is used to browse and select values of service factors for steel structural members and welded connections (γ_c), as well as those for bolted connections and connections with high-strength bolts (γ_b) according to the implemented codes;
- **Bolts** — enables to browse through an assortment of bolts with their recommended classes, which depend on the specified conditions of operation of the structure and the expected performance conditions of the bolted connection.
- **High-Strength Bolts —** provides information on the structure, sizes, and mechanical properties of high-strength bolts, and also lists nuts and washers for such bolts specifying their sizes and geometric properties;
- **Anchor Bolts —** provides information on the types of anchor bolts and on their design tensile strength;
- **Standards for Joints —** provides properties of full-strength joints between rolled members which are connected by welding with packings;

- **Setting out Lines —** provides information on the recommended positioning of bolt holes in rolled profiles;
- **Concrete Class** — provides the characteristic strength of concrete, and the design strength at the ultimate and serviceability limit states depending on the compressive strength class of concrete according to SNiP 2.03.01-84*;
- **Concrete Grade** — enables to browse the design and characteristic values of strength of various grades of concrete according to SNiP II-21-75 , and provides data on the relation between classes and grades of concrete defined by GOST 26633-91.

The *design modes* include:

- **Rigid Column Bases** — enables to perform a check and design of joints of the column bases which provide a rigid column-to-foundation connection;
- **Nominally Pinned Column Bases** — enables to perform a check and design of nominally pinned joints of column bases;
- **Beam Splices** — enables to perform a check and design of beam splices with plates or end-plates;
- **Truss Panel Points** — enables to perform a check and design of truss joints;
- **Beam-To-Column Joints** — enables to perform a check and design of pinned and rigid joints between beams and columns;
- **Standard Joints** — enables to design standard joints between beams in the same level, which use bolts or an angle cleat.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

The main window also contains a number of buttons which are common controls for all working modes. These include the **Exit**, **Settings**, and **Help** buttons. The **Help** and **Exit** buttons perform functions common for a Windows application: providing reference information and finishing the current session, respectively.

The **Settings** button invokes the **Application Settings** dialog box where you can customize the program (see Section 2.2).

The **Menu** button switches from any mode to the main window.

9.4 Information Modes

9.4.1 Steel

This mode is used to select a steel grade for the designed structure and its members and contains three tabs: **Application Conditions** (Fig. 9.4.1-1), **Structural Steels and Plates** (Fig. 9.4.1-2, 9.4.1-4), and **Pipes** (Fig. 9.4.1-3, 9.4.1-5).

The **Application Conditions** tab (Fig. 9.4.1-1) contains six groups of data.

The choice is made for four groups of structures according to:

- Table 50* of SNiP II-23-81*;
- Annex C of SP 53-102-2004;
- Annex C of SP 16.13330;
- Annex F of DBN B.2.6-163:2010,
- Annex D of DBN B.2.6-198:2014.

The procedure for classifying a structure into a certain group is described in [2], DBN B.2.6-163:2010 and DBN B.2.6-198:2014.

a *b*

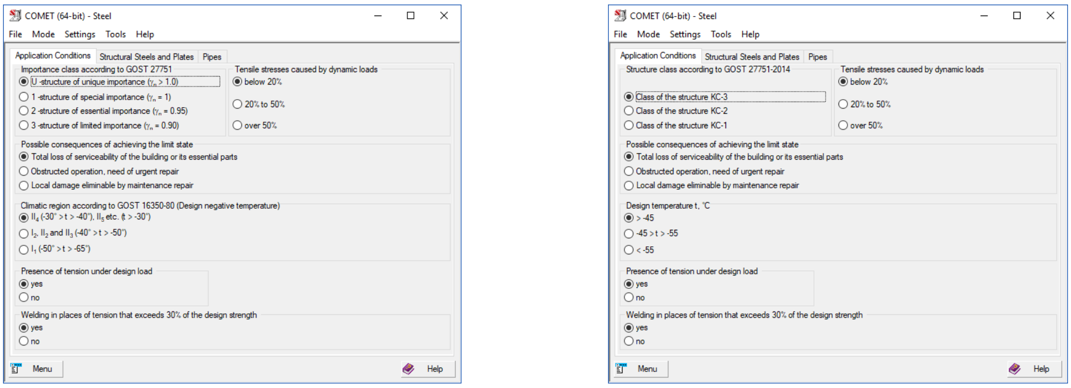


Figure 9.4.1-1. *The* **Application Conditions** *tab of the* **Steel** *dialog box*:
a – for SNiP II-23-81*; *b* – for SP 16.13330.

In the **Importance class according to GOST 27751-88** group you should select one of the four cases defined in the codes, keeping in mind that the importance factor, γ_n, for unique projects is, as a rule, defined individually for each particular project by an institution which gives an approval of the design. The default value for the objects of the said type is $\gamma_n = 1{,}2$.

The **Possible consequences of achieving the limit state** group enables to select one of the three importance classes of the considered structural member. The paper [2], DBN B.2.6-163:2010 and DBN B.2.6-198:2014 provide recommendations on how to classify some structures into these classes (Classes A, B, and C correspond to three grades of the considered group). This rating is accurate for DBN but can be used in other cases as well. Necessary extracts from these recommendations are given below.

Structures of service platforms and decks:
- main and secondary beams, girders of frames — A;
- metal sheeting — B.

Columns of industrial buildings and open crane trestles:
- columns of service platforms and decks — A;
- main members of the cross-section — A;
- main vertical bracings between columns — A;
- bracings with the stress below $0.4R_y$ — C.

Roofing structures:
- trusses, girders — A;
- skylight panels, roof panels, purlins, longitudinal bracings — B;
- other bracings — C.

Framework structures:
- girders under brick walls and above gates — A;
- columns, end and wind trusses — B;
- other members — C.

Auxiliary structures:
- stringers — A;
- landings, imposts, window and skylight casements — C.

Transporter galleries:
- span structures, column bracings — A;

- other bracings, roof beams, framework members — B.

Supports of power transmission lines and structures of outdoor switch gears (OSG):

- supports of power transmission lines, pylons for OSG switches — A;
- pylons for other OSG equipment — B.

Aerial equipment:

- trunks of masts and towers — A;
- diaphragms of towers, stairways, landings — B.

Vents and chimneys:

- chords and lattices of towers, shell of a separate pipe — A;
- flue-gas stacks, pipe shells with bracings — B;
- decks, mantles, stiffeners — C.

Cooling towers, water towers:

- chords of lattice towers, lattices — A;
- framework, decks, covering of cooling towers — C.

Bunkers, silos — A.

Other groups serve to characterize the stress state (**Presence of tension under design load** and **Tensile stresses caused by dynamic loads**), and to indicate the temperature mode of operation (**Climatic region according to GOST 16350-80**). The latter group requires that you indicate whether there is welding in the tension area (**Welding in places of tension that exceeds 30% of the design strength**).

If the analysis is performed according to SP 53-102-2004 or SP 16.13330 and the requirements of the enhanced fire resistance have to be taken into account according to Sec. 6.2 SP 53-102-2004 (Sec. 5.2 SP 16.13330), a special steel with enhanced fire resistance is selected.

Once you have filled all the data in the first tab, click on the **Structural Steels and Plates** or **Pipes** tab to open the respective tab of the dialog box.

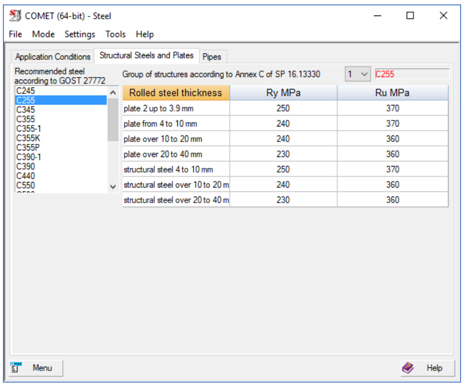


Figure 9.4.1-2. *The* **Structural Steels and Plates** *tab of the* **Steel** *dialog box*

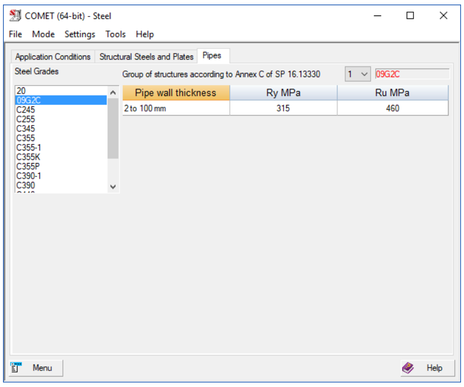


Figure 9.4.1-3. *The* **Pipes** *tab of the* **Steel** *dialog box*

When the design is performed according to SNiP, SP or DBN, these tabs (Fig. 9.4.1-2) contain a reference to a group of structures according to

- Table 50* of SNiP II-23-81*;
- Annex C of SP 53-102-2004;
- Annex C of SP 16.13330;
- Annex C of DBN B.2.6-163:2010,
- Annex A of DBN B.2.6-198:2014,

which conforms to operating conditions specified on the previous tab, a list of steel grades recommended for this group according to GOST 27772-88, and a list of steel grades according to other standards or codes which can be used instead of the recommended steel. This tab also provides reference data on the design strength based on the yield strength (R_y) and on the ultimate strength (R_u).

Steels are selected from one of the following groups: structural steels and plates and steels for hot rolled pipes selected according to the Table 51* of SNiP II-23-81* or according to the Table C6 of SP 53-102-2004, SP 16.13330, Table F.1 of DBN B.2.6-163:2010, Table D.1 of DBN B.2.6-198:2014.

A certain group of structures may require steel of a better grade than that recommended by SNiP or SP. Therefore the user can increase (by no means decrease!) the group of structures by selecting its number from the respective drop-down list. Obviously, the list of recommended steel grades will change as well.

It should be noted that the application does not list all the details which are obligatory to include in the order on steel, such as in notes to Tables 50* and 51, *b* of SNiP II-23-81* and notes to text and tables of the Annex C of SP (Annex F of DBN B.2.6-163:2010 and Annex D of DBN B.2.6-198:2014). Primary regulatory codes and specifications should be used to create an order.

9.4.2 Assortment of Rolled Profiles

This mode enables you to browse through steel profile assortments available in the database of **COMET**.

The dialog box of this mode contains a list of assortments, **Assortment of Rolled Profiles**, represented by a tree-like structure located on the left. When a profile type is selected, the information window displays a table with properties of the respective profiles (Fig. 9.4.2-1). Once you select a particular profile, this window will display the cross-section of this profile with its dimensions (Fig. 9.4.2-2).

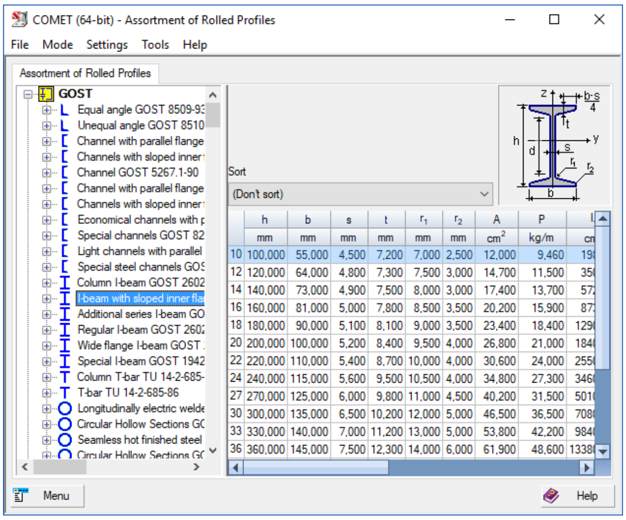


Figure 9.4.2-1. *The* **Assortment of Rolled Profiles** *dialog box*
(a profile type has been selected)

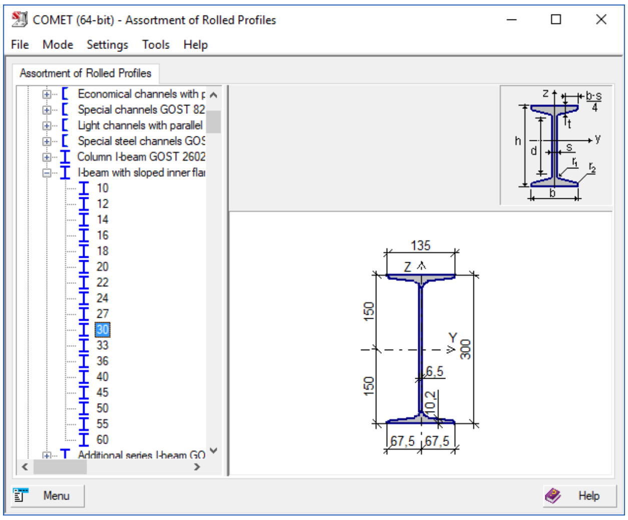


Figure 9.4.2-2. *The* **Assortment of Rolled Profiles** *dialog box*
(a particular profile has been selected)

9.4.3 Assortment of Sheet and Plate Steel

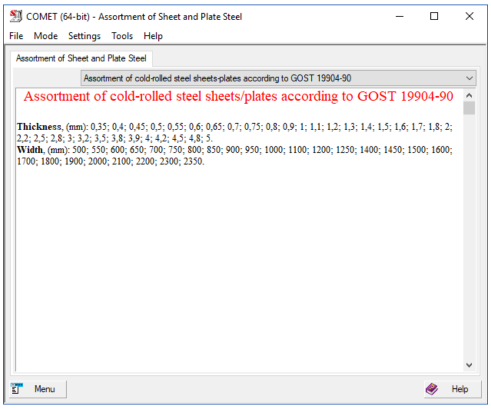


Figure 9.4.3-1. *The* **Assortment of Sheet and Plate Steel** *dialog box*

The **Assortment of Sheet and Plate Steel** mode (Fig. 9.4.3-1) enables to browse the assortments of hot-rolled sheets/plates according to GOST 19903-74, cold-rolled steel sheets/plates according to GOST 19904-90, universal hot-rolled steel wide strips according to GOST 82-70* and hot-rolled strips according to GOST 103-76*.

The **Assortment** drop-down list is used to select an assortment of sheet and plate steel. Once you have selected a particular assortment, the information window will display all the values of thickness and width available for the sheets or plates of this assortment.

9.4.4 Materials for Welding

This mode is used to select welding materials for the structure being designed. The selection procedure complies with Table 55* of SNiP II-23-81* and Sec. 6.4 and Annex D of SP 53-102-2004 (or Sec. 5.4 and Annex D of SP 16.13330, Annex G of DBN B.2.6-163:2010, Annex E of DBN B.2.6-198:2014).

The **Application Conditions** tab (Fig. 9.4.4-1) contains two groups of data.

The **Welded structure** group is used to specify the number of the group this structure belongs to according to Table 50 of SNiP II-23-81*, Annex C of SP 53-102-2004, SP 16.13330, Annex F of DBN B.2.6-163:2010, Annex D of DBN B.2.6-198:2014 or according to the results of the **Steel** mode, and steel the structure is made of.

a *b*

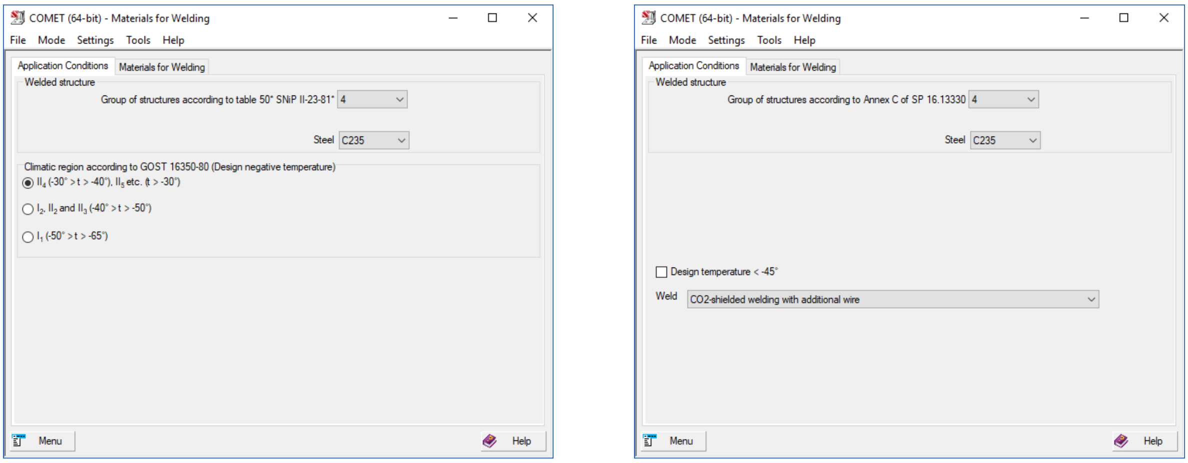


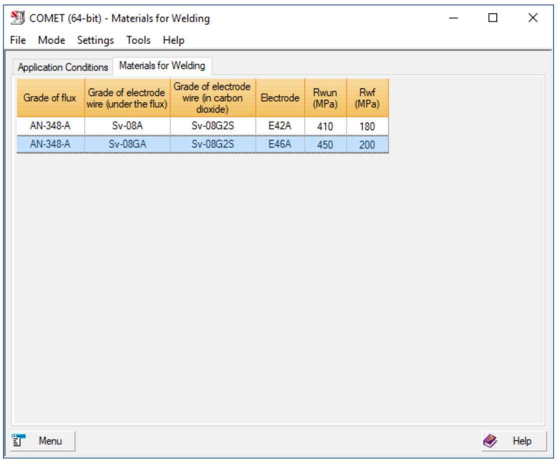
Figure 9.4.4-1. *The* **Application Conditions** *tab of the* **Materials for Welding** *dialog box*
a — according to SNiP II-23-81*, *b* — according to SP 16.13330.

The second group is used to specify the climatic region according to GOST 16350-80 when SNiP is selected, and the design temperature and the welding technology when SP is selected.

Once you have specified all the data in the first tab, open the next one by clicking on the **Materials for Welding** tab.

The **Materials for Welding** tab (Fig. 9.4.4-2) contains a list of recommended materials. If the **Materials for Welding** mode has been invoked from a design mode, then after clicking the **Apply** button the properties of the selected materials will be used in the analysis of structural members.

a



b

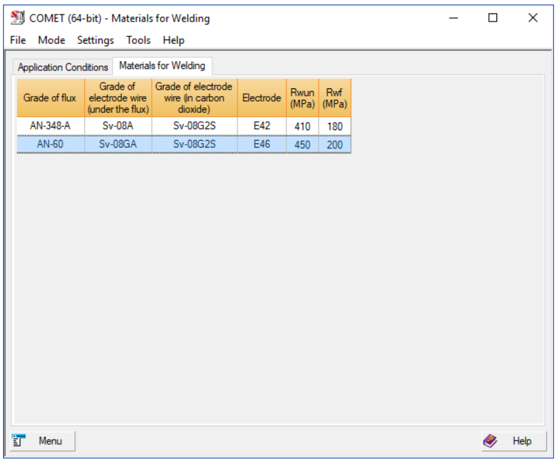


Figure 9.4.4-2. *The* **Materials for Welding** *tab of the* **Materials for Welding** *dialog box*
a — according to SNiP II-23-81*, *b* — according to SP 53-102-2004 or SP 16.13330

9.4.5 Service Factor

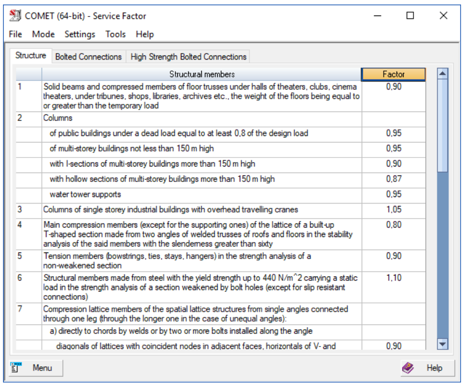


Figure 9.4.5-1. *The* **Structure** *tab of the* **Service Factor** *dialog box*

The dialog box of this mode (Fig. 9.4.5-1) contains data given in Table 6* of SNiP II-23-81* and Table 1 of SP 53-102-2004 (or SP 16.13330), Table 1.1.1 of DBN B.2.6-163:2010, Table 5.1 of DBN B.2.6-198:2014. In addition to the eleven items given in these tables, the dialog box contains additional cases implementing the instructions of the notes to these tables.

There are separate tabs with appropriate factors for bolted connections and high strength bolted connections. They can be opened by clicking on the respective tabs at the top of the dialog.

There is a separate tab for bolted connections (Fig. 9.4.5-2), which implements the requirements of Table 35* of SNiP II-23-81*, Table 38 of SP 53-102-2004, Table 41 of SP 16.13330, Table 1.12.4 of DBN B.2.6-163:2010, Table 16.4 of DBN B.2.6-198:2014.

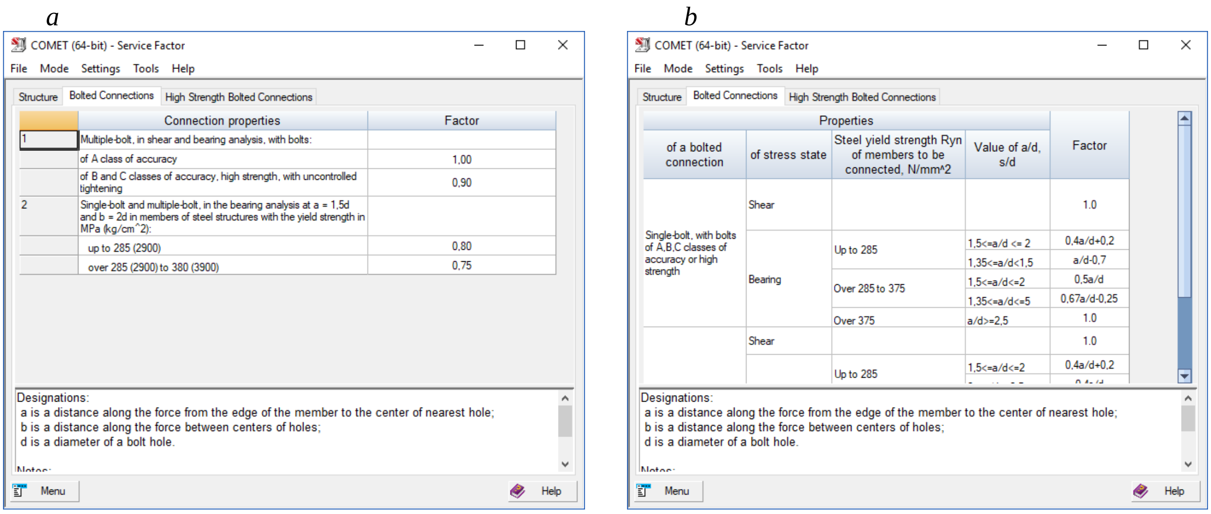


Figure 9.4.5-2. *The* **Bolted Connections** *tab of the* **Service Factor** *dialog box*
a — according to SNiP II-23-81*
b — according to SP 53-102-2004, SP 16.13330
DBN B.2.6-163:2010 or DBN B.2.6-198:2014.

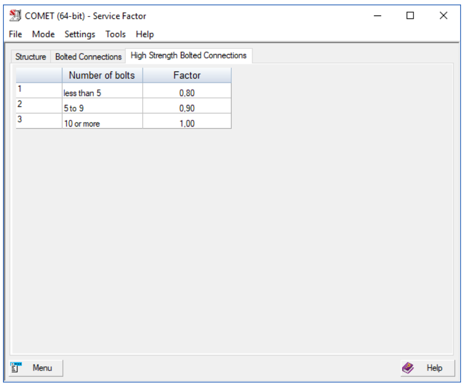


Figure 9.4.5-3. *The* **High Strength Bolted Connections** *tab of the* **Service Factor** *dialog box*

There is a separate tab for high strength bolted connections (Fig. 9.4.5-3), which implements the requirements of Sec. 11.13* of SNiP II-23-81*, Sec. 15.3.4 of SP 53-102-2004, Sec. 14.3.4 of SP 16.13330, Sec. 1.12.3.4 of DBN B.2.6-163:2010, Sec. 16.3.4 of DBN B.2.6-198:2014.

If the **Service Factor** mode has been invoked from a design mode, then after clicking the **Apply** button the selected value of the factor will be used in the analysis of structural members.

9.4.6 Bolts

The **Bolts** mode enables to browse through assortments of bolts with their recommended classes, which depend on the specified conditions of operation of the structure and the expected performance conditions of the bolted connection. Bolts are selected for four groups of structures according to Table 57* of SNiP II-23-81*, Annex D of SP 53-102-2004 or SP 16.13330, Annex C of DBN B.2.6-163:2010, and Annex A of DBN B.2.6-198:2014.

The **Service Conditions** tab (Fig. 9.4.6-1) contains three groups of data:

- **Type of structure**, where you can select one of two cases, defined by the codes, according to the requirements imposed on durability of the bolted joint;
- **Performance conditions of bolts**, where you can select a type of performance conditions for a bolt in a joint;
- **Climatic region according to GOST 16350-80 (Design temperature)**, where you can specify one of the temperature modes of operation defined by the codes (it is not used if DBN is selected).

Once you have specified all the data in the first tab, open the next one by clicking on the **Bolts** tab.

The **Bolts** tab (Fig. 9.4.6-2) contains a list of recommended bolt classes and the data on the assortment of bolts, which depends on the specified conditions of operation of the structure and the expected performance conditions of the bolted connection.

The **Standards and Codes** tab provides the user with a list of standards required for designing bolted joints.

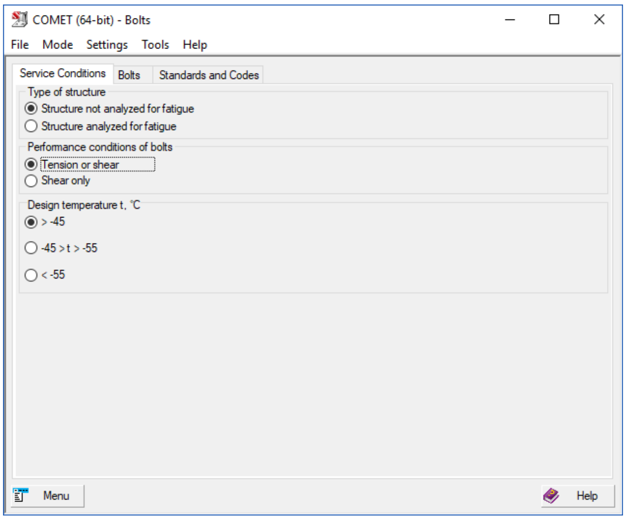


Figure 9.4.6-1. *The* **Service Conditions** *tab of the* **Bolts** *dialog box*

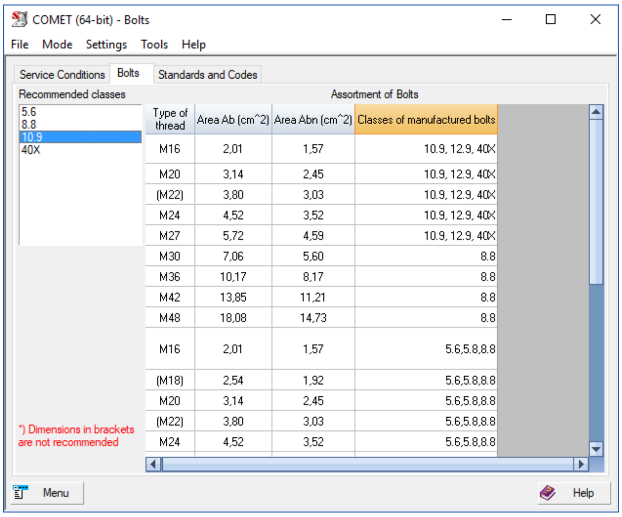


Figure 9.4.6-2. *The* **Bolts** *tab of the* **Bolts** *dialog box*

9.4.7 High-strength Bolts

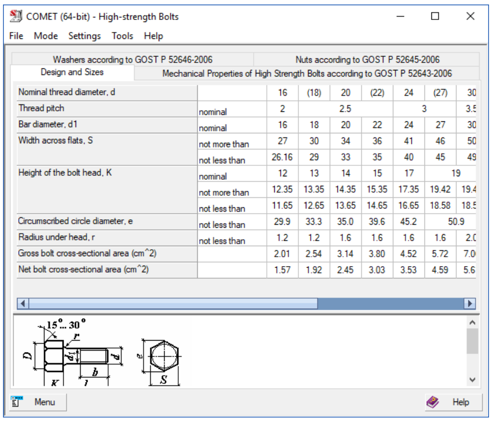


Figure 9.4.7-1. *The* **High-strength Bolts** *dialog box*

The **High-strength Bolts** mode (Fig. 9.4.7-1) provides information on the structure, sizes and mechanical properties of high-strength bolts (the **Design and Sizes** and **Mechanical Properties of High Strength Bolts according to GOST 22356-77*** tabs, respectively) which can be used in joints between steel structural members. Moreover, the **Nuts according to GOST 22354-77*** and **Washers according to GOST 22355-77*** tabs provide lists of nuts and washers, respectively, for such bolts specifying their sizes and geometric properties.

This information is given in a tabular form and is complemented by all the necessary drawings (Fig. 9.4.7-1).

9.4.8 Anchor Bolts

The **Anchor Bolts** mode provides information on the types of anchor bolts and on their design tensile strength.

The dialog box of this mode contains four tabs: **Anchor Bolts**, **Basic Dimensions**, **Steel Grades**, and **Design Tensile Strength**.

The **Anchor Bolts** tab (Fig. 9.4.8-1) provides graphic information about the types of anchor bolts. The **Basic Dimensions** tab (Fig. 9.4.8-2) contains tabular data on the geometric dimensions and the effective area of cross-sections depending on the type and diameter of anchor bolts. The **Steel Grades** tab (Fig. 9.4.8-3) provides information about steel grades used for anchor bolts depending on a climatic region of the construction site. The **Design Tensile Strength** tab (Fig. 9.4.8-4) provides values of the design tensile strength depending on the diameter and steel grade of anchor bolts.

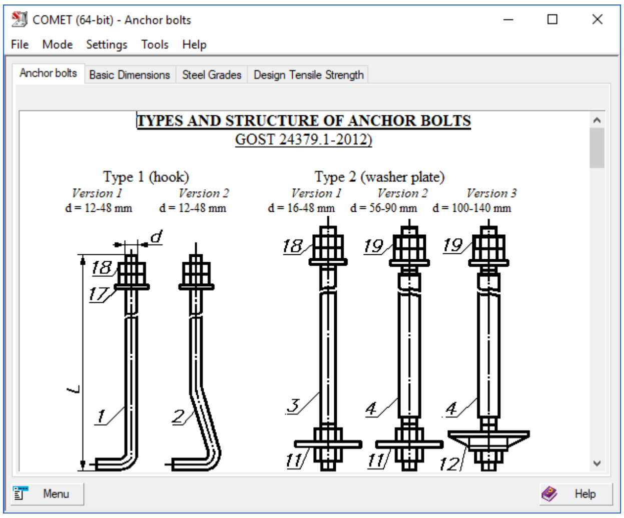


Figure 9.4.8-1. *The* **Anchor Bolts** *tab of the* **Anchor Bolts** *dialog box*

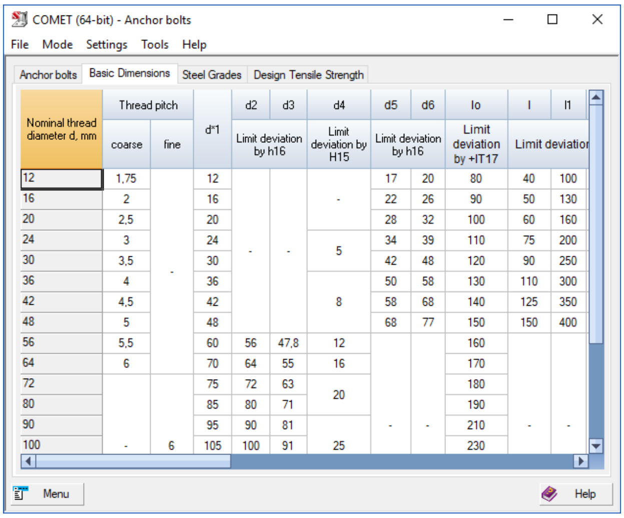


Figure 9.4.8-2. *The* **Basic Dimensions** *tab of the* **Anchor Bolts** *dialog box*

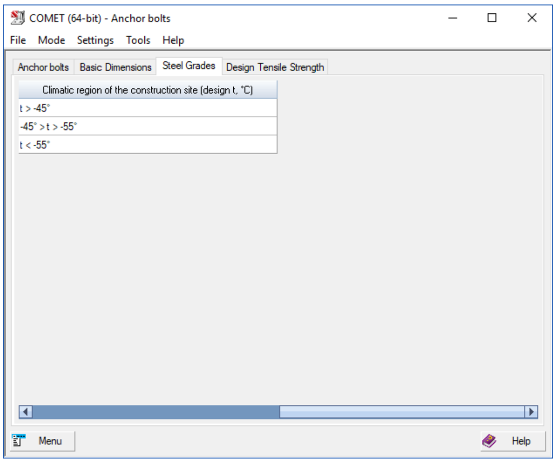


Figure 9.4.8-3. *The* **Steel Grades** *tab of the* **Anchor Bolts** *dialog box*

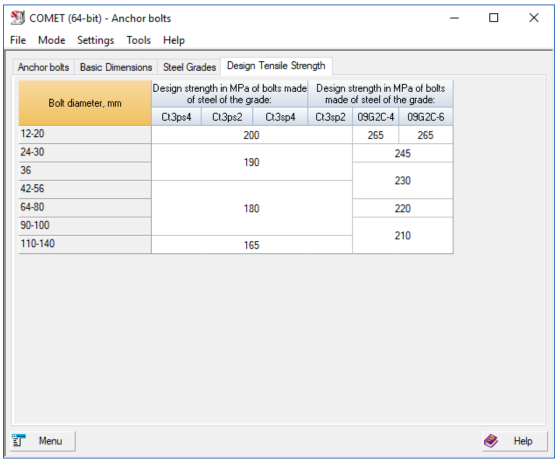


Figure 9.4.8-4. *The* **Design Tensile Strength** *tab of the* **Anchor Bolts** *dialog box*

9.4.9 Standards for Joints

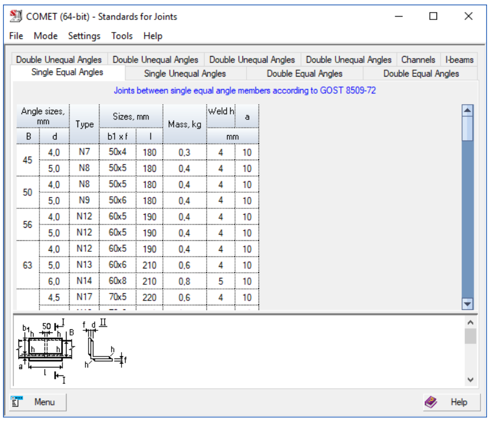


Figure 9.4.9-1. *The* **Standards for Joints** *dialog box*

The **Standards for Joints** mode (Fig. 9.4.9-1) provides properties of full-strength joints between rolled members which are connected by welding with packings [1].

The dialog box of this mode contains ten tabs which provide a full description of designs of joints between I-beam members according to GOST 8239-72, channel members according to GOST 8240-72, single and double equal angle members according to GOST 8509-72, single and double unequal angle members according to GOST 8510-72.

9.4.10 Setting out Lines

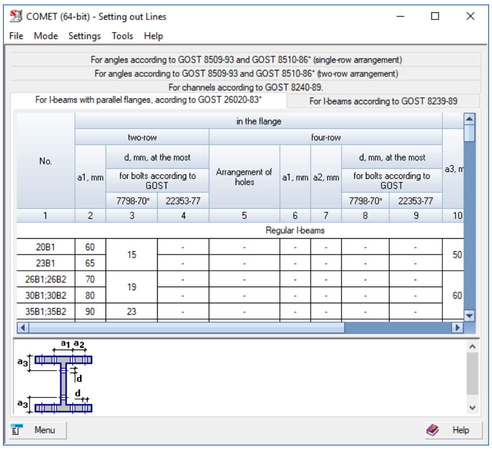


Figure 9.4.10-1. *The* **Setting out Lines** *dialog box*

The **Setting out Lines** mode (Fig. 9.4.10-1) provides information on the recommended positioning of bolt holes in rolled profiles [4]. This information is given in tabular form for angles according to GOST 8509-93 and GOST 8510-86* (for single- and two-row arrangement of bolt holes), for channels according to GOST 8240-89, for I-beams according to GOST 8239-89 and for I-beams with parallel flanges according to GOST 26020-83*. The drawing of a profile with the designations of dimensions assumed in the table is given in the information field at the bottom of the dialog box.

9.4.11 Concrete Class

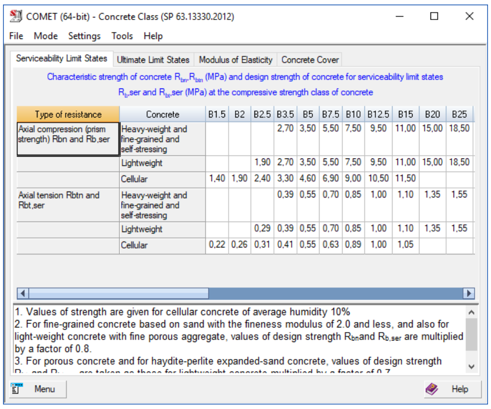


Figure 9.4.11-1. *The* **Concrete Class (SNiP 2.03.01-84*)** *dialog box*

The **Concrete Class (SNiP 2.03.01-84*)** mode (Fig. 9.4.11-1) enables to select a value for the characteristic strength of concrete and for the design strength of concrete at the ultimate limit state (the **Ultimate Limit States** tab) and the serviceability limit state (the **Serviceability Limit States** tab) depending on the compressive strength class of concrete according to SNiP 2.03.01-84*, as well as information on the minimum allowable concrete cover.

9.4.12 Concrete Grade

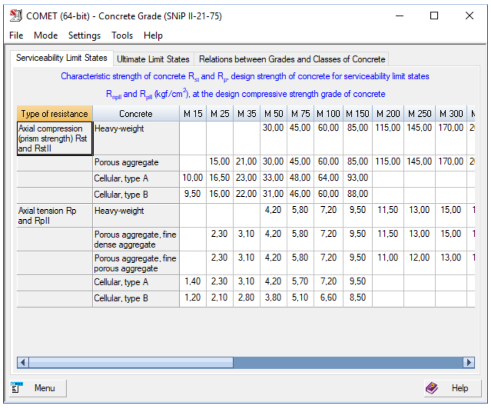


Figure 9.4.12-1. *The* **Concrete Grade (SNiP II-21-75)** *dialog box*

The **Concrete Grade (SNiP II-21-75)** mode (Fig. 9.4.12-1) enables to obtain information about the characteristic and design strength of concrete at the ultimate limit state (the **Ultimate Limit States** tab) and at the serviceability limit state (the **Serviceability Limit States** tab) depending on the compressive strength grade of concrete according to SNiP II-21-75.

The **Relations between Grades and Classes of Concrete** tab provides data on the relation between the compressive strength grades and classes of concrete according to GOST 26633-91.

9.5 Design Modes

9.5.1 Rigid Column Bases

The **Rigid Column Bases** mode enables to design and check the load-bearing capacity of joints of the column bases which provide a rigid column-to-foundation connection. This mode comprises a wide range of designs for this type of joints, such as:

- column bases without wing plates and cantilever stiffeners (Fig. 9.5.1-1);
- column bases with wing plates and cantilever stiffeners (Fig. 9.5.1-2);
- column bases with external wing plates (Fig. 9.5.1-3).

There are no additional details reinforcing the column base in the designs of the first four types of column bases. Such bases are usually designed for buildings without cranes and for buildings with small capacity cranes. Column bases with wing plates and anchor bolts have additional structural elements (wing plates and cantilever stiffeners) which provide a more uniform distribution of stresses in concrete of the foundation under the base plate.

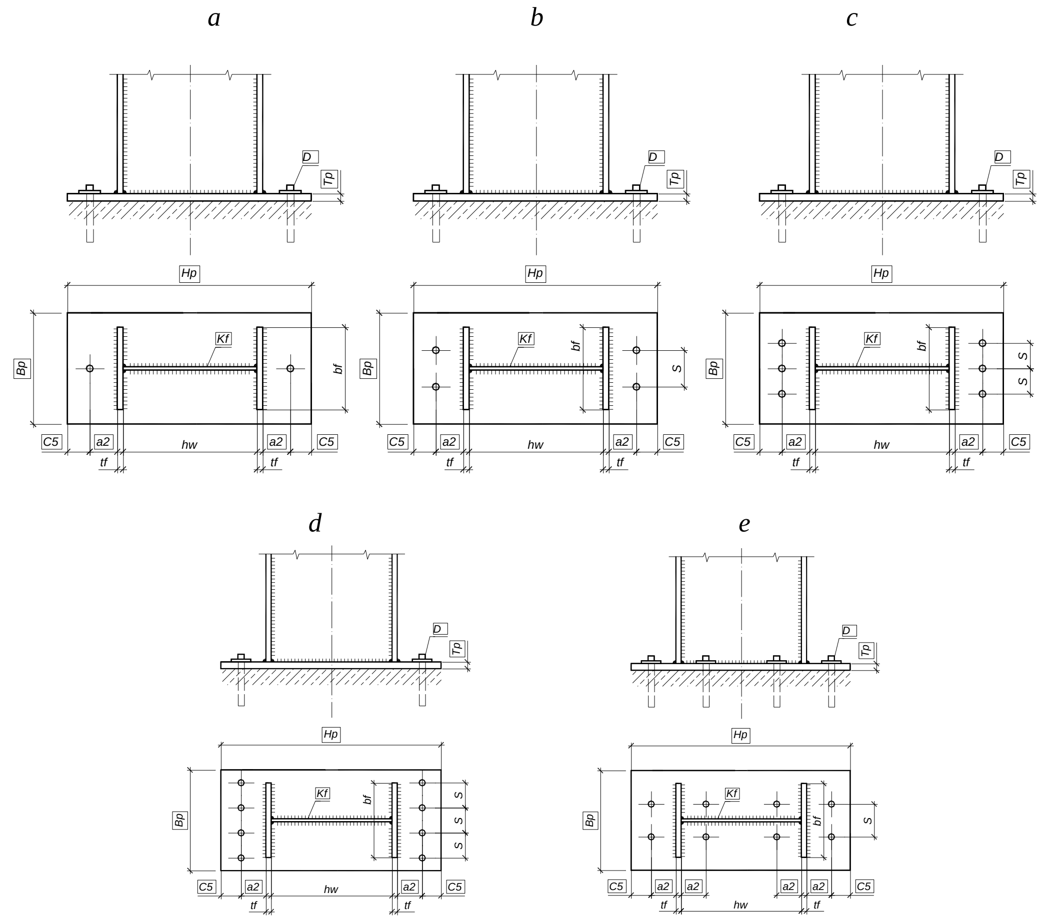


Figure 9.5.1-1. Types of designs for the joint of rigid column bases without wing plates and cantilever stiffeners

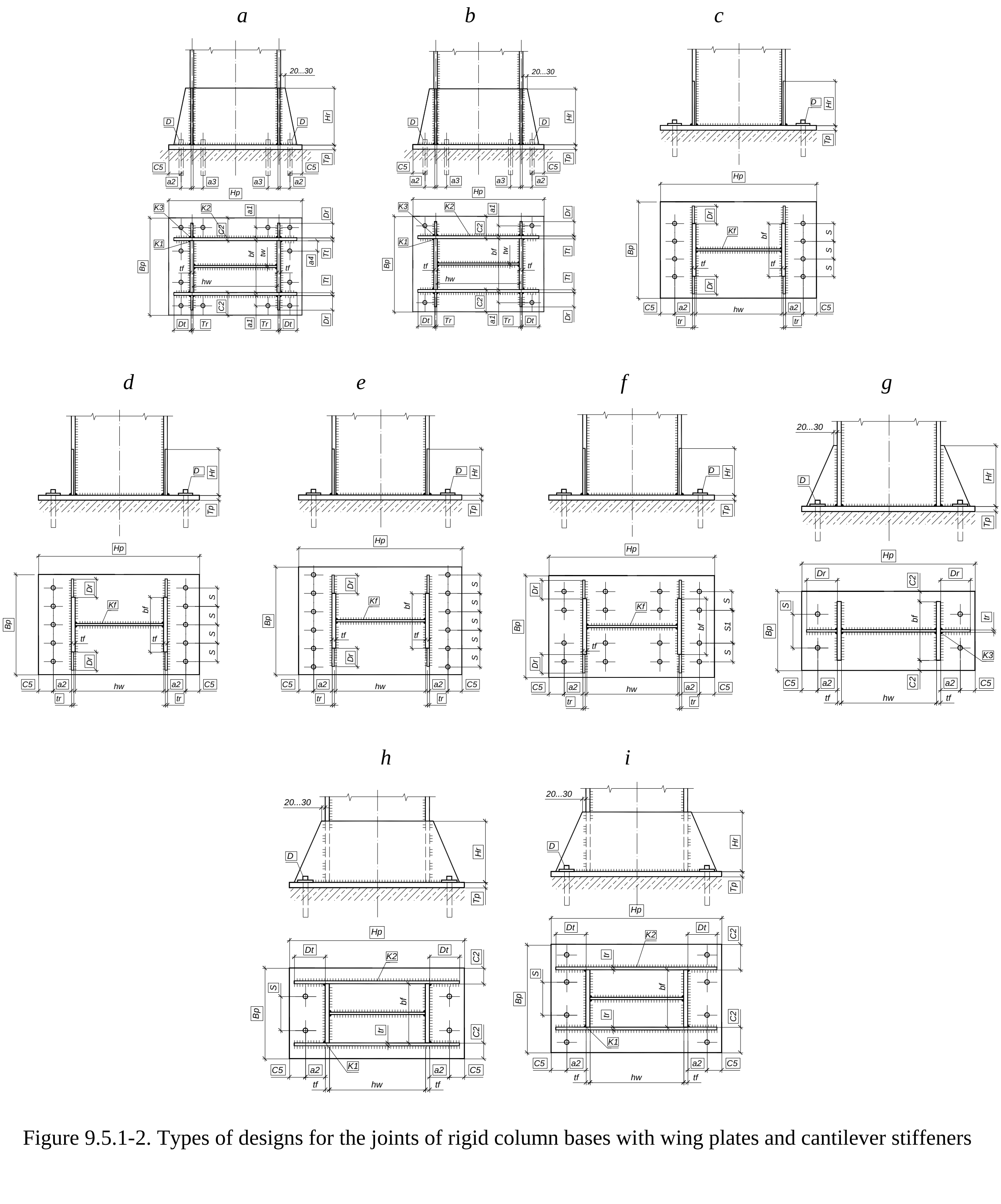


Figure 9.5.1-2. Types of designs for the joints of rigid column bases with wing plates and cantilever stiffeners

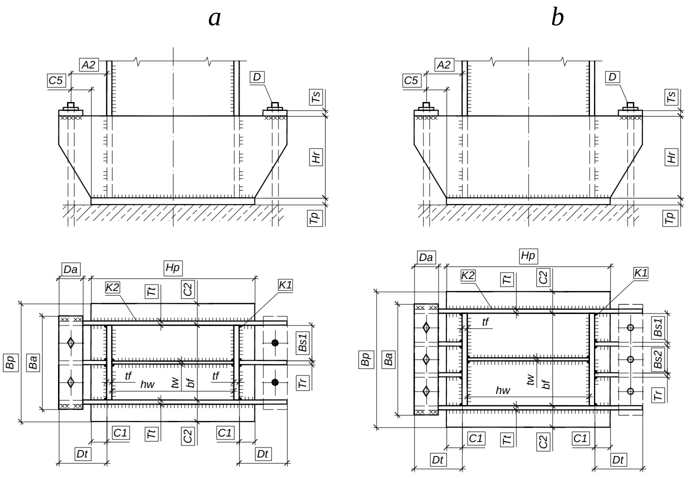


Figure 9.5.1-3. Types of designs for the joints of rigid column bases with external wing plates

This mode performs the following checks in compliance with the selected building code:

- resistance of the structural members of the column base joint (base plate, anchor bolts, anchor plates, wing plates and cantilever stiffeners, foundation concrete in local bearing);
- resistance of welded connections (between the column and the base plate, between the wing plate and column flanges, between the wing plate and the base plate, between the cantilever stiffener and column flanges, between the cantilever stiffener and the wing plate);
- a number of structural and assortment constraints.

The **Rigid Column Bases** dialog box contains the following tabs: **Configuration** (Fig. 9.5.1-4, 9.5.1-5), **Forces** (Fig. 9.5.1-9), **Structure** (Fig. 9.5.1-10), **Welding**, **Drawing** (Fig. 9.5.1-11) and **Interaction Curves** (Fig. 9.5.1-12).

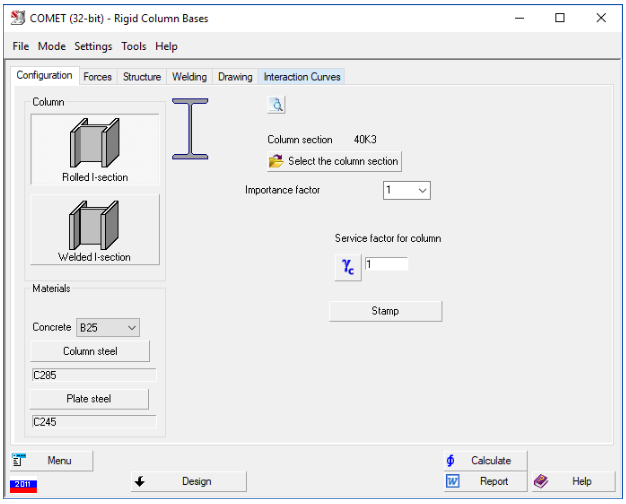


Figure 9.5.1-4. *The* **Configuration** *tab of the* **Rigid Column Bases** *dialog box* (a rolled I-section is selected as the column cross-section type)

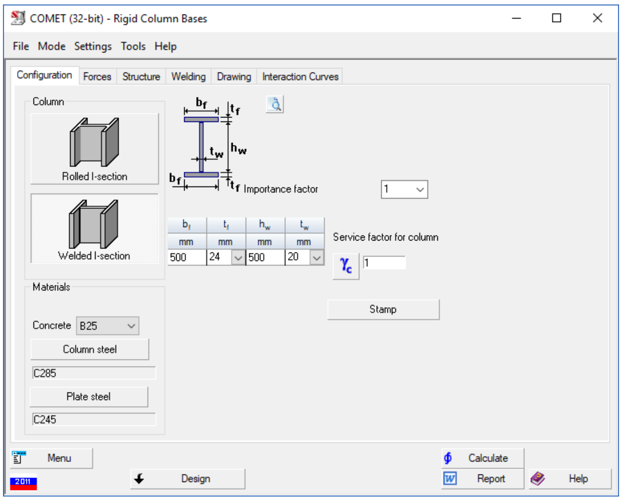


Figure 9.5.1-5. *The* **Configuration** *tab of the* **Rigid Column Bases** *dialog box* (a welded I-section is selected as the column cross-section type)

The user has to specify the initial data for the calculation in the **Configuration** tab (the column section, steel grade for the column, class of concrete of the foundation, service factor and importance factor). The type of the column section can be selected by clicking the respective button: **Rolled I-section** or **Welded I-section**. The interface of this tab depends on this choice (Fig. 9.5.1-4, Fig. 9.5.1-5).

If a rolled I-section is selected as the column cross-section type, you then have to select an assortment and the profile number in this assortment in the **Select profile** dialog box (Fig. 9.5.1-6), which can be invoked by clicking the **Select the column section** button.

When a welded I-section is selected as the column cross-section type, you have to specify the sizes of the column cross-section: the height, h_w, and the thickness, t_w, of the web; the width, b_f, and the thickness, t_f, of the flanges. The sizes of the column cross-section should be entered in the table in millimeters (Fig. 9.5.1-5). It should be noted that the thickness of the flanges and of the web can be either entered manually or selected from the drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel. The column cross-section can be checked in the **Preview** window (Fig. 9.5.1-7), which can be invoked by clicking the **Preview** button ( ).

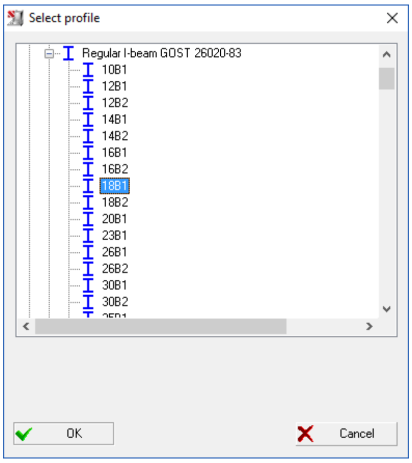


Figure 9.5.1-6. *The* **Select profile** *dialog box*

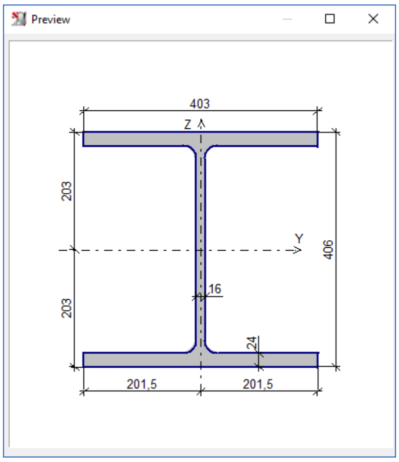


Figure 9.5.1-7. *The* **Preview** *window*

Materials used for design and analysis of a rigid column base joint can be defined by clicking the respective buttons, **Column steel** (enables to specify steel for the column) and **Plate steel** (enables to specify steel for the support base plate, wing plates and cantilever stiffeners). The **Concrete** drop-down list enables to select concrete classes for the foundation (see Fig. 9.5.1-4, 9.5.1-5).

You can enter the service factor for column in the respective text field, or it can be selected in the **Service Factor** dialog box (see Sec. 9.4.5), after clicking the nearby button ( ).

The service factor of the base plate of the column base is calculated automarically in this mode. The user has to specify the service factor not for the base plate but for the column.

The importance factor which will be further multiplied by the design values of all internal forces for all design combinations of loadings acting in the column base section has to be specified in the **Importance factor** drop-down list in this tab. If the values of the internal forces in the column base section have been obtained based on the results of the analysis of the system accounting for the importance factor (for example, when the design values of the applied loads were obtained taking into account this factor), the value equal to one has to be selected in the **Importance factor** drop-down list.

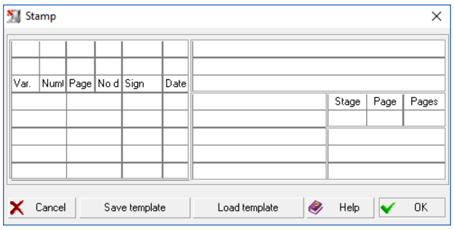


Figure 9.5.1-8. *The* **Stamp** *dialog box*

Clicking the **Stamp** button opens a **Stamp** dialog box, which enables to fill in the stamp of the drawing (Fig. 9.5.1-8), used in the draft of the design of the rigid column base joint. The **Save template** button enables to save the entered data as a template of the stamp for the current session of the application. The saved template can be used both in the current and in other modes of the application by clicking the **Load template** button.

The **Welding** tab enables to specify the parameters of the welded connections for the joint. The **Properties of joint** group contains drop-down lists which are used to select the type and method of welding, and specify the position of the weld. The **Truss Panel Points** mode implements the following methods of welding in compliance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, or Table 16.2 of DBN B.2.6-198:2014): manual welding, semiautomatic welding with solid wire less than 1.4 mm in diameter, automatic and semiautomatic welding with the electrode wire 1.4 to 2.0 mm in diameter, automatic welding with the electrode wire 3 to 5 mm in diameter, and semiautomatic welding with flux-cored wire. The position of weld can be underhand, flat, horizontal, vertical or overhead. The **Properties of welding materials** group displays values of the design resistance of the fillet welds for conventional shear of the weld metal, R_{wf} , and of the characteristic resistance of the weld metal, R_{wun} . These values can be specified in the **Materials for Welding** dialog box (see Sec. 9.4.4), which is invoked by clicking the button .

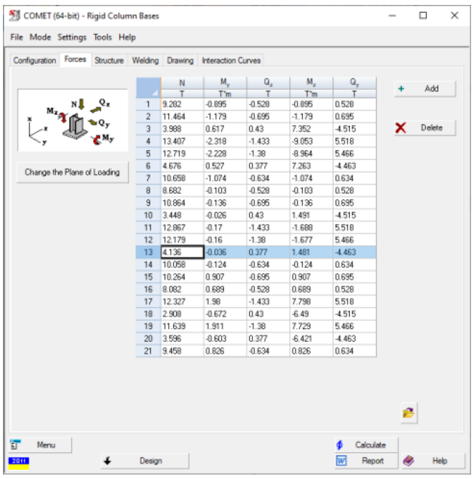


Figure 9.5.1-9. *The* **Forces** tab *of the* **Rigid Column Bases** *dialog box*

The **Forces** tab (Fig. 9.5.1-9) is used to specify the internal forces acting in the joint of the column base: an axial force, N ; bending moments in two planes, M_y and M_z ; their respective shear forces, Q_z and Q_y [7]. The general case of the column loading, biaxial bending with compression or tension, is thus implemented. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the column base members. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current combination of loads. There can be any number of design combinations of loads. Units of measurement for the internal forces acting in the joint are defined in the **Units of Measurement** tab of the **Application Settings** dialog box (see Sec 2.2). To change the load plane, use the respective button. This will transfer the values of M_y and Q_y to the respective columns of the table for M_z and Q_z, and vice versa.

The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter.

[7] To set a proper orientation of the specified internal forces with respect to the principal axes of inertia of the cross-sections that meet in a joint, each bar of the joint is referred to a local coordinate system, *xyz*. The application implements the following orientation of the local coordinate systems of bars: the $x-x$ axis goes from the beginning of a bar (its start node) to its end (its end node), the $y-y$ and $z-z$ axes (the principal central axes of inertia of the bar cross-section) make up a right-hand Cartesian coordinate system together with the $x-x$ axis. The $y-y$ axis is parallel to the *XOY* plane of the global coordinate system, and the $z-z$ axis goes to the upper half-space.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

The **Structure** tab contains a group of buttons to select a design for the joint of the rigid column base (Fig. 9.5.1-10).

To perform a check of the load-bearing capacity of a known design of the column base according to the codes, you have to specify all design parameters of the joint. The parameters include the sizes and thickness of structural members of the joint, diameters of anchor bolts, sizes which determine the mutual arrangement of members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table on the right. The diameter, the steel grade, and the number of anchor bolts (for some types of bases) are selected from the special drop-down lists of the **Anchor bolts** group. The default units of linear measurement are millimeters.

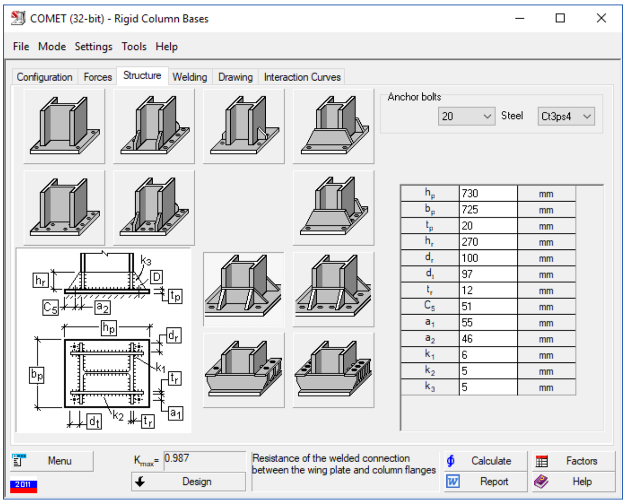


Figure 9.5.1-10. *The* **Structure** *tab of the* **Rigid Column Bases** *dialog box*

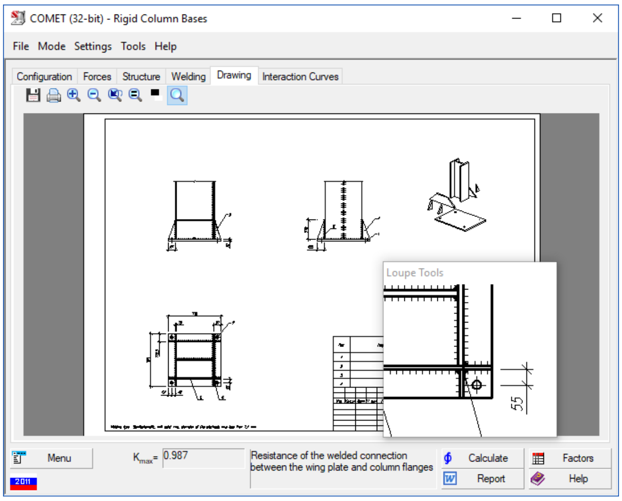


Figure 9.5.1-11. *The* **Drawing** *tab of the* **Rigid Column Bases** *dialog box*

Clicking the **Design** button drops down a menu.

If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified, and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the column base design was performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 8.2). The diameter of the anchor bolts and the thickness of the base plate, as well as the dimensions of the support base plate were taken as the controlled parameters of the rigid column base joints.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the joint (column), structural elements of the joint (base plate, wing plates, anchor plates etc.) and their connections (connections with anchor bolts and welded connections) at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the **K_{max}** field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the column base joint design of the MS stage will be generated.

A complete list of the performed checks can be obtained by clicking the **Factors** button. It will be displayed in the special **Factors Diagram** dialog box, where you can browse the values of all utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the rigid column bases performed by the application is given in Table 9.5.1-1.

Clicking the **Report** button generates a report document which contains the initial data and the results of analysis (see Sec 2.5).

Table 9.5.1-1. A list of the load-bearing capacity checks of the members and connections of the joints of the rigid column bases

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Bending resistance of the base plate under normal stresses in areas supported along the contour	Fig. 9.5.1-2, *a, b, h, i;* Fig. 1.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (103)	Sec.1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12
Bending resistance of the base plate under normal stresses in areas supported on three sides	Fig. 9.5.1-2; 9.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (104)	Sec.1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12
Bending resistance of the base plate under normal stresses in areas supported on two sides meeting at an angle	Fig. 9.5.1-2, *a, b, g*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (104)	Sec. 1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12
Bending resistance of the base plate under normal stresses in cantilever areas of the plate	Fig. 9.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (102)	Sec. 1.7.2, (1.7.1), Annex N, (N.1)	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12
Bending resistance of the base plate under normal stresses in free trapezoid	Fig. 9.5.1-1; 9.5.1-2; 9.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101)	Sec. 1.7.2, (1.7.1)	Sec.11.2, (11.1)	Sec. 7.12

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
areas of the plate							
Resistance of the foundation concrete in local bearing under the plate	Fig. 9.5.1-2; 9.5.1-3						
Resistance of the welded connection between the column and the base plate	Fig. 9.5.1-1	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the wing plate and column flanges	Fig. 9.5.1-2, *a*, *b*, *h*, *i*; Fig. 9.5.1-3	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the wing plate and the base plate	Fig. 9.5.1-2, *a*, *b*, *h*, *i*; Fig. 9.5.1-3	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the cantilever stiffener and column flanges	Fig. 9.5.1-2, *c*, *d*, *e*, *f*; Fig. 9.5.1-3	Sec. 11.4, (33)	Sec. 15.1.15, (38)	Sec. 14.1.15, (44)	Sec. 1.12.1.15, (1.5.4)	Sec. 16.1.15, (9.4)	Sec. 13.4
Resistance of the welded connection between the cantilever stiffener and the wing plate	Fig. 9.5.1-2, *a*, *b*, *g*	Sec. 11.5, (120)-(123), (126)	Sec. 15.1.16, (155), (156), Sec.15.1.17, (157), (158), Sec. 15.1.19, (161)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.5, (129)-(132), (135)
Resistance of the anchor bolts	Fig. 9.5.1-1; 9.5.1-2; 9.5.1-3	Sec. 11.7*, (129), Sec. 11.8, (130)	Sec. 15.2.9, (167), Sec. 15.2.10, (168)	Sec. 14.2.9, (186)-(188), Sec. 14.2.10, (189)	Sec. 1.12.2.9, (1.12.12) – (1.12.14), Sec. 1.12.2.10, (1.12.15),	Sec. 16.2.9, (16.12) – (16.14), Sec. 16.2.10, (16.15)	Sec. 13.7*, (138), Sec. 13.8, (139)
Bending resistance of the wing plate under shear stresses	Fig. 9.5.1-3	Sec. 5.12, (29)	Sec. 9.2.1, (36)	Sec. 8.2.1, (42)	Sec. 1.5.2.1, (1.5.2)	Sec. 9.2.1, (9.2)	Sec. 7.12, (25)

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Bending resistance of the wing plate under reduced stresses	Fig. 9.5.1-3	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4)	Sec. 9.2.1, (9.4)	Sec. 7.12, (29)
Bending resistance of the wing plate under normal stresses	Fig. 9.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.2.1, (41)	Sec. 1.5.2.1, (1.5.1)	Sec. 9.2.1, (9.1)	Sec. 7.12, (24)
Resistance of the cantilever stiffener under shear stresses	Fig. 9.5.1-3	Sec. 5.12, (29)	Sec. 9.2.1, (36)	Sec. 8.2.1, (42)	Sec. 1.5.2.1, (1.5.2)	Sec. 9.2.1, (9.2)	Sec. 7.12, (25)
Resistance of the cantilever stiffener under reduced stresses	Fig. 9.5.1-3	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4)	Sec. 9.2.1, (9.4)	Sec. 7.12, (29)
Bending resistance of the cantilever stiffener under normal stresses	Fig. 9.5.1-3	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.2.1, (41)	Sec. 1.5.2.1, (1.5.1)	Sec. 9.2.1, (9.1)	Sec. 7.12, (24)
Bending resistance of the anchor plate under shear stresses	Fig. 9.5.1-3	Sec. 5.12, (29)	Sec. 9.2.1, (36)	Sec. 8.2.1, (42)	Sec. 1.5.2.1, (1.5.2)	Sec. 9.2.1, (9.2)	Sec. 7.12, (25)
Bending resistance of the anchor plate under reduced stresses	Fig. 9.5.1-3	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4)	Sec. 9.2.1, (9.4)	Sec. 7.12, (29)
Bending resistance of the anchor plate under normal stresses	Fig. 9.5.1-3	Sec. 5.12, (28)	Sec.9.2.1, (35)	Sec. 8.2.1, (41)	Sec.1.5.2.1, (1.5.1)	Sec. 9.2.1, (9.1)	Sec. 7.12, (24)

Notes:

1. **Design resistance of the fusion line of the fillet welded connections R_{wz} is determined by the formula from Table 3 of SNiP II-23-81* or Table 4 of SP 53-102-2004 (or SP 16.13330) or Table 1.3.3 of DBN B.2.6-163:2010 or Table 7.3 of DBN B.2.6-198:2014 or acc. to Table E.9 of Appendix E ShNK 2.03.05-13.**
2. **Design resistance of butt welds R_{wy} is determined by the formula from Table 3 of SNiP II-23-81* or Table 4 of SP 53-102-2004 (or SP 16.13330) or Table 1.3.3 of DBN B.2.6-163:2010 or Table 7.3 of DBN B.2.6-198:2014 or Table E.9 of Appendix E ShNK 2.03.05-13.when there are no *physical methods of quality control*.**
3. **Analysis of the base plate takes into account the service factor γ_c according to item 11, Table 6* of SNiP II-23-81* or item 3.9, Table 1 of SP 53-102-2004 (or item 9, Table 1 of SP 16.13330) or item 9, Table 1.1.1 of DBN B.2.6-163:2010 or item 9, Table 5.1 of DBN B.2.6-198:2014 or item 13, Table F.1 of Appendix F ShNK 2.03.05-13.**
4. **Analysis of bolted connections takes into account the service factor γ_b, which is taken as 0,9 according to item 1,**

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Table 35* of SNiP II-23-81* and notes to Table 38 of SP 53-102-2004 (or notes to Table 41 of SP 16.13330) or Table 1.12.4 of DBN B.2.6-163:2010 or Table 16.4 of DBN B.2.6-198:2014 or Table F.2 of Appendix F ShNK 2.03.05-13 as for the bolts of B and C accuracy class, and for high strength bolts with non-controlled tightening.							
5. **The leg length of fillet welds are taken from the analysis, but not less than the structural minimum values given in Table 38* of SNiP II-23-81* and Table 35 of SP 53-102-2004 (or Table 38 of SP 16.13330) or Table 1.12.1 of DBN B.2.6-163:2010 or Table 16.1 of DBN B.2.6-198:2014 and not more than the maximum values defined in Sec. 12.8, a) SNiP II-23-81* and Sec. 15.1.7, a) SP 53-102-2004 (or Sec.14.1.7, a) SP 16.13330) or Sec.1.12.1.5, b) DBN B.2.6-163:2010 or Sec. 16.1.5, b) DBN B.2.6-198:2014 or given in Table 29 ShNK 2.03.05-13.**							
6. **The design length of longitudinal fillet welds is not more than the maximum value defined in Sec. 12.8, d) SNiP II-23-81* and Sec. 15.1.7, d) SP 53-102-2004 (or Sec.14.1.7, d) SP 16.13330) or Sec.1.12.1.5, e) DBN B.2.6-163:2010 or Sec. 16.1.5, e) DBN B.2.6-198:2014 or Sec. 14.15, d) ShNK 2.03.05-13.**							

Once you switch to the **Drawing** tab (Fig. 9.5.1-11), the application performs a check and design of the joint, similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

The upper part of the **Drawing** tab contains a toolbar with buttons ( ),which enable to zoom the image in or out, save the drawing as DWG (DXF) for AutoCAD, or print it out.

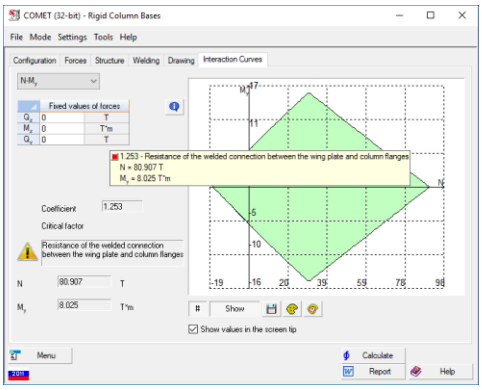


Figure 9.5.1-12. *The* **Interaction Curves** *tab of the* **Rigid Column Bases** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the rigid column base joint under various pairs of internal forces which can arise in the column base section are plotted in the **Interaction Curves** tab (Fig. 9.5.1-12).

Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the rigid base joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of internal forces corresponding to the current position of the pointer in the plot area of the interaction curve.

The maximum value of the utilization factor of restrictions K_{max}, that corresponds to the current values of the internal forces will be displayed in the **Factor** field, and the name of the type of check in which it takes place will be output in the **Critical factor** field. When the pointer is placed outside the area of the load-bearing capacity where $K_{max} > 1$, a warning sign is displayed next to the name of the type of check .

9.5.2 Nominally Pinned Column Bases

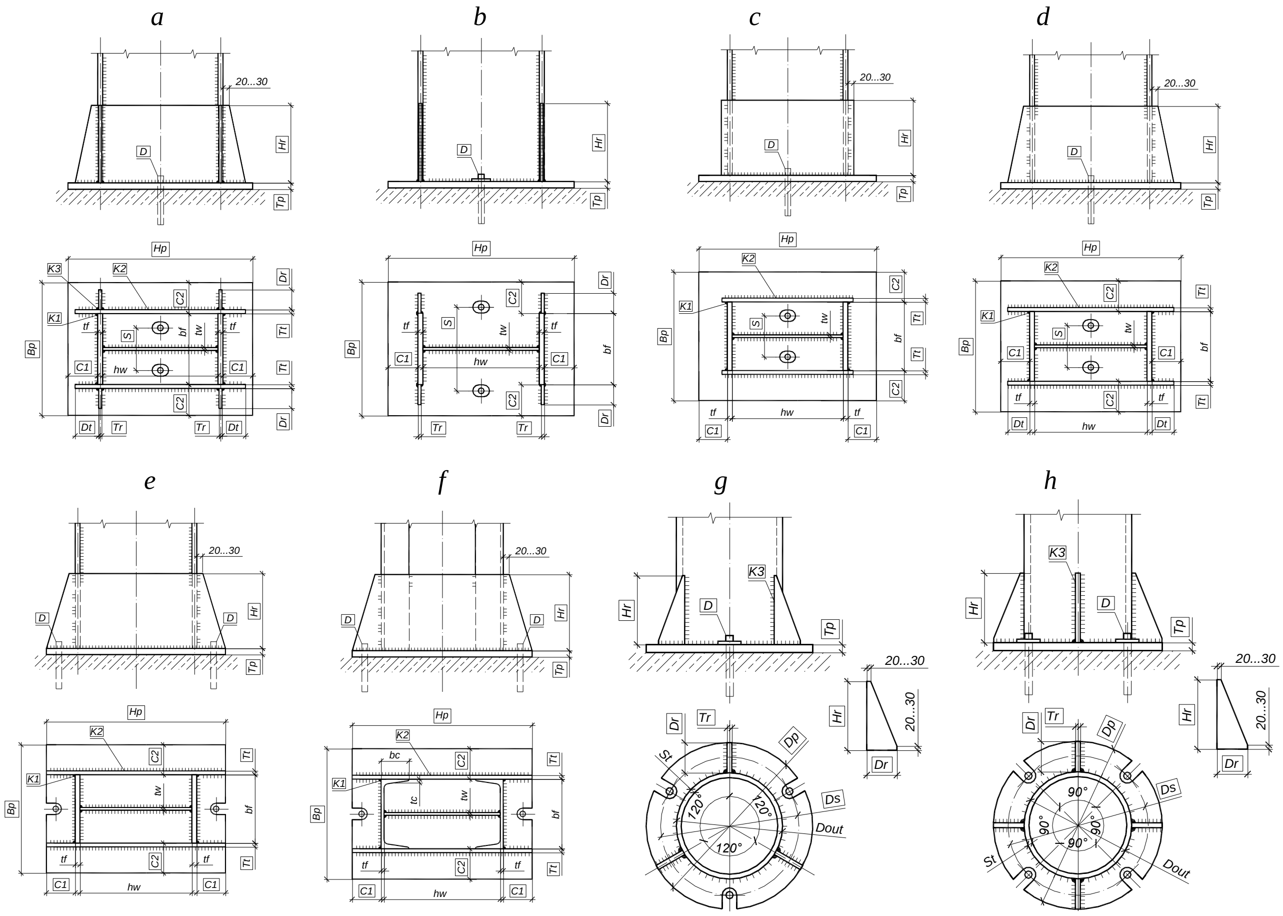
The **Nominally Pinned Column Bases** mode enables to design and check the joints of the column bases which provide a pinned column-to-foundation connection. This mode comprises a wide range of designs for this type of joints, such as:

- column bases with wing plates and cantilever stiffeners (Fig. 9.5.2-1);
- column bases without wing plates and cantilever stiffeners (Fig. 9.5.2-2).

Designs of the nominally pinned column bases provided in the software are somewhat different from those of the rigid column bases. Anchor bolts in these bases are placed taking into account the condition of providing a certain compliance of the joint with respect to the angular deformations, which enables to consider such a column-to-foundation connection as a pinned one.

This mode performs the following checks in compliance with the codes:

- resistance of the structural members of the column base joint (base plate, wing plates and cantilever stiffeners, foundation concrete in local bearing);
- resistance of welded connections (between the column and the base plate, between the wing plate and column flanges, between the wing plate and the base plate, between the cantilever stiffener and column flanges, between the cantilever stiffener and the wing plate);
- a number of structural and assortment constraints.



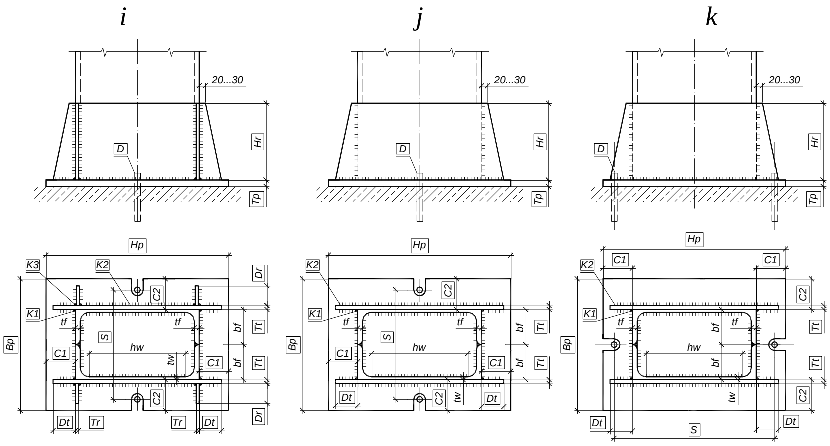


Figure 9.5.2-1. Types of designs for nominally pinned column bases with wing plates and cantilever stiffeners

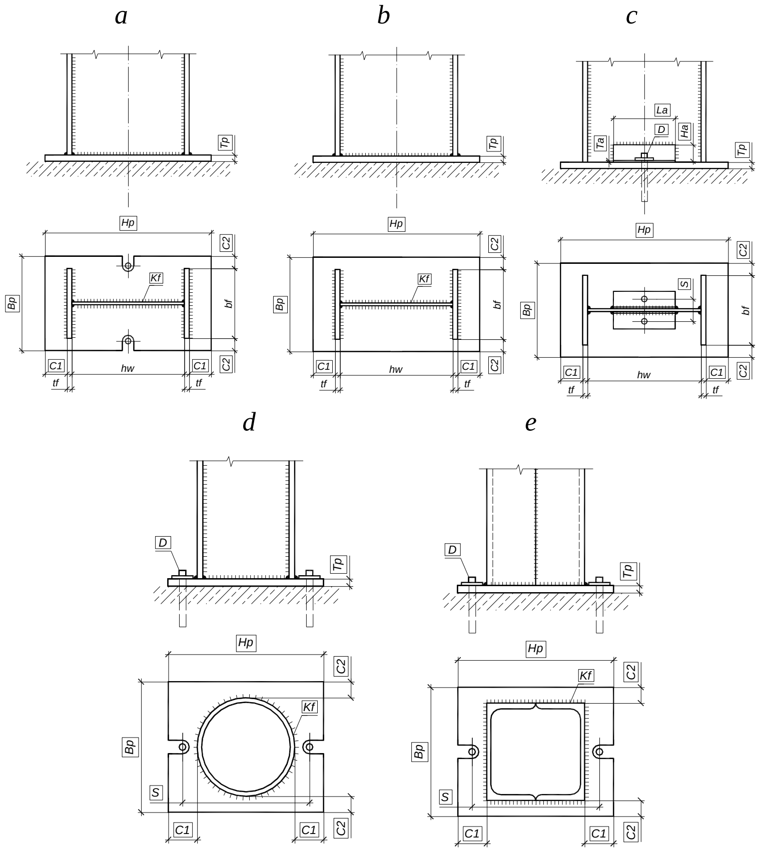


Figure 9.5.2-2. Types of designs for nominally pinned column bases without wing plates and cantilever stiffeners

The **Nominally Pinned Column Bases** dialog box contains the following tabs: **Configuration** (Fig. 9.5.2-3, 9.5.2-4), **Forces** (Fig. 9.5.2-7), **Structure** (Fig. 9.5.2-8, 9.5.2-9, 9.5.2-10), **Welding**, **Drawing** (Fig. 9.5.2-11) and **Interaction Curves** (Fig. 9.5.2-12).

First you have to select the type of the column section by clicking the respective button in the **Configuration** tab (Fig. 9.5.2-3 and 9.5.2-4). The mode provides four types of cross-sections: a rolled I-section, a welded I-section, a compound section made of two rolled channels, and a circular hollow section. The sizes of the cross-section of the welded I-section and the profile number of the rolled cross-section are specified in the same way as in the **Rigid Column Bases** mode.

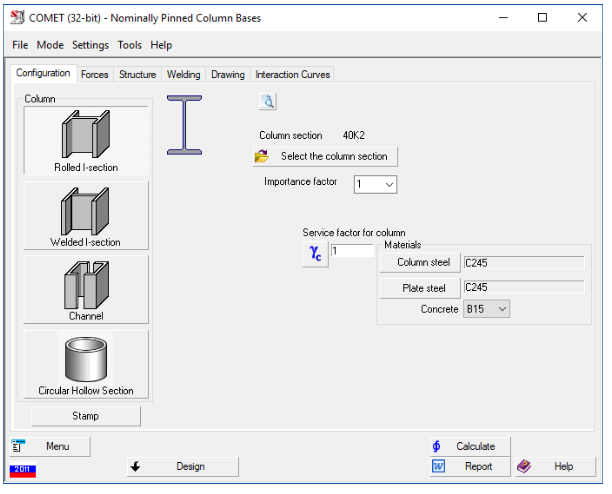


Figure 9.5.2-3. *The* **Configuration** *tab of the* **Nominally Pinned Column Bases** *dialog box* (a rolled I-section is selected as the column cross-section type)

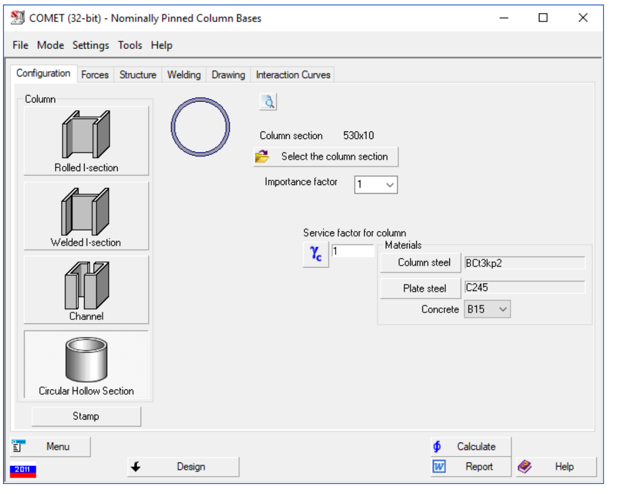
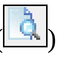


Figure 9.5.2-4. *The* **Configuration** *tab of the* **Nominally Pinned Column Bases** *dialog box* (a circular hollow section is selected as the column cross-section type)

If a compound section made of two rolled channels is selected as the column cross-section type, you then have to select an assortment and the channel number in this assortment in the **Select profile** dialog box (Fig. 9.5.2-5), which can be invoked by clicking the **Select the column section** button. You can specify the distance between the external faces of the channel webs for such section in the **B** text field of the **Structure** tab (for the first type of the base joint). If a circular hollow section is selected as the column cross-section type, its sizes are selected from the respective assortment.

The column cross-section can be checked in the **Preview** window (Fig. 9.5.2-6), which can be invoked by clicking the **Preview** button ().

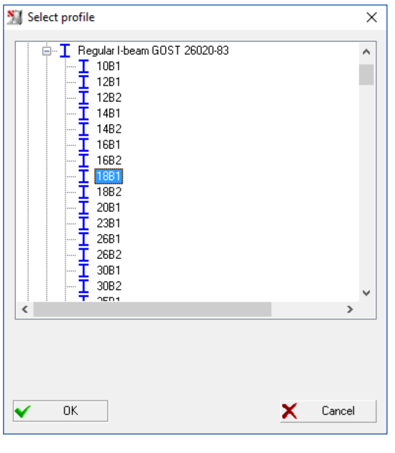


Figure 9.5.2-5. *The* **Select profile** *dialog box*

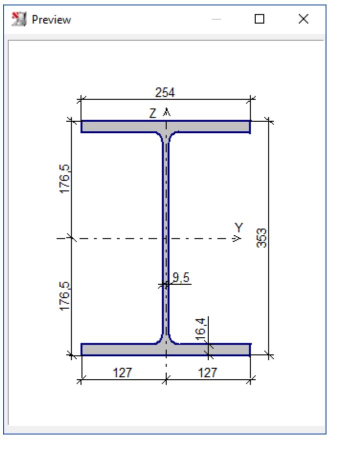


Figure 9.5.2-6. *The* **Preview** *window*

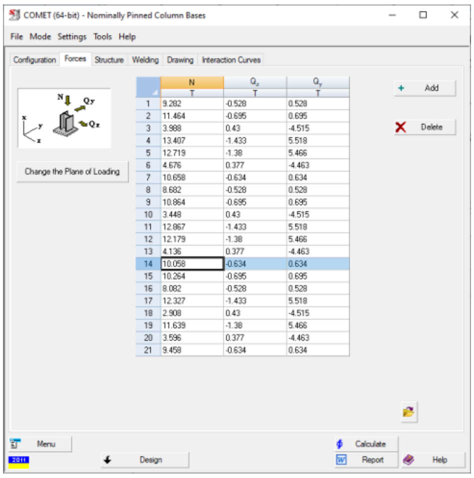
Materials used for design and analysis of a nominally pinned column base joint can be defined by clicking the respective buttons, **Column steel** (enables to specify steel for the column) and **Plate steel** (enables to specify steel for the support base plate, wing plates, cantilever stiffeners, anchor plates, etc.). The **Concrete** drop-down list enables to select concrete classes for the foundation (see Fig. 9.5.2-3).

The importance factor which will be further multiplied by the values of all internal forces acting in the column base section has to be specified in the **Importance factor** field (see Fig. 9.5.2-3 and 9.5.2-4). You can enter the service factor for column in the respective text field, or it can be selected in the **Service Factor** dialog box (see Sec. 9.4.5) after clicking the nearby button ( ).

The service factor of the base plate of the column base is calculated automarically in this mode. The user has to specify the service factor not for the base plate but for the column.

The functionality of the **Stamp** button is the same as that in the **Rigid Column Bases** mode.

The **Welding** tab enables to specify the parameters of the welded connections for the joint. The **Properties of joint** group contains drop-down lists which are used to select the type and method of welding, and specify the position of the weld. The **Truss Panel Points** mode implements the following methods of welding in compliance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, or Table 16.2 of DBN B.2.6-198:2014): manual welding, semiautomatic welding with solid wire less than 1.4 mm in diameter, automatic and semiautomatic welding with the electrode wire 1.4 to 2.0 mm in diameter, automatic welding with the electrode wire 3 to 5 mm in diameter, and semiautomatic welding with flux-cored wire. The position of weld can be underhand, flat, horizontal, vertical or overhead. The **Properties of welding materials** group displays values of the design resistance of the fillet welds for conventional shear of the weld metal, R_{wf}, and of the characteristic resistance of the weld metal, R_{wun}. These values can be specified in the **Materials for Welding** dialog box (see Sec. 9.4.4), which is invoked by clicking the button .



	N, T	Q_z, T	Q_y, T
1	9.282	-0.528	0.528
2	11.464	-0.695	0.695
3	3.988	0.43	-4.515
4	13.407	-1.433	5.518
5	12.719	-1.38	5.466
6	4.676	0.377	-4.463
7	10.658	-0.634	0.634
8	8.682	-0.528	0.528
9	10.864	-0.695	0.695
10	3.448	0.43	-4.515
11	12.867	-1.433	5.518
12	12.179	-1.38	5.466
13	4.136	0.377	-4.463
14	10.058	-0.634	0.634
15	10.264	-0.695	0.695
16	8.082	-0.528	0.528
17	12.327	-1.433	5.518
18	2.908	0.43	-4.515
19	11.639	-1.38	5.466
20	3.596	0.377	-4.463
21	9.458	-0.634	0.634

Figure 9.5.2-7. *The* **Forces** tab *of the* **Nominally Pinned Column Bases** *dialog box*

The **Forces** tab (Fig. 9.5.2-7) is used to specify the internal forces acting in the joint of the column base: an axial force, N; shear forces in two planes, Q_z and Q_y. Thus, in addition to the longitudinal force you can specify two transverse forces in this mode which are typical for the columns of braced frame modules.

Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of the axial force and the shear forces for the current combination of loads. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the column base members. There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes. Other units of measurement for the internal forces acting in the joint can be defined in the **Units of Measurement** tab of the **Application Settings** dialog box (see Sec 2.2). To change the load plane, use the respective button. This will transfer the values of M_y and Q_y to the respective columns of the table for M_z and Q_z, and vice versa.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

The **Structure** tab contains a group of buttons to select a design for the joint of the nominally pinned column base (Fig. 9.5.2-8, 9.5.2-9 and 9.5.2-10).

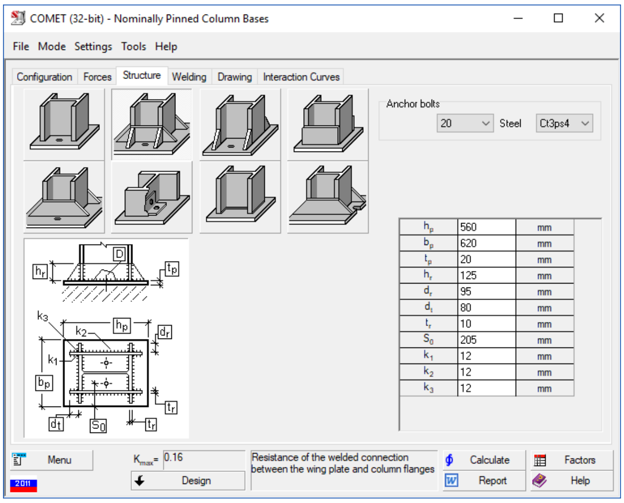


Figure 9.5.2-8. *The* **Structure** *tab*
of the **Nominally Pinned Column Bases** *dialog box*
(a rolled or welded I-section is selected as the column cross-section type)

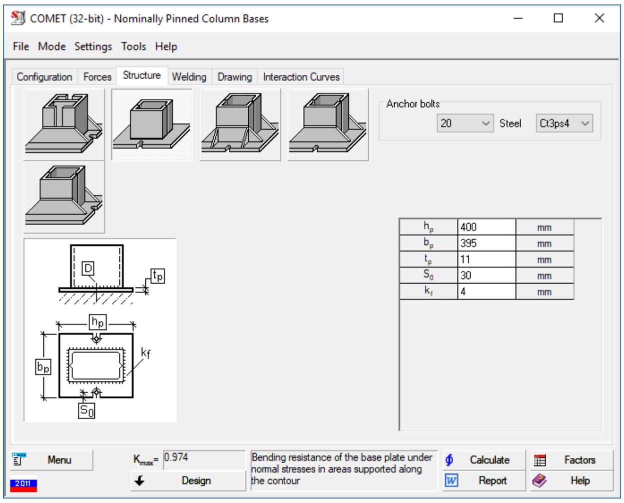


Figure 9.5.2-9. *The* **Structure** *tab*
of the **Nominally Pinned Column Bases** *dialog box*
(a channel is selected as the column cross-section type)

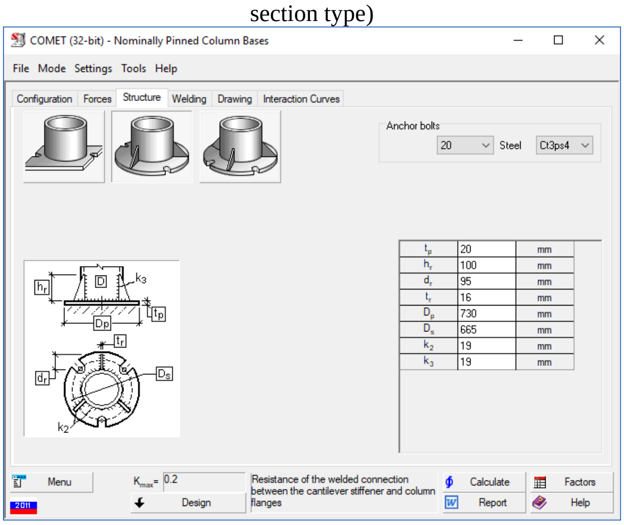


Figure 9.5.2-10. *The* **Structure** *tab*
of the **Nominally Pinned Column Bases** *dialog box*
(a circular hollow section is selected as the column cross-section type)

To perform a check of the load-bearing capacity of the specified structural design of the column base joint, you have to enter the design parameters of the joint in the table in the **Structure** tab. The diameter and the steel grade of anchor bolts are selected from the special drop-down lists of the **Anchor bolts** group. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified*** (see Sec. 9.5.1), is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified, and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the nominally pinned column base design was performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 9.2). The diameter of the anchor bolts and the thickness of the base plate, as well as the dimensions of the support base plate were taken as the controlled parameters.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the joint (column), structural elements of the joint (base plate, wing plates, cantilever stiffeners etc.) and welded connections at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the nominally pinned column base joint design of the MS stage will be generated.

A complete list of the performed checks can be obtained by clicking the **Factors** button. It will be displayed in the special **Factors Diagram** dialog box, where you can browse the values of all utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the nominally pinned column bases performed by the application is given in Table 9.5.2-1.

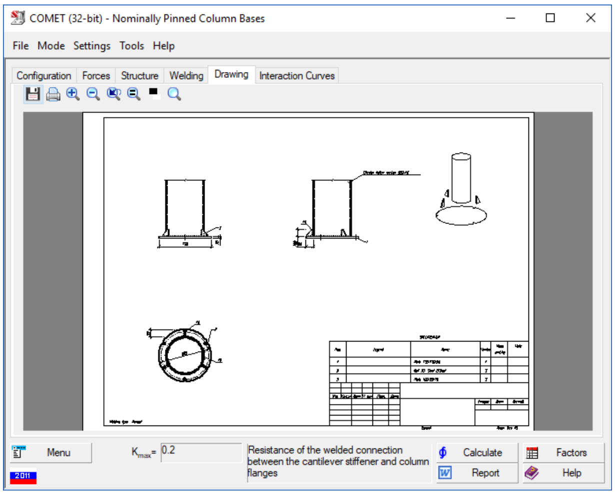


Figure 9.5.2-11. *The* **Drawing** *tab of the* **Nominally Pinned Column Bases** *dialog box*

Clicking the **Report** button generates a report document which contains the initial data and the results of analysis (see Sec. 2.5).

Once you switch to the **Drawing** tab (Fig. 9.5.2-11), the application performs a check of the load-bearing capacity of the nominally pinned base joint similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the nominally pinned base joint design of the MS stage will be generated.

The functionality of the **Report** button and of the controls in the **Drawing** tab is similar to that described for the **Rigid Column Bases** mode (see Sec. 9.5.1).

Table 9.5.2-1. A list of the load-bearing capacity checks of the members and connections of the joints of the nominally pinned column bases

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.133 30	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Bending resistance of the base plate under reduced stresses	Fig. 9.5.2-2	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4), Sec. 1.7.2, (1.7.1)	Sec. 9.2.1, (9.4), Sec. 11.2, (11.1)	Sec. 7.14, (29)
Bending resistance of the base plate under normal stresses in areas supported along the contour	Fig. . 9.5.2-1, *a*, *c*, *d*, *e*, *f*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (103)	Sec. 1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12, (24)
Bending resistance of the base plate under normal stresses in areas supported on three sides	Fig. 9.5.2-1, *a*, *b*, *d*, *e*, *f*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (104)	Sec. 1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12, (24)

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Bending resistance of the base plate under normal stresses in areas supported on two sides meeting at an angle	Fig. . 9.5.2-1, *a*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (104)	Sec. 1.7.2, (1.7.1), Annex N, (N.2), Table N.2	Sec.11.2, (11.1), Sec. M, (M.1), (M.2) Table M.2	Sec. 7.12, (24)
Bending resistance of the base plate under normal stresses in cantilever areas of the plate	Fig. . 9.5.2-1, *e*, *f*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101), (102)	Sec. 1.7.2, (1.7.1), Annex N, (N.1)	Sec.11.2, (11.1), Sec. M, (M.1), (M.2), Table M.2	Sec. 7.12, (24)
Bending resistance of the base plate under normal stresses in free trapezoid areas of the plate	Fig. . 9.5.2-1, *a*, *b*, *c*, *d* Fig. . 9.5.2-2	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.6.2, (101)	Sec. 1.7.2, (1.7.1)	Sec.11.2, (11.1)	Sec. 7.12, (24)
Resistance of the foundation concrete in local bearing under the plate	Fig. . 9.5.2-1, Fig. . 9.5.2-2						
Resistance of the welded connection between the column and the base plate	Fig. . 9.5.2-2, *a*, *b*	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the wing plate and column flanges	Fig. . 9.5.2-1, *a*, *c*, *d*, *e*, *f*	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the wing plate and the base plate	Fig. . 9.5.2-1, *a*, *c*, *d*, *e*, *f*	Sec. 11.2*, (120)-(121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129)-(130)
Resistance of the welded connection between the cantilever stiffener and column flanges	Fig. . 9.5.2-1, *b*	Sec. 11.4, (33)	Sec. 15.1.15, (38)	Sec. 14.1.15, (44)	Sec. 1.12.1.15, (1.5.4)	Sec. 16.1.15, (9.4)	Sec. 13.4
Resistance of the welded connection between the cantilever stiffener and the wing plate	Fig. . 9.5.2-1, *a*	Sec. 11.5, (120)-(123), (126)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec.1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.5, (129)-(132), (135)

Check	Type of base	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the bolted connection between the anchor angle and the base plate	Fig. . 9.5.2-2, *c*	Sec. 11.7*, (127)-(128), Sec. 11.8, (130)	Sec. 15.2.9, (167), Sec. 15.2.10, (168)	Sec. 14.2.9, (186), (187), Sec. 14.2.10, (189)	Sec. 1.12.2.9, (1.12.12), (1.12.13), Sec. 1.12.2.10, (1.12.15)	Sec. 16.2.9, (16.12), (16.13), Sec. 16.2.10, (16.15)	Sec. 13.7, (136)-(137), Sec. 13.8, (139)
Notes: see Table 9.5.1-1.							

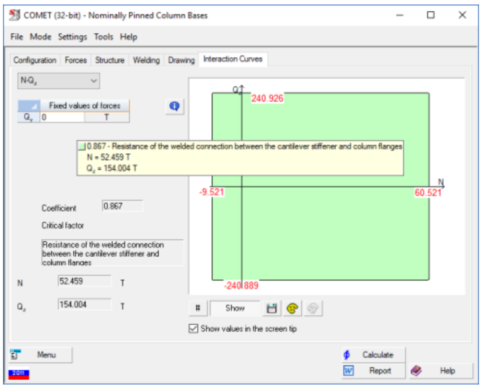


Figure 9.5.2-12. *The* **Interaction Curves** *tab of the* **Nominally Pinned Column Bases** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the nominally pinned column base joint under various pairs of internal forces which can arise in the column base section are plotted in the **Interaction Curves** tab (Fig. . 9.5.2-12).

Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the nominally pinned base joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

The maximum value of the utilization factor of restrictions K_{max}, that corresponds to the current values of the internal forces will be displayed in the **Factor** field, and the name of the type of check in which it takes place will be output in the **Critical factor** field. When the pointer is placed outside the area of the load-bearing capacity where $K_{max} > 1$, a warning sign is displayed next to the name of the type of check .

9.5.3 Beam Splices

The **Beam Splices** mode enables to design and check the load-bearing capacity of erection joints between I-beams with high strength bolts or ordinary bolts using plates or end-plates. This mode comprises a wide range of designs for erection joints between beams:

- beam splices with plates using ordinary bolted connections with bolts of normal and improved strength and connections with high strength bolts (Fig. 9.5.3-1);
- end-plate beam splices of various configurations with high strength bolts with their regular placement along the beam webs (Fig. 9.5.3-2);
- end-plate beam splices of various configurations with high strength bolts with their regular placement along the beam webs and flanges (Fig. 9.5.3-3).

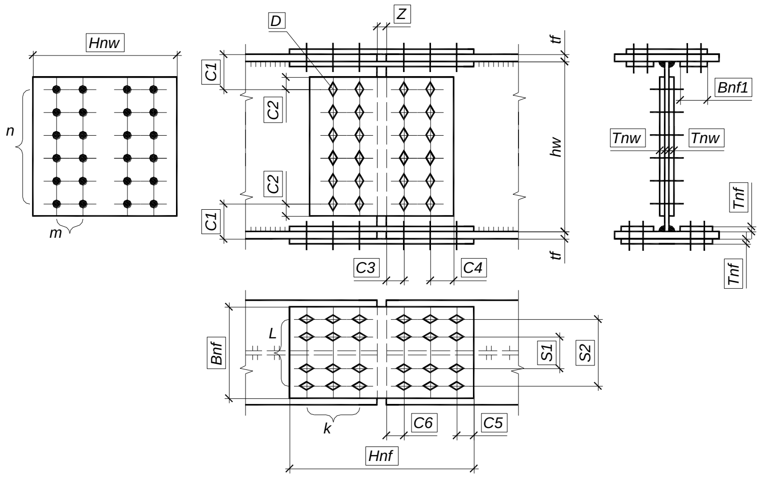
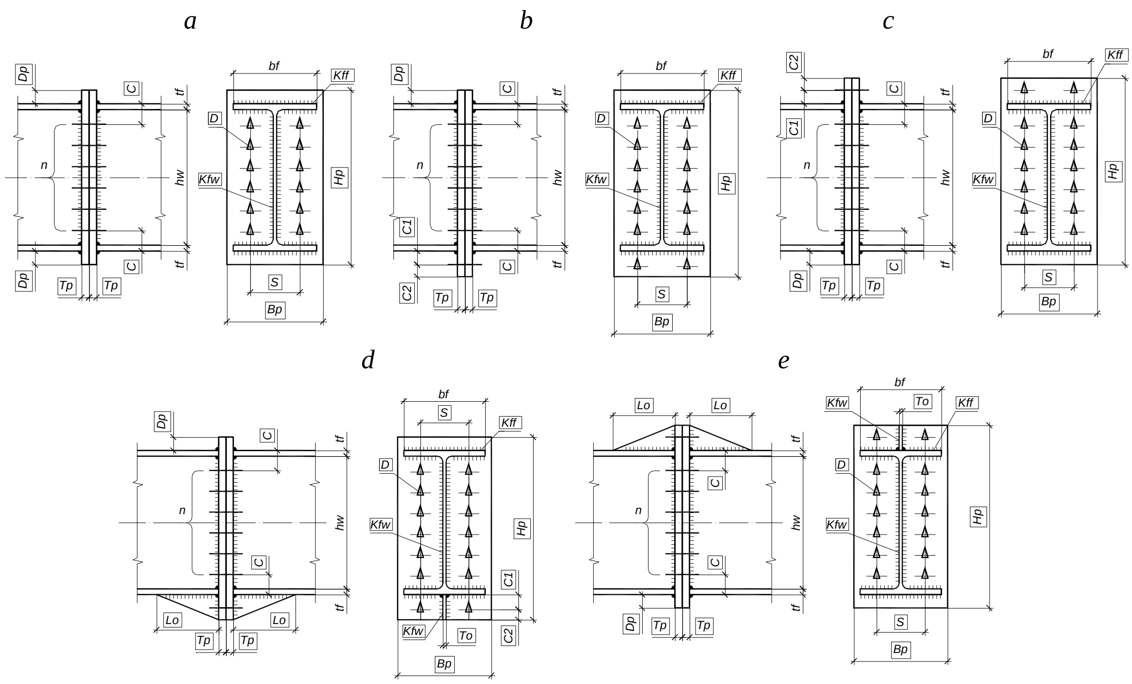


Figure 9.5.3-1. Design of a beam splice with plates



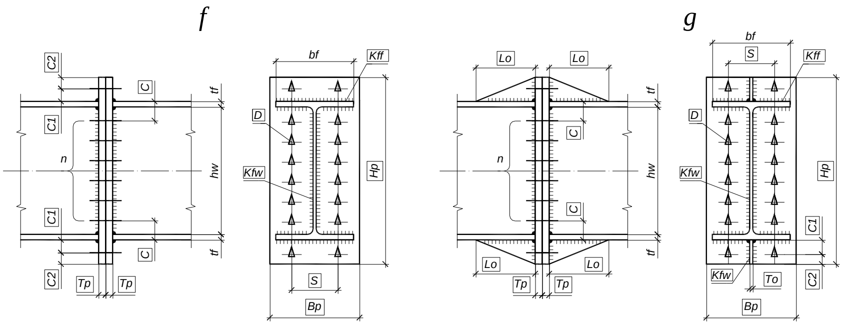


Figure 9.5.3-2. Types of designs for end-plate beam splices with the regular placement of bolts along the beam webs

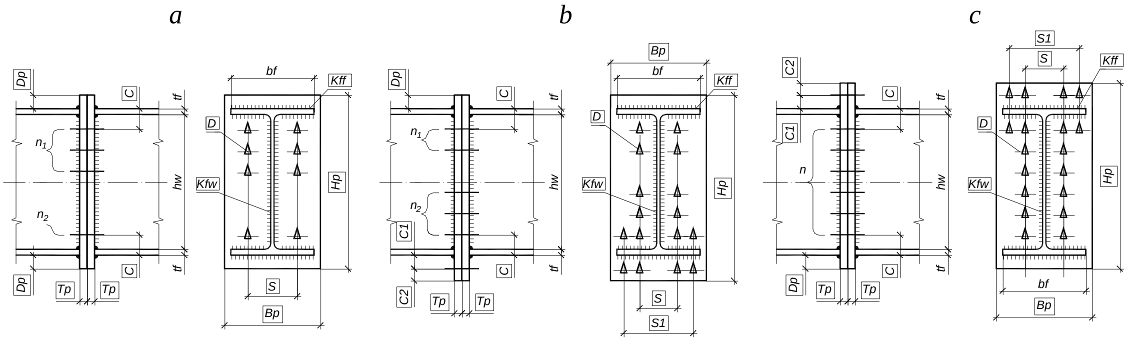


Figure 9.5.3-3. Examples of types of end-plate beam splices with the irregular placement of high strength bolts: *a* – along the beam web; *b* – along the beam web and flange; *c* – along the beam flange

The erection beam splice with plates has an advantage over the end-plate beam splices in that it does not require the members to be manufactured with a high accuracy. However, this kind of joint usually requires a much greater number of bolts in comparison to end-plate joints, therefore it takes more effort to mount the structures. Moreover, the joint with plates entails the weakening of the cross-sections of connected members by holes, which may in some cases require using more steel for the main structural members.

It should be noted that the computer-aided calculation of beam splices with plates assumes that the cross-sectional dimensions of the plates covering the beam splice are taken as close as possible to the cross-sectional dimensions of the beam. It refers to both the thickness of the plates, and to their linear dimensions (in particular, to the height of the plate on the beam webs and the width of the plates on the beam flanges). In the case when the user has specified the height of the plate on the beam webs as less than the effective height of the beam web, the stress concentrations should be taken into account in the check of this joint. Since there are no standard methods for performing such a calculation, this option is not implemented in the application. Thus, if the user specifies the dimensions of the plates not corresponding to the sizes of the beam section, an error message will be generated.

End-plate joints are usually designed in such a way so that the height of the end-plate corresponds to that of the beam (Fig. 9.5.3-2, *a*, Fig. 9.5.3-3, *a*). If the bending moment acting in the beam splice can hardly be resisted by bolts placed between the beam flanges, it becomes necessary to use the designs that involve external bolt rows. The latter expand the end-plate dimension downward (Fig. 9.5.3-2, *b*, *d*, Fig. 9.5.3-3, *b*) or upward (Fig. 9.5.3-2, *c*, *e*, Fig. 9.5.3-3, *c*), depending on the prevailing sign of the bending moment. If there are significant alternating-sign bending moments, you should use end-plate joints with external bolts on both sides of the beam (Fig. 9.5.3-2, *f*, *g*).

This mode performs the following checks in compliance with the codes:

- resistance of plates and end-plates;
- resistance of bolted and welded connections of the joint;
- a number of structural and assortment constraints.

The **Beam Splices** dialog box contains the following tabs: **Materials** (Fig. 9.5.3-4, 9.5.3-5), **Forces**

(Fig. 9.5.3-6), **Structure** (Fig. 9.5.3-7, 9.5.3-8, 9.5.3-9, 9.5.3-10), **Welding**, **Drawing** (Fig. 9.5.3-11) and **Interaction Curves** (Fig. 9.5.3-12).

First you have to specify the materials used in the beam splice. A steel grade for the connected beams can be selected in the **Steel** dialog box (see Sec. 9.4.1), which is invoked by clicking the **Beam steel** button in the **Materials** tab.

You can enter the service factor for beams in the respective text field, or it can be selected in the **Service Factor** dialog box (see Sec. 9.4.5) after clicking the nearby button ( ).

The service factor of beams at their strength analysis for a section weakened by holes is calculated automatically in this mode.

The importance factor which will be further multiplied by the design values of all internal forces for all design combinations of loadings acting in the splice has to be specified in the **Importance factor** drop-down list in this tab.

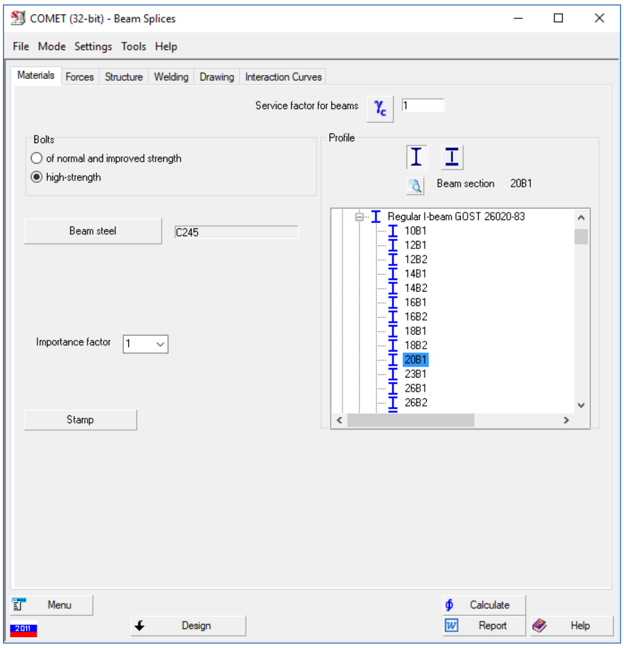


Figure 9.5.3-4. *The* **Materials** *tab of the* **Beam Splices** *dialog box* (a rolled I-section is selected as the beam cross-section type)

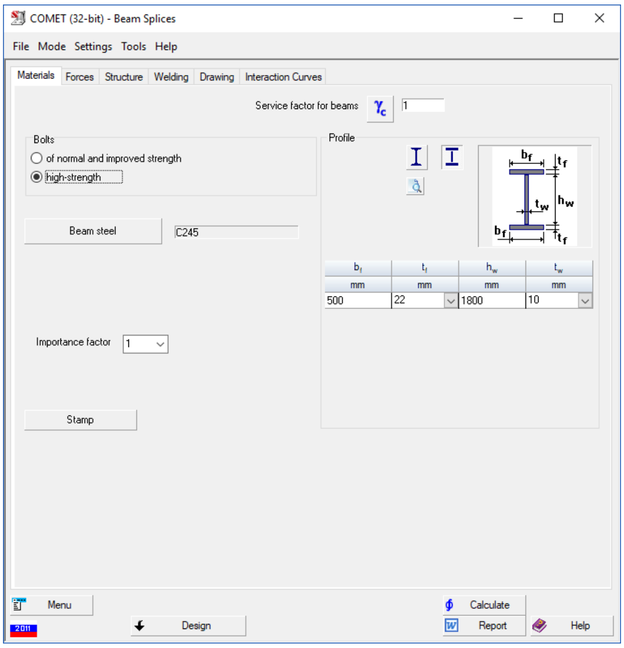


Figure 9.5.3-5. *The* **Materials** *tab of the* **Beam Splices** *dialog box* (a welded I-section is selected as the beam cross-section type)

Controls of the **Profile** group are used to define the type and sizes of the cross-sections of spliced beams. The **Beam Splices** mode provides two types of beam cross-sections: rolled or welded I-section. The section type is selected by clicking the respective button (or). The interface of the right part of the **Materials** tab depends on this choice (Fig. 9.5.3-4, 9.5.3-5).). If a rolled I-section is selected as the beam cross-section type, you then have to select an assortment and the profile number in this assortment from the tree-like list. When a welded I-section is selected as the beam cross-section type, you have to specify the sizes of the beam cross-section: the height, h_w, and the thickness, t_w, of the beam web; the width, b_f, and the thickness, t_f, of the beam flanges. The thickness of the flanges and of the web can be either entered manually or selected from drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel.

The beam cross-section can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ( ).

Clicking the **Stamp** button opens a dialog box which enables to fill in the stamp of the drawing, which will be generated automatically once the structural design of the beam splice is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

The **Welding** tab enables to specify the parameters of the welded connections for the joint. The **Properties of joint** group contains drop-down lists which are used to select the type and method of welding, and specify the position of the weld. The **Truss Panel Points** mode implements the following methods of welding in compliance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, or Table 16.2 of DBN B.2.6-198:2014): manual welding, semiautomatic welding with solid wire less than 1.4 mm in diameter, automatic and semiautomatic welding with the electrode wire 1.4 to 2.0 mm in diameter, automatic welding with the electrode wire 3 to 5 mm in diameter, and semiautomatic welding with flux-cored wire. The position of weld can be underhand, flat, horizontal, vertical or overhead. The **Properties of welding materials** group displays values of the design resistance of the fillet welds for conventional shear of the weld metal, R_{wf}, and of the characteristic resistance of the weld metal, R_{wun}. These values can be specified in the **Materials for Welding** dialog box (see Sec. 9.4.4), which is invoked by clicking the button .

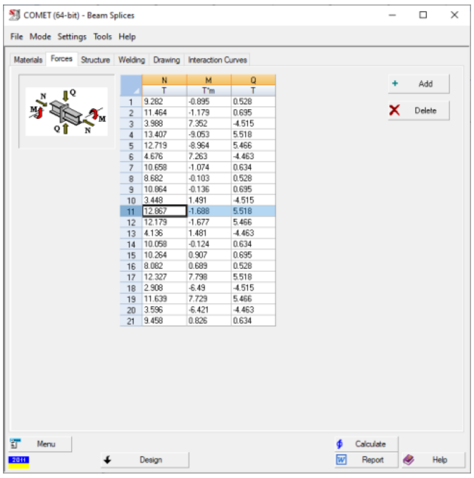


Figure 9.5.3-6. *The* **Forces** *tab of the* **Beam Splices** *dialog box*

The **Forces** tab (Fig. 9.5.3-6) is used to specify the internal forces acting in the beam splice: the axial force, N, the bending moment, M, and its respective shear force, Q. Thus the general case of loading is implemented when the given splice can be used both in the beams of floors and roofs, and in the girders of transverse steel frames.

Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current design combination of loads.

There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter. The positive direction of internal forces is defined by the picture to the left from the table of internal forces.

It should be noted that if the values of the internal forces in the beam splice have been obtained from the analysis of the respective finite element model of the system accounting for the importance of the designed building or structure, the value equal to one has to be selected in the **Importance factor** drop-down list in the **Materials** tab.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

The **Structure** tab contains buttons of the **Type of joint** group, which are used to select a design for the beam splice (Fig. 9.5.3-7, 9.5.3-8).

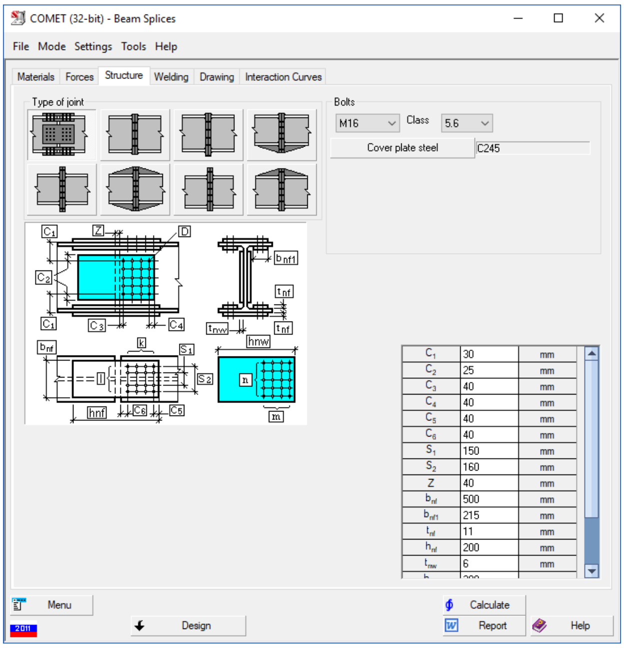


Figure 9.5.3-7. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (a joint with plates and bolts of normal strength is selected)

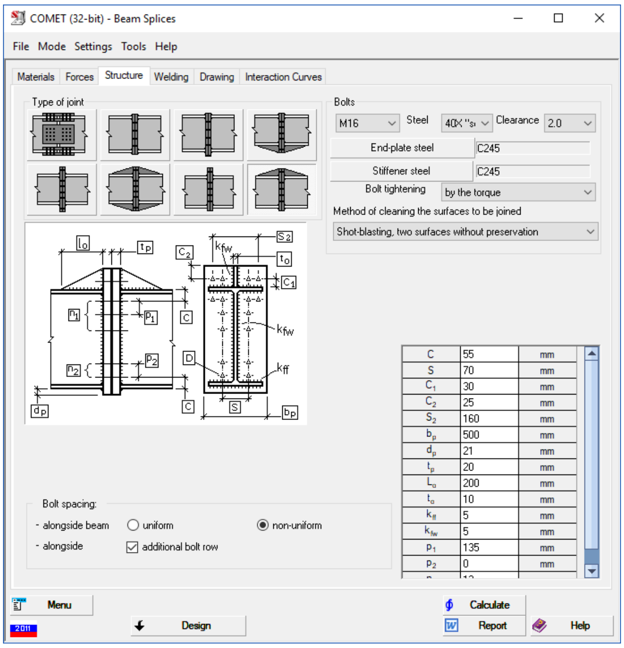


Figure 9.5.3-8. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (an end-plate joint with high strength bolts is selected)

If the bolts of normal and improved strength are used, you have to select the grade (diameter) of the bolts and their strength class from the drop-down lists of the **Bolts** group. If high strength bolts are used, you have to specify the grade (diameter) of the bolts, their steel grade, the clearance (the difference between the diameter of the bolt hole and of the bolt itself), a bolt tightening control method, and a method of cleaning the surfaces to be joined.

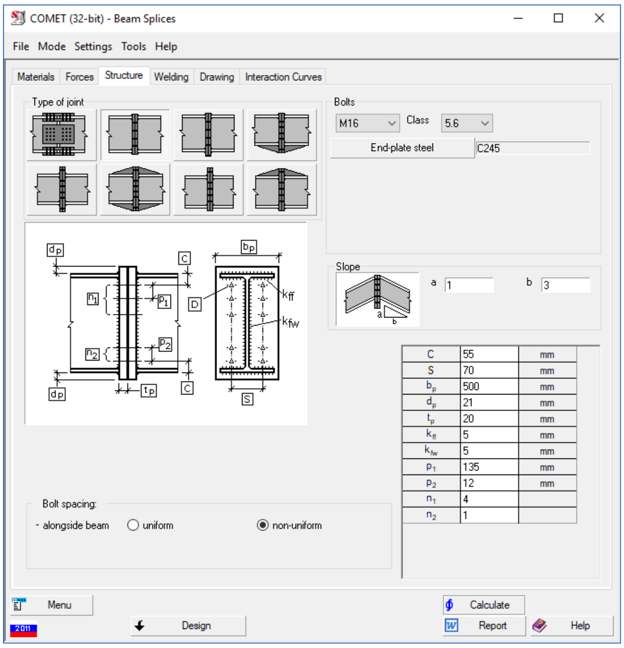


Figure 9.5.3-9. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (end-plate beam splice with the irregular placement of bolts along the web is selected)

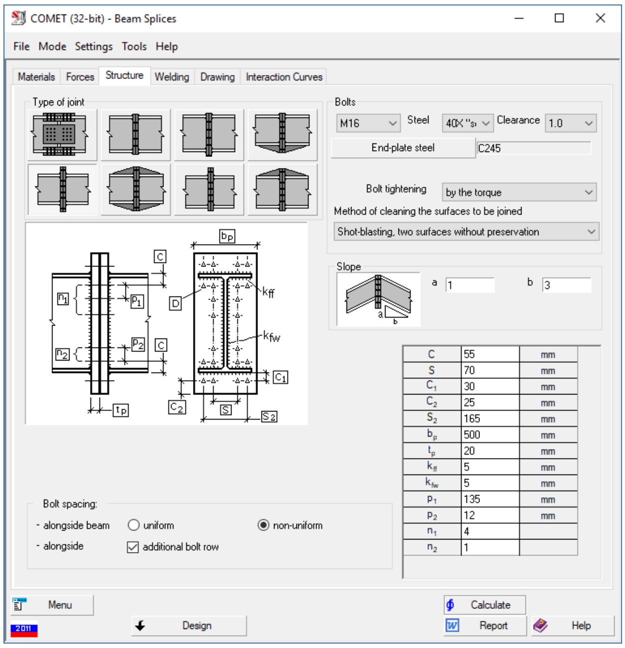


Figure 9.5.3-10. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (end-plate beam splice with the irregular placement of bolts along the web and flanges is selected)

The **Structure** tab also enables to specify different steel grades for the structural elements of the beam splice (end-plates or plates depending on the design of the splice) by clicking the **End-plate steel** and **Web plate and flange plate steel** buttons.

In the case when the user has to perform the analysis or design of end-plate beam splices with the irregular placement of bolts along the elements of the beam section (flanges and/or web), he can use the **Placement of bolts with respect to the beam** group of interface elements which enables to specify the regular or irregular placement of bolts along the web (Fig. 9.5.3-9), and to select the design of a splice with an additional bolt row along the beam flanges (usually in tension) (Fig. 9.5.3-10).

To perform a check of the load-bearing capacity of a known (specified) structural design of the beam splice, you have to specify all parameters of the joint: the sizes and thickness of structural members of the joint, diameters of bolts, sizes which determine the mutual arrangement of members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table on the right in the **Structure** tab. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu (see Sec. 9.5.1). If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the beam splice design is performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 8.2). The diameter of bolts and the number of bolt rows have been selected as the controlled parameters of the beam splice joint structures for an erection beam splice with plates, and the diameter of bolts, thickness of the end-plate and the number of bolt rows – for the erection end-plate beam splices.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the considered joint (beam), structural elements of the splice (end-plates, plates, stiffeners etc.), welded and bolted connections in the splice at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the beam splice joint design of the MS stage will be generated.

A complete list of the performed checks can be obtained by clicking the **Factors** button. It will be displayed in the special **Factors Diagram** dialog box, where you can browse the values of all utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the beam splices performed by the application is given in Table 9.5.3-1.

Clicking the **Report** button generates a report document which contains the initial data and the results of analysis (see Sec. 2.5).

Once you switch to the **Drawing** tab (Fig. 9.5.3-11) the application performs a check of the beam splice joint similarly to the **Calculate** mode. If the results of the check of the parameters of the joint do not contradict the structural and standard requirements, a drawing of the beam splice joint design of the MS stage will be generated.

The functionality of the **Report** button and of the controls in the **Drawing** tab is similar to that described for the **Rigid Column Bases** mode (see Sec. 9.5.1).

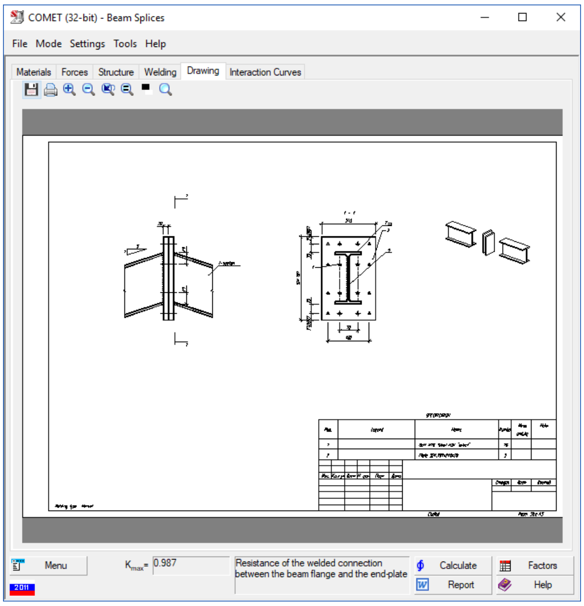


Figure 9.5.3-11. *The* **Drawing** *tab of the* **Beam Splices** *dialog box*

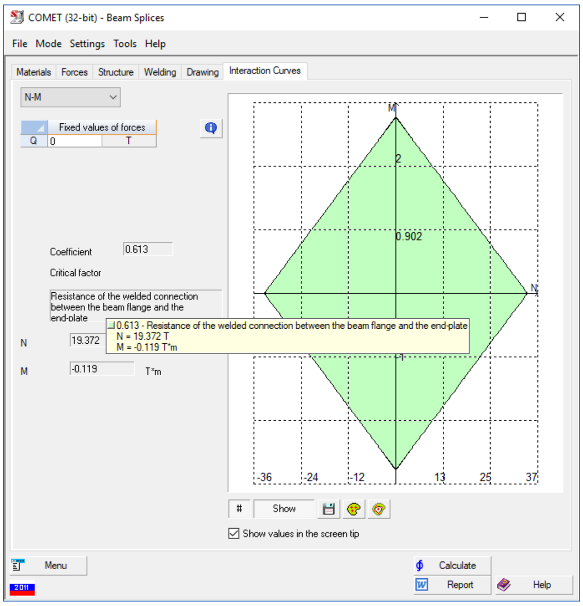


Figure 9.5.3-12. *The* **Interaction Curves** *tab of the* **Beam Splices** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the beam splice joint under various pairs of internal forces which can arise in the beam section adjacent to the splice are plotted in the **Interaction Curves** tab (Fig. 9.5.3-12). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the beam splice joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

Table 9.5.3-1. A list of the load-bearing capacity checks of the members and connections of the joints of the beam splices

Check	Type of splice	SNiP II-23-81* and the Guide [3]	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Shear strength of bolts in the bolted connection of the beam web	Fig. 9.5.3-1	Sec. 11.7*, (127), (130)	Sec. 15.2.9, (165), (168)	Sec. 14.2.9, (186), (189)	Sec. 1.12.2.9, (1.12.12), (1.12.15)	Sec. 16.2.9, (16.12), (16.15)	Sec. 13.7, (136), (139)
Shear strength of bolts in the bolted connection of the beam flange	Fig. 9.5.3-1	Sec. 11.7*, (127), (130)	Sec. 15.2.9, (165), (168)	Sec. 14.2.9, (186), (189)	Sec. 1.12.2.9, (1.12.12), (1.12.15)	Sec. 16.2.9, (16.12), (16.15)	Sec. 13.7, (136), (139)
Bearing resistance of the beam web and plates	Fig. 9.5.3-1	Sec. 11.7*, (128), (130)	Sec. 15.2.9, (166), (168)	Sec. 14.2.9, (187), (189)	Sec. 1.12.2.9, (1.12.13), (1.12.15)	Sec. 16.2.9, (16.13), (16.15)	Sec. 13.7, (137), (139)
Bearing resistance of the beam flange and plates	Fig. 9.5.3-1	Sec. 11.7*, (128), (130)	Sec. 15.2.9, (166), (168)	Sec. 14.2.9, (187), (189)	Sec. 1.12.2.9, (1.12.13), (1.12.15)	Sec. 16.2.9, (16.13), (16.15)	Sec. 13.7, (137), (139)
Strength of bolts in the bolted connection of the beam web	Fig. 9.5.3-1	Sec. 11.13*, (131), (132)	Sec. 15.3.3, (170), (171)	Sec. 14.3.3, (191), (192)	Sec. 1.12.3.3, (1.12.17), (1.12.18)	Sec. 16.3.3, (16.17), (16.18)	Sec. 13.13, (140), (141)
Strength of bolts in the bolted connection of the beam flange	Fig. 9.5.3-1	Sec. 11.13*, (131), (132)	Sec. 15.3.3, (170), (171)	Sec. 14.3.3, (191), (192)	Sec. 1.12.3.3, (1.12.17), (1.12.18)	Sec. 16.3.3, (16.17), (16.18)	Sec. 13.13, (140), (141)
Strength of bolts located in the area of the top beam flange	Fig. 9.5.3-2	Sec. 27.13, 27.14 [3]	Sec. 16.9.2	Sec. 15.9.2	Sec. 1.13.12.2	Sec. 17.12.2	–
Strength of bolts located in the area of the bottom beam flange	Fig. 9.5.3-2	Sec. 27.13, 27.14 [3]	Sec. 16.9.2	Sec. 15.9.2	Sec. 1.13.12.2	Sec. 17.12.2	–
Strength of bolts located in the area of the beam web	Fig. 9.5.3-2	Sec. 27.13, 27.14 [3]	Sec. 16.9.2	Sec. 15.9.2	Sec. 1.13.12.2	Sec. 17.12.2	–
Bending resistance of the end-plate	Fig. 9.5.3-2	Sec. 27.13, 27.14 [3]	Sec. 16.9.2	Sec. 15.9.2	Sec. 1.13.12.2	Sec. 17.12.2	–
Resistance of the welded connection between the beam web and the end-plate	Fig. 9.5.3-2	Sec. 11.2*, (120)-(123)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, 13.3, (129)-(132)
Resistance of the welded connection between the beam flange and the end-plate	Fig. 9.5.3-2	Sec. 11.2*, (120)-(123)	Sec. 15.1.16, (155), (156), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, 13.3, (129)-(132)

Check	Type of splice	SNiP II-23-81* and the Guide [3]	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of a plate on beam flanges	Fig. 9.5.3-1	Sec. 5.1, (5)	Sec. 8.1.1, (5)	Sec. 7.1.1, (5)	Sec. 1.4.1.1, (1.4.1)	Sec. 8.1.1, (8.1)	Sec. 7.1, (1)
Resistance of a plate on beam webs under normal stresses	Fig. 9.5.3-1	Sec. 5.25*, (50)	Sec. 10.1.1, (91), Sec. 9.2.1 (35)	Sec. 8.2.1, (41), Sec. 9.1.1, (106)	Sec. 1.5.2.1, (1.5.1), Sec. 1.6.1.1, (1.6.2)	Sec. 9.2.1, (9.1), Sec. 10.1.1, (10.2)	Sec. 7.25, (48)
Resistance of a plate on beam webs under shear stresses	Fig. 9.5.3-1	Sec. 5.18*, (41)	Sec. 9.2.1, (36), (39)	Sec. 8.2.1, (42), (45)	Sec. 1.5.2.1, (1.5.2), (1.5.5)	Sec. 9.2.1, (9.2), (9.5)	Sec. 7.18, (39)
Resistance of a plate on beam webs under reduced stresses	Fig. 9.5.3-1	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4)	Sec. 9.2.1, (9.4)	Sec. 7.14, (29)
Notes: see Table 9.5.1-1.							

9.5.4 Truss Panel Points

The **Truss Panel Points** mode enables to design and check the joints of the truss with the bars made of double angles or rectangular (square) hollow sections. The mode implements a wide range of the types of joints:

- joint between a brace member and the truss chord (regular joint) (Fig. 9.5.4-1, 9.5.4-2);
- joint where the chord changes its cross-section (Fig. 9.5.4-4);
- erection joints (Fig. 9.5.4-5);
- support joints (Fig. 9.5.4-3, 9.5.4-6).

This mode performs the following checks in compliance with the codes:

- resistance of structural members (plates, bearing stiffeners (end-plates), gusset plates);
- resistance of welded connections (connections between the brace members and the gusset plates, connections between the chord members and the gusset plates, connection between the plates and the chord in the joints where the chord section changes and in erection joints, connections between the support gusset plate and the bearing stiffener (end plate) in support joints of the truss);
- resistance of bolted connections (connection between the bearing stiffener (end-plate) and the support structure);
- a number of structural and assortment constraints.

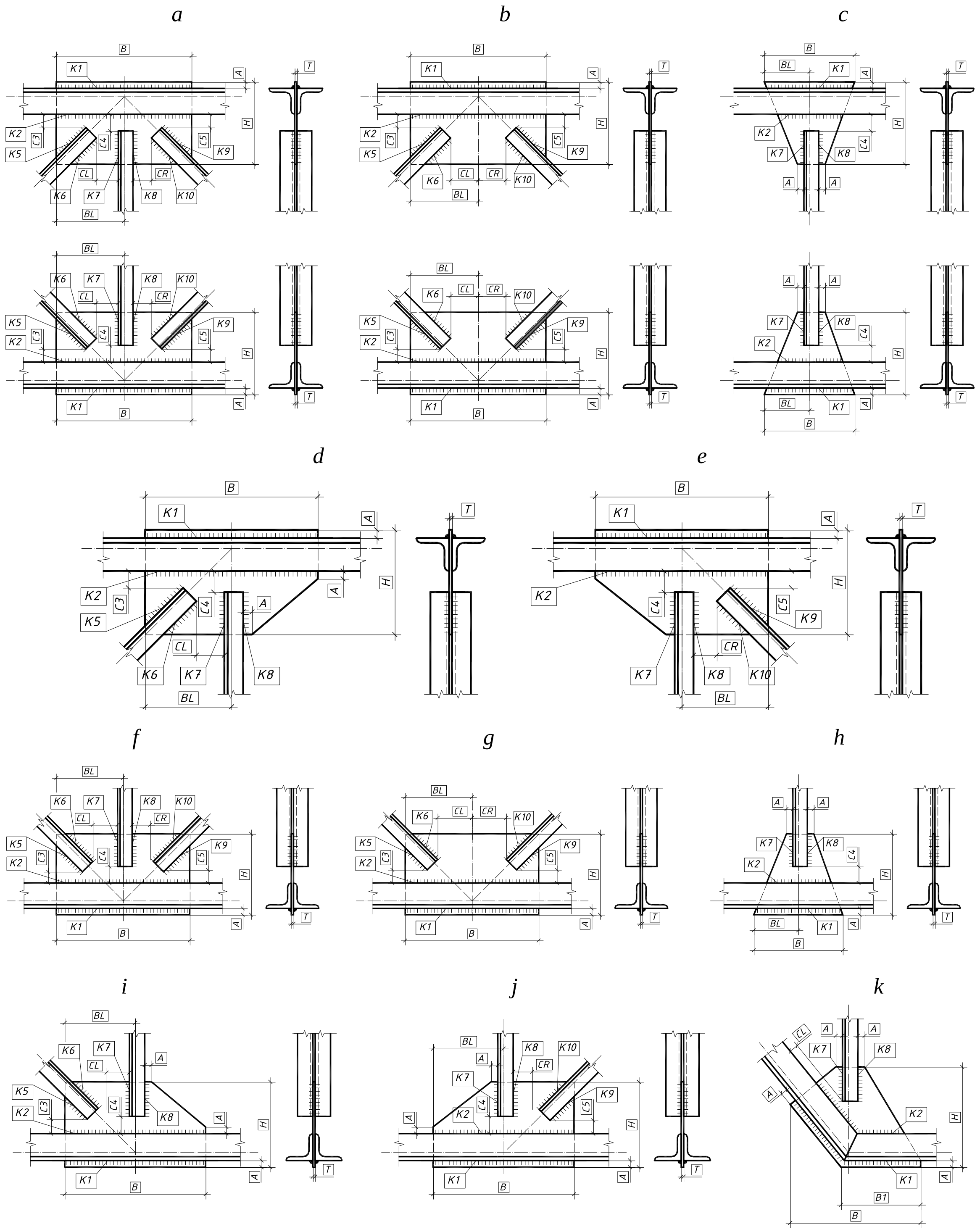


Figure 9.5.4-1. Regular joints in trusses with elements made of double angles

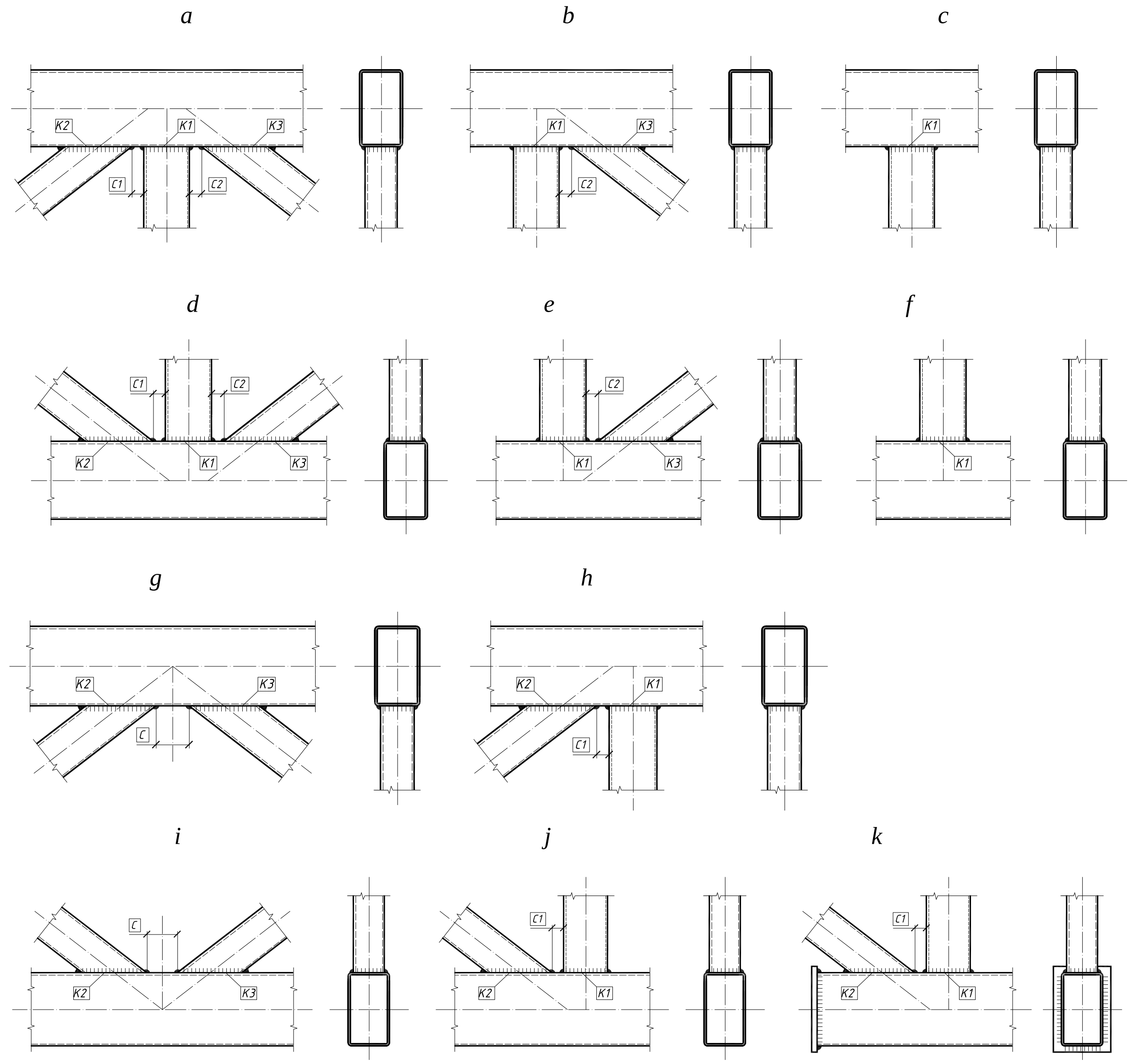


Figure 9.5.4-2. Regular joints in trusses with elements made of rectangular (square) hollow sections

a *b*

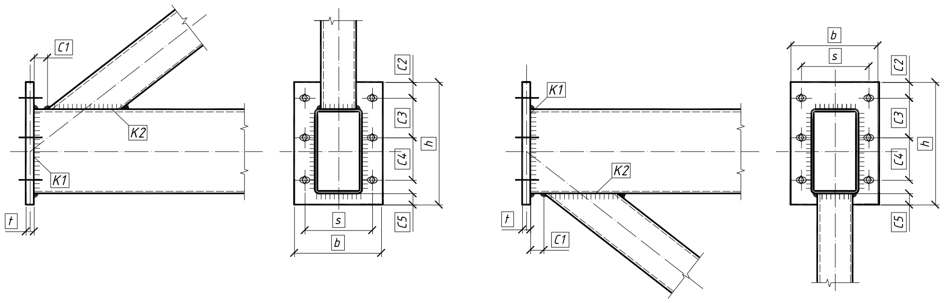


Figure 9.5.4-3. . Support joints in trusses with elements made of rectangular (square) hollow sections

a *b*

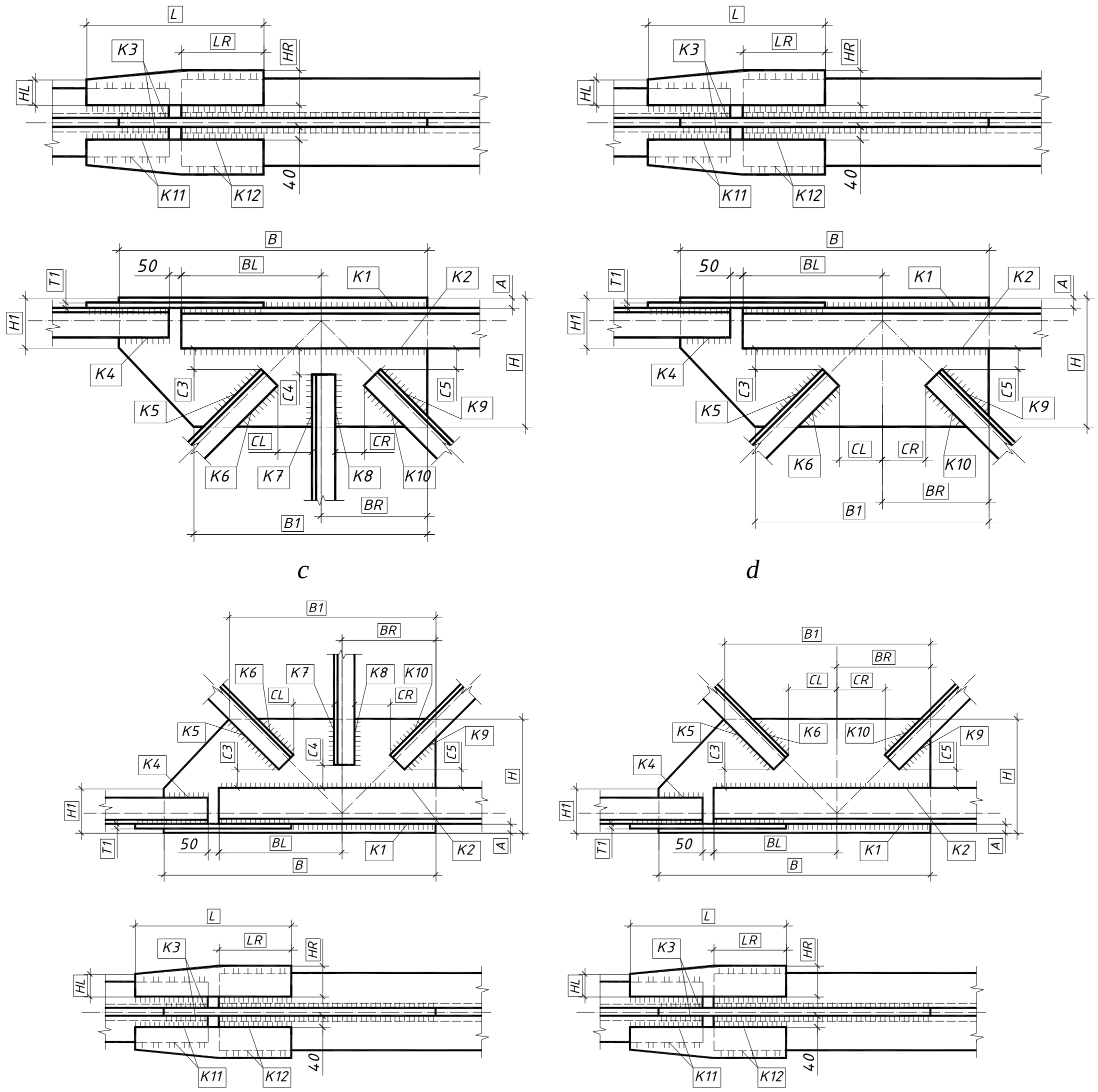
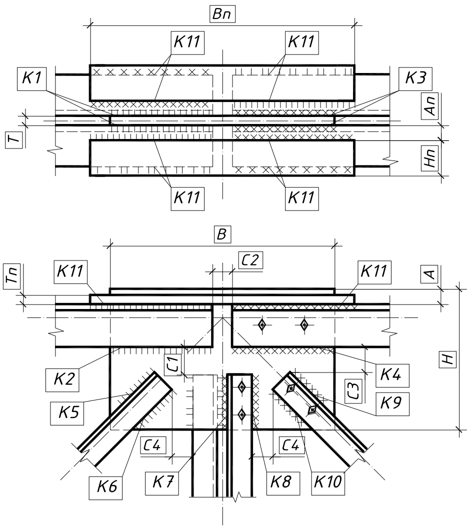
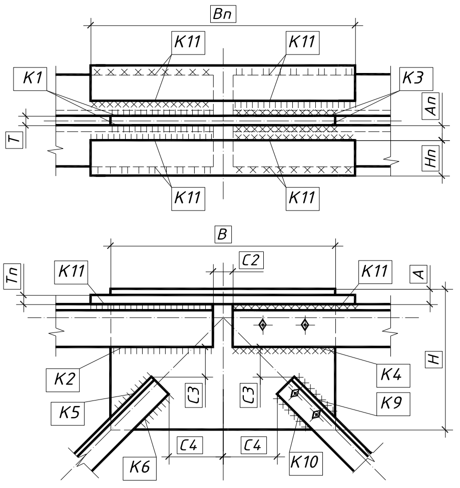


Figure 9.5.4-4. . Joints where the chord changes its cross-section in trusses with elements made of double angles

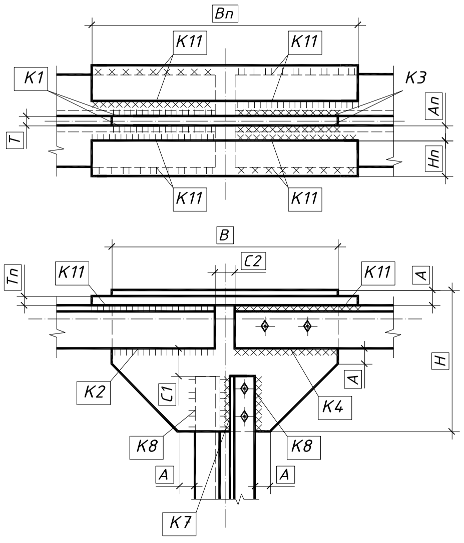
a


Bn
K11
K11
K1
K3
An
T
Hn
K11
K11
B
C2
Tn
K11
K11
A
H
K2
C1
K4
C3
K5
K9
C4
C4
K6
K7
K8
K10

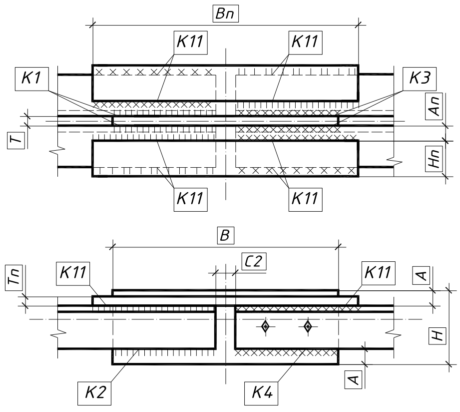
b


Bn
K11
K11
K1
K3
An
T
Hn
K11
K11
B
C2
Tn
K11
K11
A
H
K2
K4
C3
C3
K5
K9
C4
C4
K6
K10

c


Bn
K11
K11
K1
K3
An
T
Hn
K11
K11
B
C2
Tn
K11
K11
A
H
K2
C1
A
K4
K8
K8
A
A
K7

d


Bn
K11
K11
K1
K3
An
T
Hn
K11
K11
B
C2
Tn
K11
K11
A
H
K2
K4
A

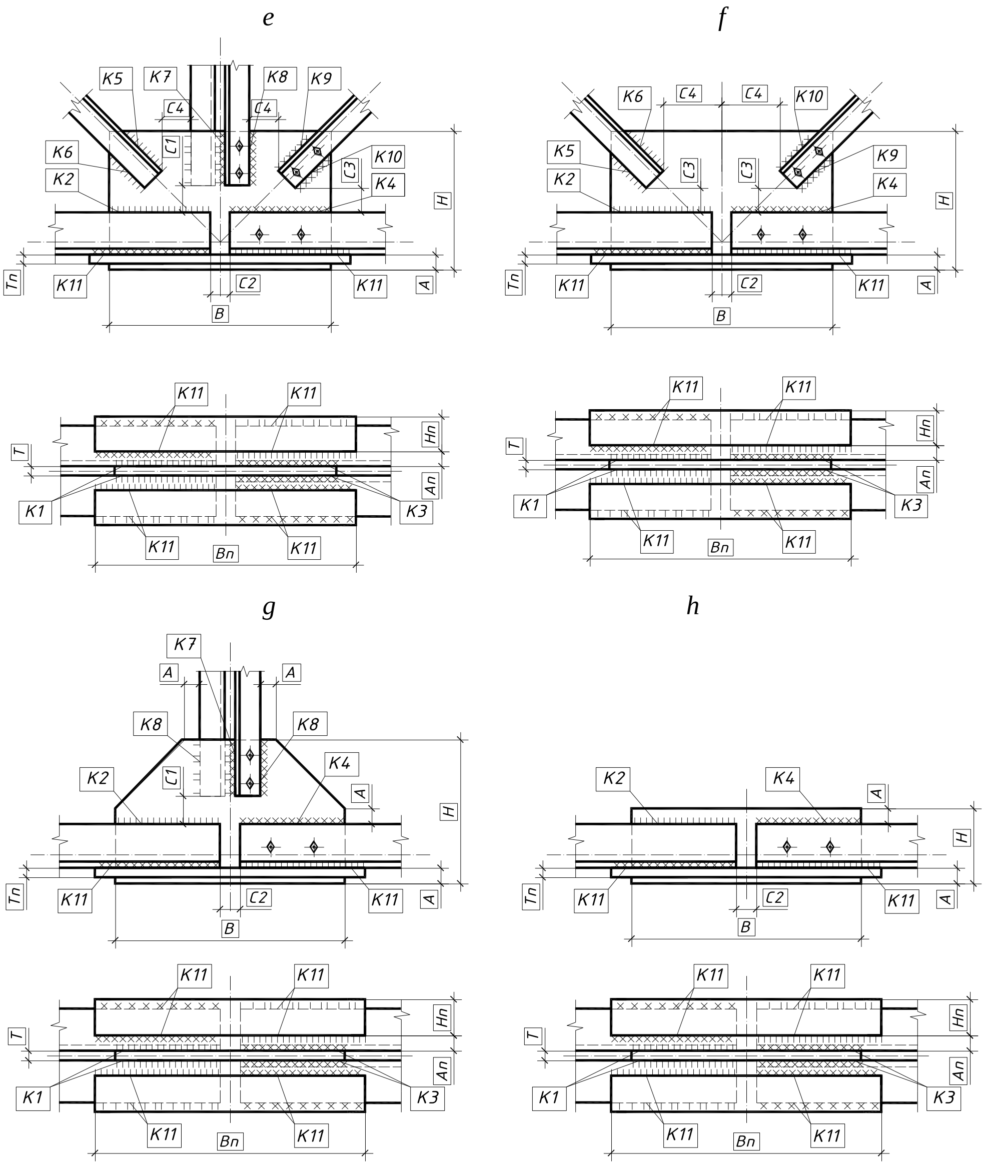


Figure 9.5.4-5. Erection joints in trusses with elements made of double angles

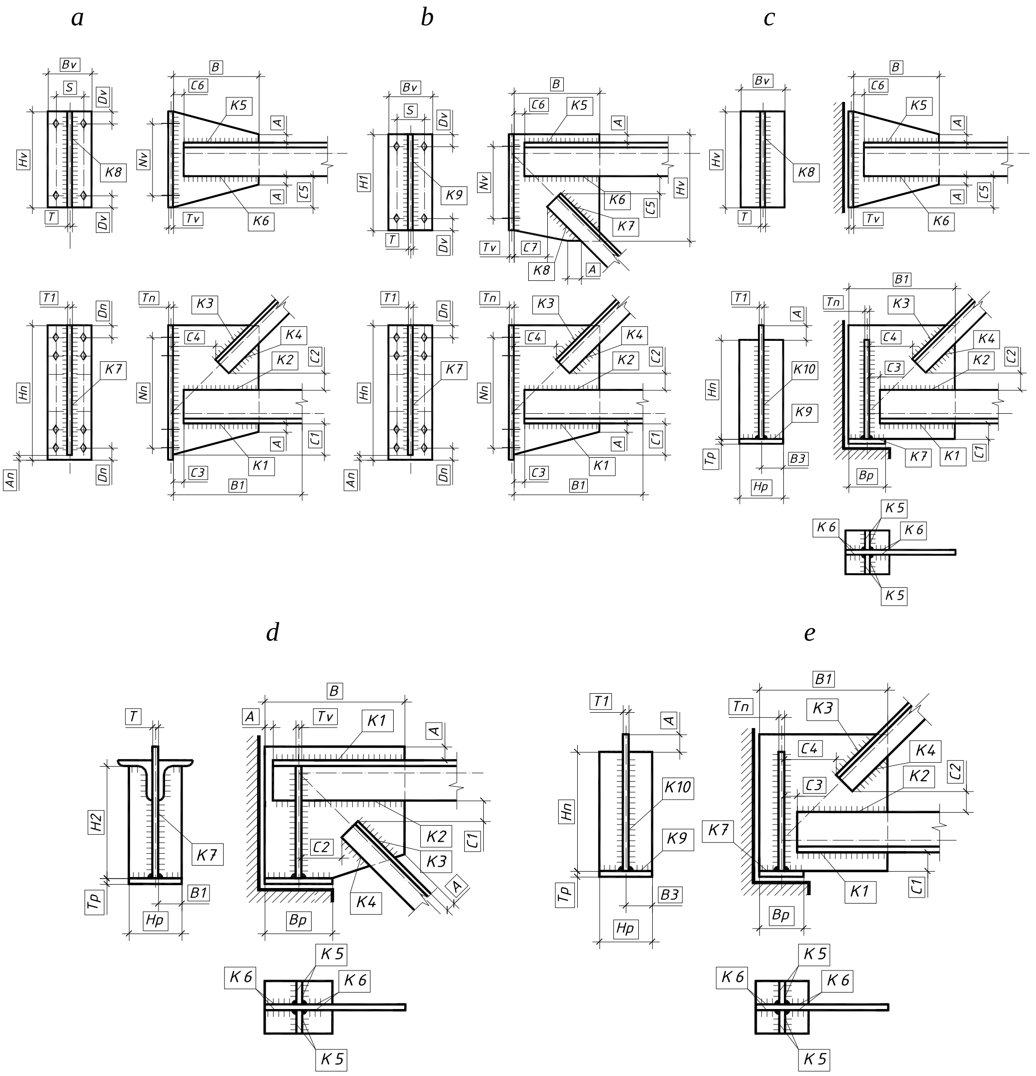


Figure 9.5.4-6. Support joints in trusses with elements made of double angles

The **Truss Panel Points** dialog box contains seven tabs: **Type of Joint** (Fig. 9.5.4-7), **Materials** (Fig. 9.5.4-8), **Forces** (Fig. 9.5.4-9), **Joint Members** (Fig. 9.5.4-10), **Structure** (Fig. 9.5.4-11), **Drawing** (Fig. 9.5.4-12) and **Interaction Curves** (Fig. 9.5.4-13).

The **Type of Joint** tab (Fig. 9.5.4-7) contains buttons which are used to select the type of a truss joint (**Regular Joint**, **Section Change**, **Erection Joint**, or **Support Joint**) and to specify its configuration. You can define the type of cross-section for the bar members of the truss by using the respective buttons in the **Structure** group. This tab also enables to specify a steel grade used for the load-bearing members of the truss (lattice members, chord members, etc.), and a steel grade used for the truss joint with angles (gusset plates). A steel grade for these members can be selected in the **Steel** dialog box (see Sec. 9.4.1) which is invoked by clicking the **Profile steel** and **Gusset plate steel** buttons.

The importance factor which will be further multiplied by the design values of all internal forces for all design combinations of loadings acting in the sections of the truss members adjacent to the considered joint has to be specified in the **Importance factor** drop-down list. If the values of the internal forces have been obtained in the result of the static analysis accounting for the importance factor (for example, when the design values of the loads were specified already multiplied by this factor), the value equal to one has to be selected in this drop-down list.

This tab also contains the **Stamp** button used to fill in the stamp of the drawing, which will be generated automatically once the structural design of the truss joint is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

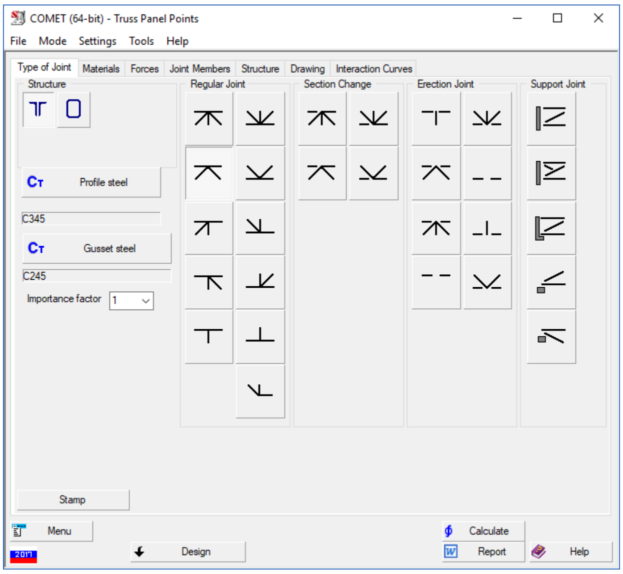


Figure 9.5.4-7. *The* **Type of Joint** *tab of the* **Truss Panel Points** *dialog box*

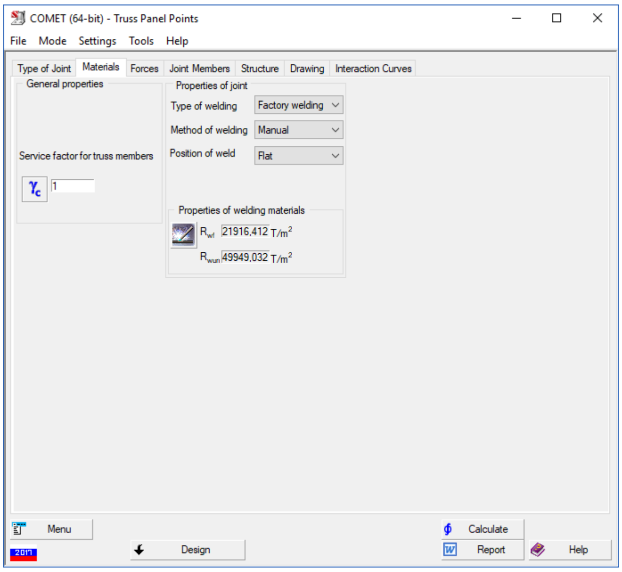


Figure 9.5.4-8. *The* **Materials** *tab of the* **Truss Panel Points** *dialog box*

The **Materials** tab (Fig. 9.5.4-8) enables to specify the parameters of the welded connections and bolted connections (only for the support joints) for the truss joint. The **Properties of joint** group contains drop-down lists which are used to select the type and method of welding, and specify the position of the weld. The **Truss Panel Points** mode implements the following methods of welding in compliance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, or Table 16.2 of DBN B.2.6-198:2014): manual welding, semiautomatic welding with solid wire less than 1.4 mm in diameter, automatic and semiautomatic welding with the electrode wire 1.4 to 2.0 mm in diameter, automatic welding with the electrode wire 3 to 5 mm in diameter, and semiautomatic welding with flux-cored wire. The position of weld can be underhand, flat, horizontal, vertical or overhead. The **Properties of welding materials** group displays values of the design resistance of the fillet welds for conventional shear of the weld metal, R_{wf} , and of the characteristic resistance of the weld metal, R_{wun} . These values can be specified in the **Materials for Welding** dialog box (see Sec. 9.4.4), which is invoked by clicking the button .

The **Forces** tab (Fig. 9.5.4-9) is used to specify axial forces N_i acting in the bar members of the truss joint. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current design combination of loads. There can be any number of design combinations of loads. The default units of measurement for axial forces are tonnes. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the truss members.

The **Joint Members** tab (Fig. 9.5.4-10) is used to specify the dimensions (width and height) of panels adjacent to the considered truss panel point (parameters *a*, *b*, *c*, and *d*). The default units of measurement for the dimensions of truss panels are meters. The **Section** group is used to specify the cross-sections of members connected in the considered joint and their orientation with respect to the truss plane.

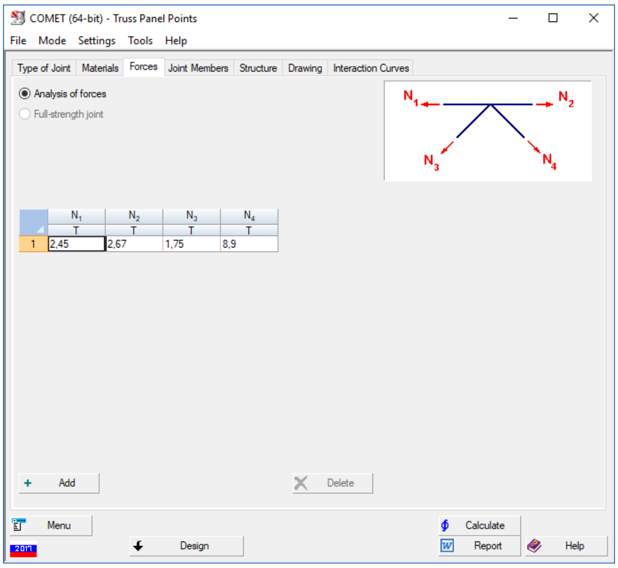


Figure 9.5.4-9. *The* **Forces** *tab of the* **Truss Panel Points** *dialog box*

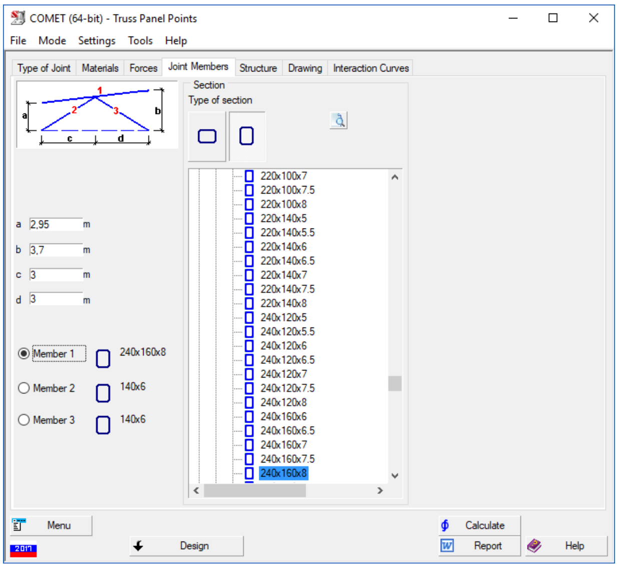


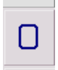
Figure 9.5.4-10. *The* **Joint Members** *tab of the* **Truss Panel Points** *dialog box*

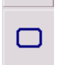
The **Type of section** buttons enable you to select:

 a section of double equal angles or of double unequal angles with the longer leg perpendicular to the truss plane;

 a section of double equal angles or of double unequal angles with the longer leg parallel to the truss plane;

 a section of equal angles arranged as a cross, which is used for the verticals of an erection joint;

 a rectangular hollow section with the longer side parallel to the truss plane;

 a rectangular hollow section with the longer side perpendicular to the truss plane.

It should be noted that the set of **Type of section** buttons depends on the design of the joint specified in the **Structure** group of the **Type of Joint** tab.

Each member of the truss joint is assigned a number (e.g., ). To assign profiles to the truss members, you have to select the respective radio button of the member and select a profile from the list of assortments of rolled profiles. The specified cross-sections of the members of the truss joint can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ().

You have to specify the position of the gusset plate by clicking the respective buttons in the **Position of the gusset plate** group. The thickness of the gusset plate can be selected from the respective drop-down list, which provides a set of thickness values according to the assortment of sheet and plate steel.

The **Structure** tab (Fig. 9.5.4-11, 9.5.4-12) provides a draft of the design for the truss joint. A steel grade used for the members of the truss joint (plates – in the joints where the chord changes its cross-section and in the erection joints; end-plates, base plates, and bearing stiffeners – in the support joints) can be specified in the bottom of this tab. A steel grade for these members can be selected in the **Steel** dialog box, which is invoked by clicking the respective buttons.

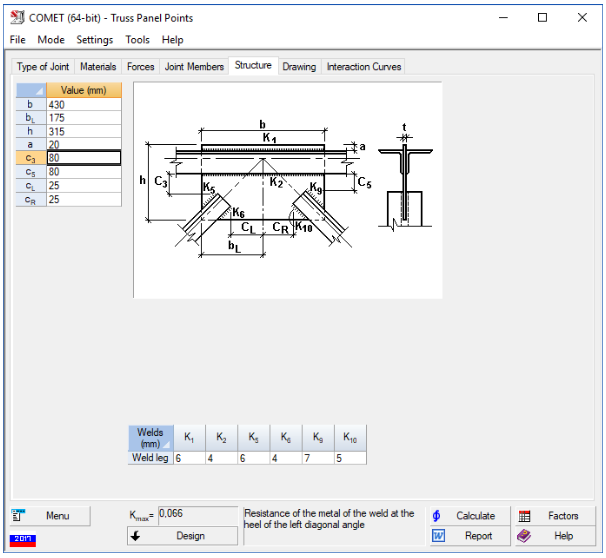


Figure 9.5.4-11. *The* **Structure** *tab of the* **Truss Panel Points** *dialog box* (erection joint)

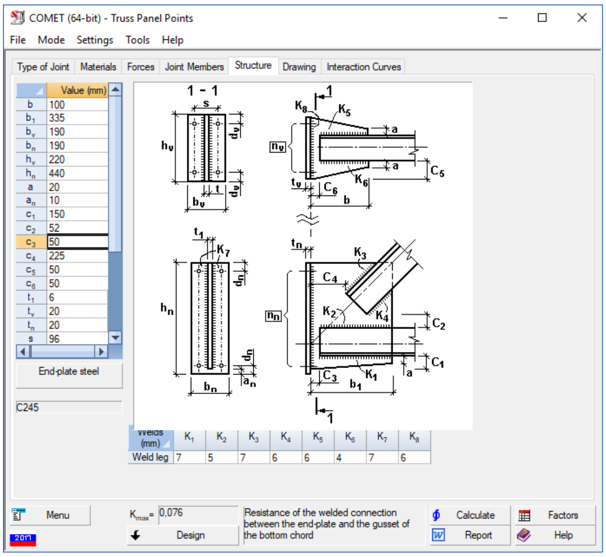


Figure 9.5.4-12. *The* **Structure** *tab of the* **Truss Panel Points** *dialog box* (support joint)

To perform a check of the load-bearing capacity of the specified structural design of the truss joint, you have to specify all its design parameters. The parameters include the sizes and thickness of structural members of the joint, leg lengths of welds, sizes which determine the mutual arrangement of members, diameters of bolts, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table on the left. Leg lengths of the fillet welds are entered in the table at the bottom of the dialog. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu (see Sec. 9.5.1). If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the truss members adjacent to the considered joint, structural members of the joint (plates, end-plates, bearing stiffeners, bearing end-plates, etc.), welded and bolted connections in the joint at the specified (or previously selected) values of the parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the truss joint design of the MS stage will be generated.

Once you switch to the **Drawing** tab (Fig. 9.5.4-13), the application performs a check of the truss joint similarly to the **Calculate** mode. If the results of the check of the parameters of the joint elements do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

.

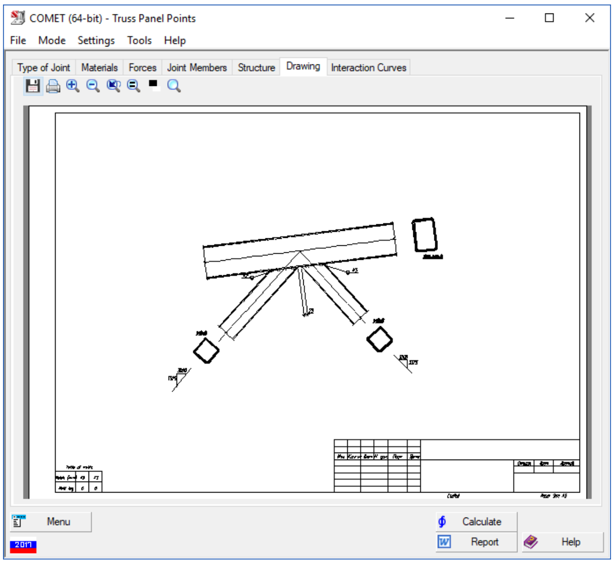


Figure 9.5.4-13. *The* **Drawing** *tab of the* **Truss Panel Points** *dialog box*

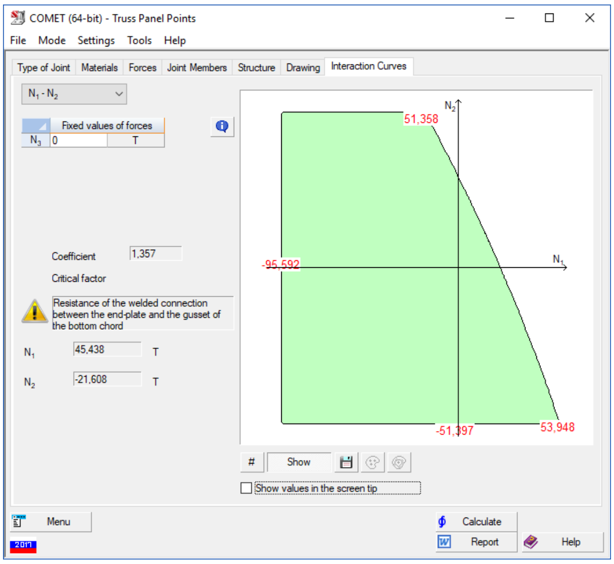


Figure 9.5.4-14. *The* **Interaction Curves** *tab of the* **Truss Panel Points** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the truss joint under various pairs of internal forces which can arise in the truss members adjacent to the joint are plotted in the **Interaction Curves** tab (Fig. 9.5.4-14). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of truss joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

A complete list of checks and values of the respective utilization factors of restrictions can be obtained by clicking the **Factors** button. The list of the load-bearing capacity checks of the members and connections of the truss joints performed by the application is given in Table 9.5.4-1.

Table 9.5.4-1. A list of the load-bearing capacity checks of the members and connections of the truss joints

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Regular joints in trusses with elements made of double angles:	**Fig. 9.5.4-1**						
Resistance of the welded connection between the truss chord and the gusset plate	*a, b, c, d, e, f, g, h, i, j, k*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the welded connection between the vertical and the gusset plate	*a, c, d, e, f, h, i, j, k*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the left diagonal and the gusset plate	*a, b, d, f, g, i, k*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the right diagonal and the gusset plate	*a, b, e, f, g, j*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Regular joints in trusses with elements made of rectangular (square) hollow sections:	**Fig. 9.5.4-2**						
Punching shear resistance of the chord web	*a, b, c, d, e, f, g, h, i, j, k*	Sec. 15.10, 15.11, (92), (94) [3]	Sec. R.2.2, (R.1), Sec. R.2.3, (R.2)	Sec. L.2.2, (L.1), Sec. L.2.3, (L.2) Sec. 14.3.2 (86), (87) [139]		Sec. U.1.2, (U.1), Sec. U.1.3, (U.2)	Sec. M.6, M.7, (M.7), (M.8)
Resistance of the chord web in the joint plane at the connection with the vertical	*a, b, c, d, e, f, h, j, k*	Sec. 15.12, (95) [3]	Sec. R.2.4, (R.3)	Sec. L.2.4, (L.3) Sec. 14.3.2 (88) [139]		Sec. U.1.4, (U.3)	Sec. M.8 (M.9)
Resistance of the chord web in the joint plane at the connection with the left diagonal	*a, d, g, h, i, , j, k*	Sec. 15.12, (95) [3]	Sec. R.2.4, (R.3)	Sec. L.2.4 (L.3) Sec. 14.3.2 (88) [139]		Sec. U.1.4, (U.3)	Sec. M.8 (M.9)
Resistance of the chord web in the joint plane at the connection with the left diagonal	*a, b, d, e, g, i*	Sec. 15.12, (95) [3]	Sec. R.2.4, (R.3)	Sec. L.2.4 (L.3) Sec. 14.3.2 (88) [139]		Sec. U.1.4, (U.3)	Sec. M.8 (M.9)
Resistance of the vertical at the connection with the chord	*a, b, c, d, e, f, h, , j, k*	Sec. 15.13, (96), (97) [3]	Sec. R.2.5, (R.4), (R.5)	Sec. L.2.5 (L.4), (L.5) Sec. 14.3.2 (89), (90) [139]		Sec. U.1.5, (U.4), (U.5)	Sec. M.9, (M.10), (M.11)
Resistance of the left diagonal at the connection with the chord	*a, d, g, h, i, , j, k*	Sec. 15.13, (96), (97) [3]	Sec. R.2.5, (R.4), (R.5)	Sec. L.2.5 (L.4), (L.5) Sec. 14.3.2 (89), (90) [139]		Sec. U.1.5, (U.4), (U.5)	Sec. M.9, (M.10), (M.11)

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the right diagonal at the connection with the chord	*a, b, d, e, g, i*	Sec. 15.13, (96), (97) [3]	Sec. R.2.5, (R.4), (R.5)	Sec. L.2.5 (L.4), (L.5) Sec. 14.3.2 (89), (90) [139]		Sec. U.1.5, (U.4), (U.5)	Sec. M.9, (M.10), (M.11)
Resistance of the welded connection between the vertical and the chord	*a, b, c, d, e, f, h, , j, k*	Sec. 15.14, (98), (99) [3]	Sec. R.2.6, (R.6), (R.7)	Sec. L.2.6 (L.6), (L.7) Sec. 14.3.2 (91), (92) [139]		Sec. U.1.6, (U.6), (U.7)	Sec. M.10 , (M.12), (M.13)
Resistance of the welded connection between the left diagonal and the chord	*a, d, g, h, i, , j, k*	Sec. 15.14, (98), (99) [3]	Sec. R.2.6, (R.6), (R.7)	Sec. L.2.6 (L.6), (L.7) Sec. 14.3.2 (91), (92) [139]		Sec. U.1.6, (U.6), (U.7)	Sec. M.10 , (M.12), (M.13)
Resistance of the welded connection between the right diagonal and the chord	*a, b, d, e, g, i*	Sec. 15.14, (98), (99) [3]	Sec. R.2.6, (R.6), (R.7)	Sec. L.2.6 (L.6), (L.7) Sec. 14.3.2 (91), (92) [139]		Sec. U.1.6, (U.6), (U.7)	Sec. M.10 , (M.12), (M.13)
Support joints in trusses with elements made of rectangular (square) hollow sections:	**Fig. 9.5.4-3**						
Local bearing resistance of the bearing stiffener	*a, b*	Sec. 5.38	Sec. 16.12	Sec. 15.12	Sec. 1.8	Sec. 12	Sec. 7.38
Stability of the bearing stiffener	*a, b*	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17, Sec. 7.1.3, (7)	Sec. 1.5.5.13, Sec. 1.4.1.3, (1.4.3)	Sec. 9.5.13, Sec. 8.1.3, (8.3)	Sec. 9.13
Punching shear resistance of the chord web	*a, b*	Sec. 15.10, 15.11, (92), (94) [3]	Sec. R.2.2, (R.1), Sec. R.2.3, (R.2)	Sec. L.2.2, (L.1), Sec. L.2.3, (L.2) Sec. 14.3.2 (86), (87) [139]			Sec. M.6, M.7, (M.7), (M.8)
Resistance of the chord web in the joint plane at the connection with the support diagonal	*a, b*	Sec. 15.12, (95) [3]	Sec. R.2.4, (R.3)	Sec. L.2.4, (L.3) Sec. 14.3.2 (88) [139]			Sec. M.8 (M.9)
Resistance of the support diagonal at the connection with the chord	*a, b*	Sec. 15.13, (96), (97) [3]	Sec. R.2.5, (R.4), (R.5)	Sec. L.2.5, (L.4), (L.5) Sec. 14.3.2 (89), (90) [139]			Sec. M.9, (M.10), (M.11)
Resistance of the welded connection between the chord and the bearing stiffener	*a, b*	Sec. 11.2, (120), (121) [3]	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the welded connection between the support diagonal and the chord	*a*, *b*	Sec. 15.14, (98), (99) [3]	Sec. R.2.6, (R.6), (R.7)	Sec. L.2.6, (L.6), (L.7) Sec. 14.3.2 (91), (92)			Sec. M.10, (M.12), (M.13)
Joints where the chord changes its cross-section in trusses with elements made of double angles:	**Fig. 9.5.4-4**						
Resistance of the welded connection between the truss chord and the gusset plate	*a*, *b*, *c*, *d*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the vertical and the gusset plate	*a*, *c*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the left diagonal and the gusset plate	*a*, *b*, *c*, *d*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the right diagonal and the gusset plate	*a*, *b*, *c*, *d*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the plate	*a*, *b*, *c*, *d*	Sec. 5.1, (5)	Sec. 8.1.1, (5)	Sec. 7.1.1, (5)	Sec. 1.4.1.1, (1.4.1)	Sec. 8.1.1, (8.1)	Sec. 7.1, (1)
Resistance of the welded connection between a plate and a chord flange with a greater cross-section	*a*, *b*, *c*, *d*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between a plate and a chord flange with a smaller cross-section	*a*, *b*, *c*, *d*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Erection joints in trusses with elements made of double angles:	**Fig. 9.5.4-5**						

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the welded connection between the chord of the left truss panel and the gusset plate	*a, b, c, d, e, f, g, h*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the chord of the right truss panel and the gusset plate	*a, b, c, d, e, f, g, h*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the vertical of the left truss panel and the gusset plate	*a, c, e, g*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the vertical of the right truss panel and the gusset plate	*a, c, e, g*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec.1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the left diagonal and the gusset plate	*a, b, e, f*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the right diagonal and the gusset plate	*a, b, e, f*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the plate	*a, b, c, d, e, f, g, h*	Sec. 5.1, (5)	Sec. 8.1.1, (5)	Sec. 7.1.1, (5)	Sec.1.4.1.1, (1.4.1)	Sec. 8.1.1, (8.1)	Sec. 7.1, (1)
Resistance of the welded connection between the plate and the chord flange of the left truss panel	*a, b, c, d, e, f, g, h*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the plate and the chord flange of the right truss panel	*a, b, c, d, e, f, g, h*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Support joints in trusses with elements made of double angles:	**Fig. 9.5.4-6**						

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Shear and bearing strength of bolts	*e*	Sec. 11.7*, (127), (128), Sec. 11.8, (130)	Sec. 15.2.9, (165), (166), Sec. 15.2.10, (168)	Sec. 14.2.9, (186), (187), Sec. 14.2.10, (189)	Sec. 1.12.2.9, (1.12.12), (1.12.13), Sec. 1.12.2.10, (1.12.15)	Sec. 16.2.9, (16.12), (16.13), Sec. 16.2.10, (16.15)	Sec. 13.7, (136), (137), Sec. 13.8, (139)
Tensile strength of bolts	*a*, *b*, *e*	Sec. 11.7*, (129), Sec. 11.8, (130)	Sec. 15.2.9, (165), (166), Sec. 15.2.10, (168)	Sec. 14.2.9, (188), Sec. 14.2.10, (189)	Sec. 1.12.2.9, (1.12.12), (1.12.13), Sec. 1.12.2.10, (1.12.15)	Sec. 16.2.9, (16.14), Sec. 16.2.10, (16.15)	Sec. 13.7, (138), Sec. 13.8, (139)
Bending resistance of the end-plate	*a*, *b*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.2.1, (41)	Sec. 1.5.2.1, (1.5.1)	Sec. 9.2.1, (9.1)	Sec. 7.12, (24)
Local bearing resistance of the bearing stiffener	*e*	Sec. 5.38	Sec. 16.12	Sec. 15.12	Sec. 1.8	Sec. 12	Sec. 7.38
Stability of the bearing stiffener	*e*	Sec. 7.12	Sec. 9.5.13	Sec. 8.5.17, Sec. 7.1.3, (7)	Sec. 1.5.5.13, Sec. 1.4.1.3, (1.4.3)	Sec. 9.5.13, Sec. 8.1.3, (8.3)	Sec. 9.13
Local stability of the overhangs of the bearing stiffener flanges	*e*	Sec. 7.23*, Table 29*	Sec. 8.3.7, (31)	Sec. 7.3.8, (37)	Sec. 1.4.3.7, (1.4.27)	Sec. 8.3.7, (8.27)	Sec. 9.23
Resistance of the welded connection between the chord and the support gusset	*a*, *b*, *c*, *d*, *e*, *f*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the support diagonal and the support gusset	*b*, *c*, *d*, *e*, *f*	Sec. 11.2, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the bearing stiffener and the support gusset	*d*, *e*, *f*	Sec. 11.2, (120), (121), Sec. 11.3*, (122), (123), Sec. 11.5, (126)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, (129), (130), Sec. 13.3, (131), (132), Sec. 13.5, (135)
Resistance of the welded connection between the end-plate and the support gusset	*a*, *b*, *c*	Sec. 11.2, (120), (121), Sec. 11.3*, (122), (123), Sec. 11.5, (126)	Sec. 15.1.16, (155), (156),Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, (129), (130), Sec. 13.3, (131), (132), Sec. 13.5, (135)

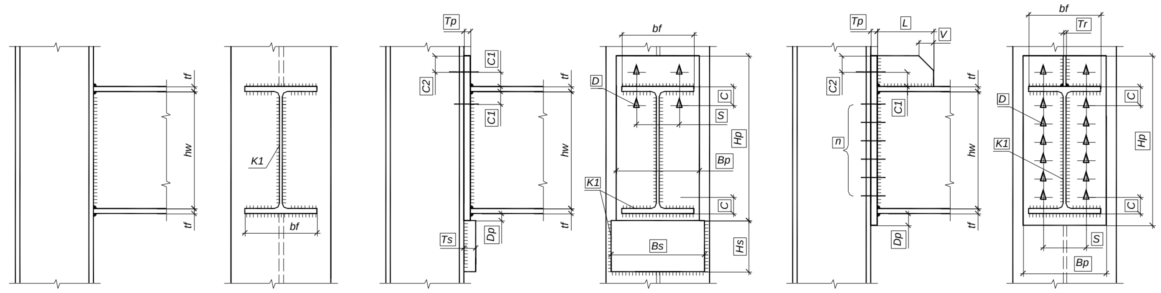
Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the base plate	*d*, *f*	Sec. 5.12, (28)	Sec. 9.6.2 (86-88)	Sec. 8.6.2 (101-103)	Sec. 1.7.2, (1.7.1), Annex N, (N.1), (N2), Table N.2	Sec. 11.2, (11.1), Annex. M, Sec. M.2, (M.1), (M.2)	Sec. 7.12, (24)
Notes: see Table 9.5.1-1.							

9.5.5 Beam-To-Column Joints

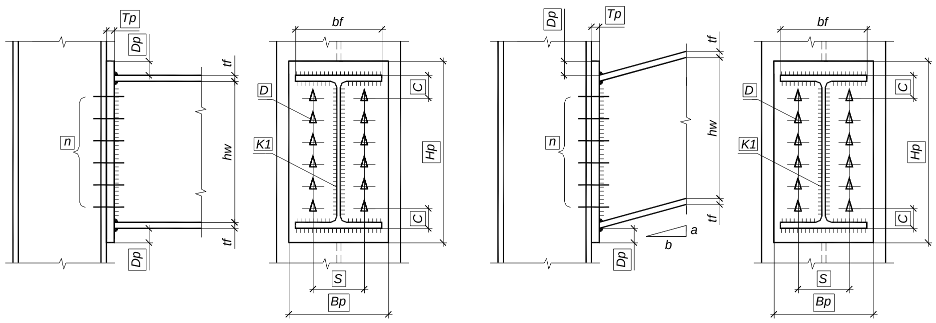
The **Beam-To-Column Joints** mode enables to design and check the load-bearing capacity of beam-to-column joints. The considered joints can be classified into the following types by the conditions of resistance to internal forces acting in the joint and by the possibility of the mutual rotation of the beam with respect to the column:

- rigid ones, which nearly immobilize the beam with respect to the column (Fig. 9.5.5-1);
- pinned ones, which can hardly resist the rotation of the beam with respect to the column (Fig. 9.5.5-2).

a *b* *c*



d *e*



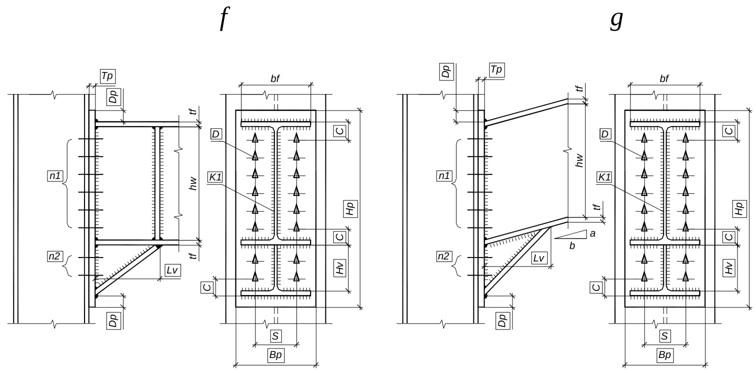


Figure 9.5.5-1. Types of designs for the rigid beam-to-column joints

Fig. 9.5.5-1 presents the following types of structural designs for rigid beam-to-column joints implemented in the application: a welded joint (Fig. 9.5.5-1, *a*) and joints with high strength bolts (Fig. 9.5.5-1, *b*...*g*). Structural designs of the beam-to-column joints which use a bearing end-plate without an angle cleat (Fig. 9.5.5-1, c…g) are usually developed as friction joints with high strength bolts. In cases when a significant bending moment acts in the joint, and its value exceeds the resistance of the beam, the application provides the designs with haunches (Fig. 9.5.5-1, f, g).

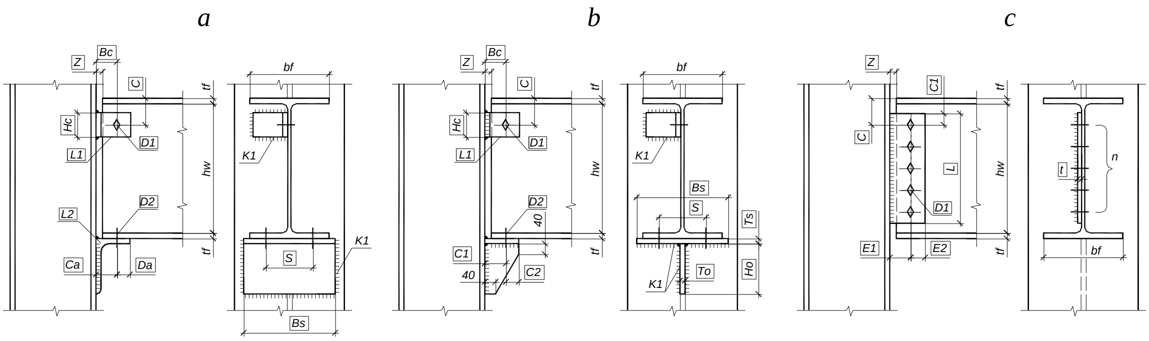


Figure 9.5.5-2. Types of designs for the pinned beam-to-column joints

Pinned beam-to-column joints implemented in the application are shown in Fig. 9.5.5-2. Their structural designs use an angle cleat (Fig. 9.5.5-2, *a* and 9.5.5-2, *b*), and a plate on the beam web (Fig. 9.5.5-2, *c*).

This mode performs the following checks in compliance with the codes:

- resistance of end-plates, stiffeners and haunches;
- resistance of bolted and welded connections included in the joint;
- a number of structural and assortment constraints.

The **Beam-To-Column Joints** dialog box contains the following tabs: **Configuration** (Fig. 9.5.5-3), **Forces** (Fig. 9.5.5-6), **Beam 1** (Fig. 9.5.5-7), **Beam 2** (Fig. 9.5.5-8), **Welding (Beam 1)**, **Welding (Beam 2)**, **Drawing,** and **Interaction Curves** (Fig. 9.5.5-9).

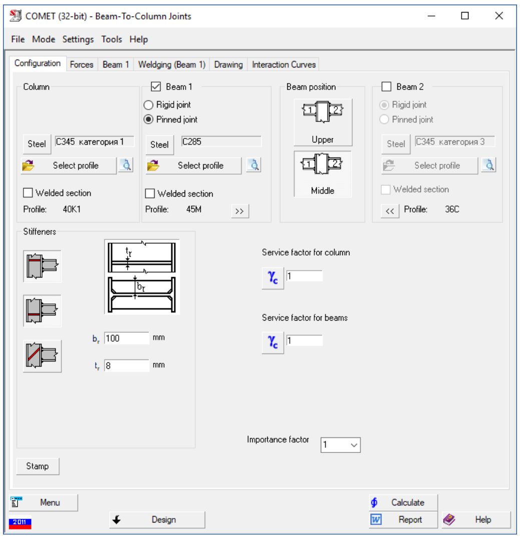


Figure 9.5.5-3. *The* **Configuration** *tab of the* **Beam-To-Column Joints** dialog box

First you have to define the configuration of your joint in the **Configuration** tab (Fig. 9.5.5-3). If the beam is connected to the column on the left, you have to check the checkbox () **Beam 1**, and if it is connected to the column on the right, check the **Beam 2** checkbox. If there are 2 beams in the joint (both on the left and on the right), both checkboxes (**Beam 1** and **Beam 2**) have to be checked. The group of buttons **Beam position** is used to define the position of beams in their connection to the column. The upper position of the beams corresponds to a joint between the roof beams and the head of the column, and the middle position corresponds to the joint between the floor beams and the column.

The **Column** group of controls is used to define the cross-section and the steel grade for the column. A steel grade for the column can be selected by clicking the **Steel** button. If the column cross-section type is a welded I-section, check the **Welded section** checkbox and click the **Select profile** button. The **Sizes of section** dialog box will appear (Fig. 9.5.5-4) where you have to specify the sizes of the column cross-section: height h_w and thickness t_w of the column web, width b_f and thickness t_f of the column flange; and then click the **OK** button. The default units of measurement for the sizes of the column cross-section are millimeters.

The thickness of the flanges and of the web can be either entered manually or selected from the drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel. If the column cross-section type is a rolled one, click the **Select profile** button (the **Welded section** checkbox should be unchecked). Select an assortment and the profile number in this assortment from the tree-like list in the **Select profile** dialog box (Fig. 9.5.5-5), and then click the **OK** button.

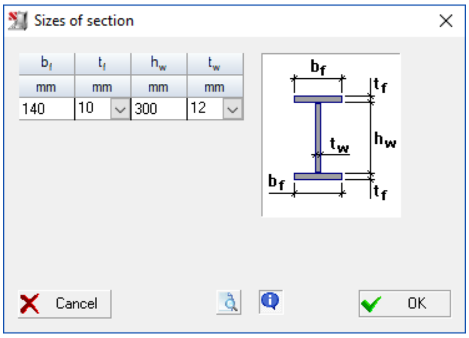


Figure 9.5.5-4. *The* **Sizes of section** *dialog box*

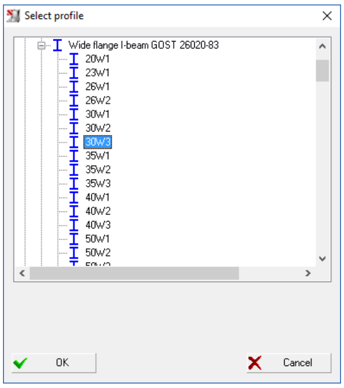
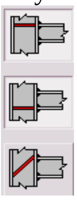


Figure 9.5.5-5. *The* **Select profile** *dialog box*

The specified column cross-section can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ().

In the same way you can define the steel grade and the cross-section for the beams of the joint using the controls of the respective groups (**Beam 1** and **Beam 2**). You also have to specify the type of connection for each beam using the respective radio buttons: **Rigid joint** or **Pinned joint**. Clicking the button >> in the **Beam 1** group will assign the type and sizes of the cross-section selected for beam 1 (on the left) to beam 2 (on the right). Clicking the button << in the **Beam 2** group will assign the type and sizes of the cross-section specified for beam 2 to beam 1.

You can enter the values of service factors for the column and beams of the joint in the respective text fields. The values of these factors can be also selected in the **Service Factor** dialog box (see Sec. 9.4.5) after clicking the nearby button.



When there are significant bending moments acting in the beam-to-column joints, it often becomes necessary to additionally reinforce the column web by transverse stiffeners. The arrangement of the stiffeners of the column web is specified by clicking one of the buttons of the **Stiffeners** group, which define their arrangement in the level of the upper beam flange, in the level of the lower beam flange, or obliquely. The sizes of the stiffeners b_r and t_r have to be specified in the respective text fields.

The importance factor which will be further multiplied by the design values of all internal forces for all design combinations of loadings acting in the sections of the members (columns and beams) connected in the considered joint has to be specified in the **Importance factor** drop-down list. If the values of the internal forces have been obtained in the result of the static analysis accounting for the importance factor (for example, when the design values of the loads were specified already multiplied by this factor), the value equal to one has to be selected in this drop-down list.

Clicking the **Stamp** button opens a dialog box which enables to fill in the stamp of the drawing, which will be generated automatically once the structural design of the beam-to-column joint is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

The **Welding (Beam 1)** and **Welding (Beam 2)** tabs enable to specify the parameters of the welded connections for the joint. The **Properties of joint** group contains drop-down lists which are used to select the type and method of welding, and specify the position of the weld. The **Truss Panel Points** mode implements the following methods of welding in compliance with Table 34* of SNiP II-23-81* (Table 36 of SP 53-102-2004, Table 38 of SP 16.13330, Table 1.12.2 of DBN B.2.6-163:2010, or Table 16.2 of DBN B.2.6-198:2014): manual welding, semiautomatic welding with solid wire less than 1.4 mm in diameter, automatic and semiautomatic welding with the electrode wire 1.4 to 2.0 mm in diameter, automatic welding with the electrode wire 3 to 5 mm in diameter, and semiautomatic welding with flux-cored wire. The position of weld can be underhand, flat, horizontal, vertical or overhead. The **Properties of welding materials** group displays values of the design resistance of the fillet welds for conventional shear of the weld metal, R_{wf}, and of the characteristic resistance of the weld metal, R_{wun}. These values can be specified in the **Materials for Welding** dialog box (see Sec. 9.4.4), which is invoked by clicking the button.

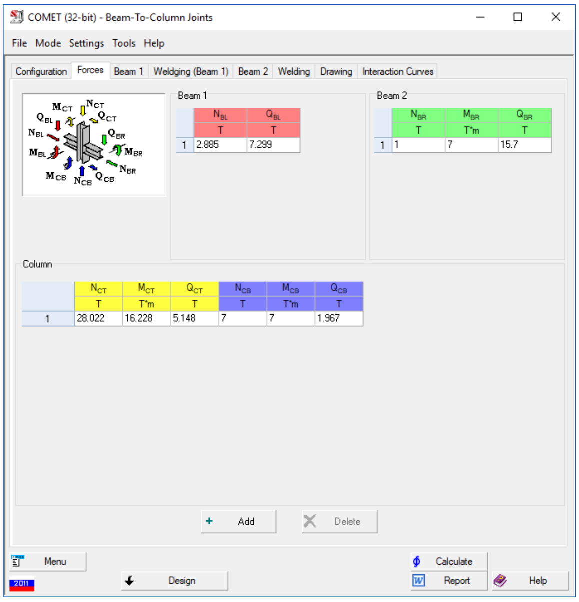


Figure 9.5.5-6. *The* **Forces** tab
of the **Beam-To-Column Joints** dialog box

The **Forces** tab (Fig. 9.5.5-6) is used to specify the internal forces acting in the beam-to-column joint. In the general case, any joint is subjected to an axial force *N*, bending moment *M* and its respective shear force *Q* transferred to it from the beam. It should be noted that in a pinned beam-to-column joint the bending moment, *M*, must be equal to zero. Moreover, the following forces are acting in the design column sections: axial force *N*, bending moments in both planes M_x, M_y, and their respective shear forces Q_y and Q_x. Forces acting in the design column sections are specified for the cross-sections above and below the considered joint.

Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the design values of internal forces for the current combination of loads. There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter. The drawing in the top left corner of the tab defines the positive directions of internal forces.

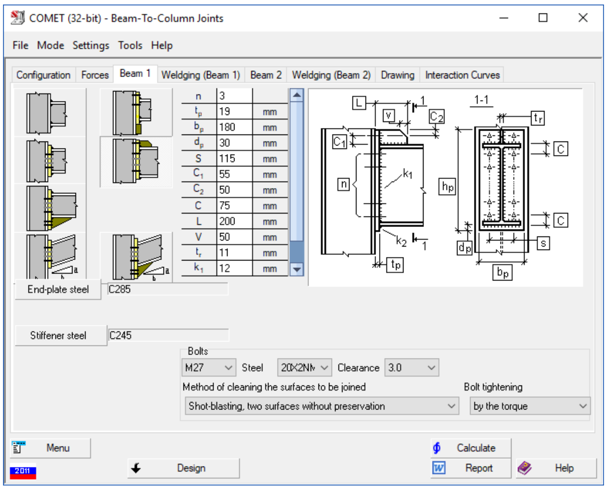


Figure 9.5.5-7. *The* **Beam 1** *tab*
of the **Beam-To-Column Joints** *dialog box*
(rigid joint between beam 1 and the column)

The **Beam 1** tab (Fig. 9.5.5-7) contains a group of buttons to select a design for the beam-to-column joint (the beam is on the left of the column). If a rigid joint has been specified for beam 1 in the **Configuration** tab, the **Beam 1** tab will display structural designs for rigid beam-to-column joints. It should be noted that rigid joints of load-bearing members are made using end-plate connections with high strength bolts or using welded connections with fillet welds. A steel grade of end-plates can be specified for end-plate beam-to-column joints by clicking the respective button.

A slope of the beam can be specified for some types of beam-to-column joints (Fig. 9.5.5-1, e, g) by entering the values of dimensionless parameters *a* and *b* (Fig. 9.5.5-1, e, g) in the **Slope** group (Fig. 9.5.5-9).

If high strength bolts are used, you have to specify the following data using the respective drop-down lists of the **Bolts** group (see Fig. 9.5.5-7): the steel grade of the bolts, their clearance (the difference between the diameter of the bolt hole and that of the bolt itself), a bolt tightening control method, and a method of cleaning the surfaces to be joined. If the diameter of high strength bolts used in the joint is known, you have to select their grade (diameter) from the respective drop-down list. When bolts of normal and improved strength are used, you have to select the strength class of the bolts and, if their diameter is known, the grade (diameter) from the respective drop-down lists.

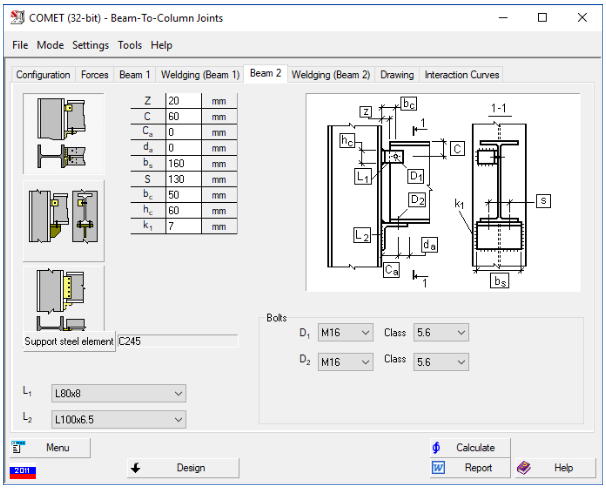


Figure 9.5.5-8. *The* **Beam 2** *tab of the* **Beam-To-Column Joints** *dialog box* (pinned joint between beam 2 and the column)

The **Beam 2** tab (Fig. 9.5.5-8) contains a group of buttons to select a design for the beam-to-column joint (the beam is on the right of the column). If a pinned joint has been specified for beam 2 in the **Configuration** tab, the **Beam 2** tab will display structural designs for pinned beam-to-column joints. The joints are made with bolts of normal strength (for joints with an angle cleat, Fig. 9.5.5-2, *a*, *b*) and with high strength bolts (for joints without an angle cleat, Fig. 9.5.5-2, *c*).

Equal angles according to GOST 8509-86 are used to fix the position of the beam with respect to the column and also as angle cleats in some types of pinned joints (Fig. 9.5.5-2, *a*, *b*). If the user knows the sizes of these angles, they can be specified in the drop-down lists L_1 and L_2.

A steel grade of the plate can be specified for a pinned beam-to-column joint with a plate by clicking the respective button.

It should be noted that the computer-aided calculation of a pinned beam-to-column joint with a plate assumes that the cross-sectional dimensions of the plate are taken as close as possible to the cross-sectional dimensions of the beam (in particular, the height of the plate on the beam web). In the case when the user has specified the height of the plate on the plate as less than the effective height of the beam web, the stress concentrations should be taken into account in the check of this joint. Since there are no standard methods for performing such a calculation, this option is not implemented in the application. Thus, if the user specifies the height of the plate not corresponding to the effective height of the beam web, an error message will be generated.

To perform a check of the load-bearing capacity of the specified structural design of the beam-to-column joint, you have to specify all design parameters of the joint. The parameters include the sizes and thickness of structural members of the joint, diameters of bolts, sizes which determine the mutual arrangement of members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table of the **Beam 1** or/and **Beam 2** tab. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the beam-to-column joint design is performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 8.2). Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the joint (column and beams), structural elements of the joint (end-plates, plates, angle cleats etc.), and their connections (welded and bolted) at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the beam-to-column joint design of the MS stage will be generated.

A complete list of checks and values of the respective utilization factors of restrictions can be obtained by clicking the **Factors** button. The list of the load-bearing capacity checks of the members and connections of the rigid and pinned beam-to-column joints performed by the application is given in Tables 9.5.5-1, 9.5.5-2.

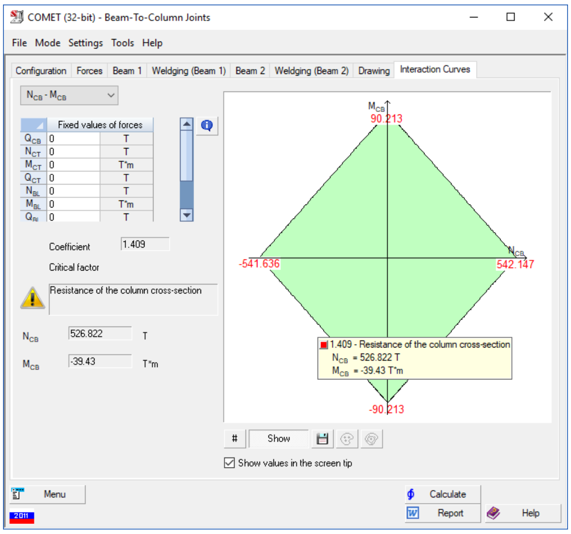


Figure 9.5.5-9. *The* **Interaction Curves** *tab of the* **Beam-To-Column Joints** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the beam-to-column joint under various pairs of internal forces which can arise in the members adjacent to the joint (in the column and in the girders) are plotted in the **Interaction Curves** tab (Fig. 9.5.5-9). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the beam-to-column joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

Once you switch to the **Drawing** tab, the application performs a check and design of the joint, similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

The functionality of the **Report** button and of the controls in the **Drawing** tab is similar to that described for the **Rigid Column Bases** mode (see Sec. 9.5.1).

Table 9.5.5-1. A list of the load-bearing capacity checks of the members and connections of the rigid beam-to-column joints

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the bearing stiffener from the condition of bearing of the end surface	Fig. 9.5.5-1, *b*	Sec. 5.38	Sec. 16.12	Sec. 15.12	Sec. 1.8	Sec. 12	Sec. 7.38
Resistance of the bearing stiffener from the condition of general stability	Fig. 9.5.5-1, *b*	Sec. 5.3, (7)	Sec.8.1.3, (7)	Sec. 7.1.3 (7)	Sec. 1.4.1.3, (1.4.3), Sec. 1.5.5.13	Sec. 9.5.13, Sec. 8.1.3, (8.3)	Sec. 9.13
Resistance of the bearing stiffener from the condition of local stability of the flange overhang	Fig. 9.5.5-1, *b*	Sec. 7.24	Sec. 8.5.18, (97)	Sec. 9.5.14 (82)	Sec. 1.4.3.7, (1.4.27), Sec. 1.5.5.13	Sec. 9.5.13, Sec. 8.1.3, (8.3)	Sec. 9.13
Bending resistance of the end-plate, allowing for the weakening by holes	Fig. 9.5.5-1, *b-g*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.2.1, (41)	Sec. 1.5.2.1, (1.5.1)	Sec. 9.2.1, (9.1)	Sec. 7.12, (24)

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the welded connection between the beam and the column flange	Fig. 9.5.5-1, *a*	Sec. 11.2*, (120), (121), Sec. 11.3*, (122), (123)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, (129), (130), Sec. 13.3, (131), (132)
Resistance of the welded connection between the beam and the bearing stiffener (end-plate)	Fig. 9.5.5-1, *b-g*	Sec. 11.2*, (120), (121), Sec. 11.3*, (122), (123)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, (129), (130), Sec. 13.3, (131), (132)
Resistance of the bolted connection between the bearing stiffener (end-plate) and the column flange	Fig. 9.5.5-1, *b-g*	Sec. 11.13*, (131)*, (132)*	Sec. 15.3.3, (170), Sec.15.3.4, (171)	Sec. 14.3.3, (191), Sec. 14.3.4, (192)	Sec. 1.12.3.3, (1.12.17), Sec. 1.12.3.4, (1.12.18)	Sec. 16.3.3, (16.17), Sec. 16.3.4, (16.18)	Sec. 13.13, (140), (141)
Resistance of the welded connection between the angle cleat and the column flange	Fig. 9.5.5-1, *b*	Sec. 11.2*, (120), (121)	Sec. 15.1.16, (155), (156)	Sec.14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3),	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the column web under normal stresses	Fig. 9.5.5-1, *a-g*	Sec. 5.25*, (50)	Sec. 10.1.1, (91)	Sec. 9.1.1, (106)	Sec. 1.6.1.1, (1.6.2)	Sec. 10.1.1, (10.2)	Sec. 7.25, (48)
Resistance of the column web under shear stresses	Fig. 9.5.5-1, *a-g*	Sec. 5.12, (29)	Sec. 9.2.1, (36)	Sec. 8.2.1, (42)	Sec. 1.5.2.1, (1.5.2)	Sec. 9.2.1, (9.2)	Sec. 7.12, (25)
Resistance of the column web under local stresses	Fig. 9.5.5-1, *a-g*	Sec. 5.13, (31)	Sec. 9.2.2, (40), (41)	Sec. 8.2.1, (46), (47)	Sec. 1.5.2.2, (1.5.6), (1.5.7)	Sec. 9.2.2, (9.6), (9.7)	Sec. 7.13, (27)
Resistance of the column web under reduced stresses	Fig. 9.5.5-1, *a-g*	Sec. 5.14*, (33)	Sec. 9.2.1, (38)	Sec. 8.2.1, (44)	Sec. 1.5.2.1, (1.5.4)	Sec. 9.2.1, (9.4)	Sec. 7.14, (29)
Local stability of the column web	Fig. 9.5.5-1, *a-g*	Sec. 7.4, (74), 7.6*, (79), Sec. 7.2* (72), (73)	Sec. 9.5.2, (71), (72), Sec. 9.5.3, (73)	Sec. 9.4.2, (125), (126), Sec. 9.4.3, (131)	Sec. 1.6.4.2, (1.6.20), (1.6.21), (1.6.22)	Sec. 10.4.2, (10.20), (10.21), (10.22)	Sec. 9.5, (81), 9.7, (86), Sec. 9.2, 9.3 (79), (80)

Notes: see Table 9.5.1-1.

Table 9.5.5-2. A list of the load-bearing capacity checks of the members and connections of the pinned beam-to-column joints

Check	Type of joint	SNiP II-23-81*	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Resistance of the angle cleat plate from the condition of bearing of the end surface	Fig. 9.5.5-2, *a*, *b*	Sec. 5.38	Sec. 16.12	Sec. 15.12	Sec. 1.8	Sec. 12	Sec. 7.38
Bending resistance of the angle cleat plate	Fig. 9.5.5-2, *a*, *b*	Sec. 5.12, (28)	Sec. 9.2.1, (35)	Sec. 8.2.1, (41)	Sec. 1.5.2.1, (1.5.2)	Sec. 9.2.1, (9.2)	Sec. 7.12, (24)
Shear resistance of the plate, allowing for the weakening by holes	Fig. 9.5.5-2, *c*	Sec. 5.12, (29)	Sec. 9.2.1, (36), (39)	Sec. 8.2.1, (42), (45)	Sec. 1.5.2.1, (1.5.2), (1.5.5)	Sec. 9.2.1, (9.2), (9.5)	Sec. 7.12, (25)
Resistance of the bolted connection between the beam and the column flange through a plate	Fig. 9.5.5-2, *c*	Sec. 11.13*, (131)*, (132)*	Sec. 15.3.3, (170), Sec. 15.3.4, (171)	Sec. 14.3.3, (191), Sec. 14.3.4, (192)	Sec. 1.12.3.3, (1.12.17), Sec. 1.12.3.4, (1.12.18)	Sec. 16.3.3, (16.17), Sec. 16.3.4, (16.18)	Sec. 13.13, (140), (141)
Resistance of the welded connection between the angle cleat and the column flange	Fig. 9.5.5-2, *a*, *b*	Sec. 11.2*, (120), (121)	Sec. 15.1.16, (155), (156)	Sec. 14.1.16, (176), (177)	Sec. 1.12.1.16, (1.12.2), (1.12.3)	Sec. 16.1.16, (16.2), (16.3)	Sec. 13.2, (129), (130)
Resistance of the welded connection between the plate and the column flange	Fig. 9.5.5-2, *c*	Sec. 11.2*, (120), (121), Sec. 11.3*, (122), (123)	Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183)	Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9)	Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9)	Sec. 13.2, (129), (130), Sec. 13.3, (131), (132)
Notes: see Table 9.5.1-1.							

9.5.6 Standard Joints

The **Standard Joints** mode enables to design standard beam-to-beam joints in one level where either bolts or an angle cleat is used. The application implements two types of beam-to-beam joints in one level; they use plates bolted to the stiffener of the main beam and to the web of the secondary beam on one or two sides. This mode also implements six types of beam-to-beam joints with an angle cleat welded to the upper flange of the main beam and the end of the web of the secondary beam. The following types of cross-section are used for the connected beams in the **Standard Joints** mode: a regular I-beam according to GOST 26020-83, a wide flange I-beam according to GOST 26020-83 and a channel with sloped inner flange surfaces according to GOST 8240-89.

The designs of the beam-to-beam joints are taken according to DC27-4-2-90 “The album of standard joints of the metal structures of the bearing frame of public buildings" (part I; second edition), developed by the administration of the "MOSPROJECT-2" of the Glavmosarchitecture.

The **Standard Joints** dialog box contains two tabs: **Structure** (Fig. 9.5.6-1) and **Drawing** (Fig. 9.5.6-2).

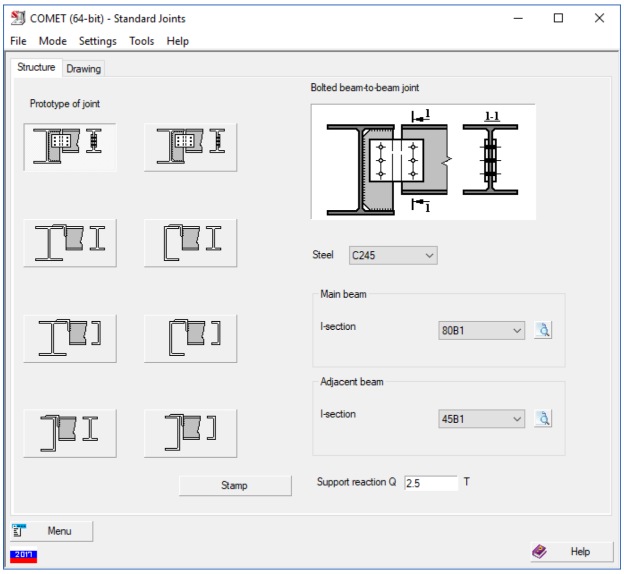


Figure 9.5.6-1. *The* **Structure** *tab of the* **Standard Joints** *dialog box*

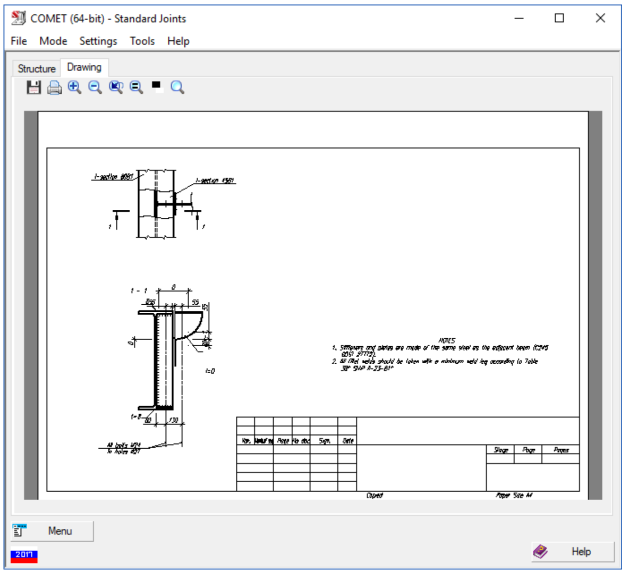


Figure 9.5.6-2. *The* **Drawing** *tab of the* **Standard Joints** *dialog box*

The **Structure** tab contains a group of buttons **Prototype of joint**. It is used to select a structural design for the beam-to-beam joint. This tab also contains a window that displays the structural design of the joint. Use the **Steel** drop-down list to specify a steel grade for the connected members.

The **Main beam** group and the **Adjacent beam** group are used to specify cross-sections for the main beam and for the adjacent beam by selecting the profile numbers from the respective drop-down lists. The specified cross-sections of the connected beams can be checked in the **Preview** windows, which can be invoked by clicking the **Preview** buttons () in the respective groups (**Main beam** or **Adjacent beam**).

The **Support reaction** field is used to enter a numerical value of the support reaction which is transferred from the adjacent beam to the main one. The default units of measurement for the support reaction are tonnes.

Clicking the **Stamp** button opens a dialog box which enables to fill in the stamp of the drawing, which will be generated automatically once the structural design of the beam-to-beam joint is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

Once you switch to the **Drawing** tab (Fig. 9.5.6-2), the application performs a check of the load-bearing capacity of the beam-to-beam joint, and if the result is positive, a drawing of the joint design is generated in the respective information window.

9.6 Design Codes the Requirements of Which are Implemented in COMET

Mode	References to sections of standards or codes					
	SNiP II-23-81	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
Steel	Table 50*	Sec. 6.1,6.2, Annex C, Table C1, C2, C5-C8	Sec. 5.1,5.2, Annex C, Table C1, C2, C5-C8			Table E.3 Annex E
	Table 51*					Table E.4 Annex E
	Table 51, *b*					
	Project SNiP 53-01-96					
Assortment of Rolled Profiles	GOST 26020-83; GOST 8239-89	GOST 26020; GOST 8239	GOST 26020; GOST 8239			GOST 26020-83; GOST 8239-89
	GOST 8240-89	GOST 8240-89	GOST 8240-89			GOST 8240-89
	GOST 8509-93; GOST 8510-86*	GOST 8509-93; GOST 8510-86*	GOST 8509-93; GOST 8510-86*			GOST 8509-93; GOST 8510-86*
Assortment of Sheet and Plate Steel						
Materials for Welding						Table D.5 Annex D
Bolts	Table 57*	Sec. 6.5, Annex. D, Table D3, D6, D6, D8, D9	Sec. 5.5, Annex D, Table D3, D6, D6, D8, D9			Table E.11 Annex E
	GOST 1759.4-87					Table E.12 Annex E
Service Factor	Table 6*	Sec. 5.3.2	Sec. 4.3.2			Table F.1 Annex F
High-strength Bolts						Table E.11 Annex E
Anchor Bolts						Table E.14 Annex E
Standards for Joints						
Setting out Lines						
Concrete Class	Table 12, 13 SNiP 2.03.01-84*					
Concrete Grade	Table 11, 13 SNiP II-21-75					

Mode	References to sections of standards or codes					
	SNiP II-23-81	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Rigid Column Bases	Sec. 5.12, (28), (29) Sec. 5.14*, (33), Sec. 11.2*, (120), (121), Sec. 11.4, (33), Sec. 11.5, (120)-(123), (126), Sec. 11.7*, (129), Sec. 11.8, (130)	Sec. 9.2.1, (35), (36), (38), Sec. 15.2.9, (167), Sec. 15.1.10, (168), Sec. 15.1.15, (38), Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162)	Sec. 8.2.1, (41), (42), (44), Sec. 8.6.2, (101)-(104), Sec. 14.1.15, (44), Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183), Sec. 14.2.9, (186), (188)	Sec. 1.5.2.1, (1.5.1), (1.5.2), (1.5.4), Sec. 1.7.2, (1.7.1), Annex N, (N.1), (N.2), Table N.2, Sec. 1.12.1.15, (1.5.4), Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9), Sec. 1.12.2.9, (1.12.14), Sec. 1.12.2.10, (1.12.15)	Sec. 9.2.1, (9.1), (9.2), (9.4), Sec. 11.2, (11.1), Annex M, (M.1), (M.2), Table M.2, Sec. 16.1.15, (9.4), Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9), Sec. 16.2.9, (16.12-16.14), Sec. 16.2.10, (16.15)	Sec. 7.12, (24), (25) Sec. 7.14, (29), Sec. 13.2, (129), (130), Sec. 13.4, Sec. 13.5, (129)-(132), (135), Sec. 13.7, (138), Sec. 13.8, (139)
Nominally Pinned Column Bases	Sec. 5.12, (28), Sec. 5.14*, (33), Sec. 11.2*, (120)-(121), Sec. 11.4, (33), Sec. 11.5, (120)-(123), (126), Sec. 11.7*, (127)-(128), Sec. 11.8, (130)	Sec. 9.2.1, (35), (38), Sec. 15.1.15, (38), Sec. 15.1.16, (155), (156) Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162), Sec. 15.2.9, (167), Sec. 15.2.10, (168)	Sec. 8.2.1, (44) Sec. 8.6.2, (101)-(104) Sec. 14.1.15, (44), Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183), Sec. 14.2.9, (186), (187), Sec. 14.2.10, (189)	Sec. 1.5.2.1, (1.5.4), Sec. 1.7.2, (1.7.1), Annex N, (N.1), (N.2), Table N.2, Sec. 1.12.1.15, (1.5.4), Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9), Sec. 1.12.2.9, (1.12.12), (1.12.13), Sec. 1.12.2.10, (1.12.15)	Sec. 9.2.1, (9.4), Sec. 11.2, (11.1), Annex M, (M.1), (M.2), Table M.2, Sec. 16.1.15, (9.4), Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9), Sec. 16.2.9, (16.12), (16.13), Sec. 16.2.10, (16.15)	Sec. 7.12, (24), Sec. 7.14, (29), Sec. 13.2, (129)-(130), Sec. 13.4, Sec. 13.5, (129)-(132), (135), Sec. 13.7, (136)-(137), Sec. 13.8, (139)

Mode	References to sections of standards or codes					
	SNiP II-23-81	**SP 53-102-2004**	**SP 16.13330**	**DBN B.2.6-163:2010**	**DBN B.2.6-198:2014**	**ShNK 2.03.05-13**
Beam Splices	Sec. 5.1, (5), Sec. 5.14*, (33), Sec. 5.18*, (41), Sec. 5.25*, (50), Sec. 11.2*, (120)-(123), Sec. 11.7*, (127), (128)	Sec. 8.11, (5), Sec. 9.2.1 (35), (36), (38), (39), Sec. 10.1.1, (91), Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162), Sec. 15.2.9, (165), (166)	Sec. 7.1.1, (5), Sec. 8.2.1, (41), (42), (44), (45), Sec. 9.1.1, (106), Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183), Sec. 14.2.9, (186), (187)	Sec. 1.4.1.1, (1.4.1), Sec. 1.5.2.1, (1.5.1), (1.5.2), (1.5.4), (1.5.5), Sec. 1.6.1.1, (1.6.2), Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9) Sec. 1.12.2.9, (1.12.12), (1.12.13)	Sec. 8.1.1, (8.1), Sec. 9.2.1, (9.1), (9.2), (9.4), (9.5), Sec. 10.1.1, (10.2), Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9) Sec. 16.2.9, (16.12), (16.13)	Sec. 7.1, (1), Sec. 7.14, (29), Sec. 7.18, (39), Sec. 7.25, (48), Sec. 13.2, (129)-(132), Sec. 13.7, (136), (137)

Mode	References to sections of standards or codes					
	SNiP II-23-81	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Truss Panel Points	Sec. 5.1, (5), Sec. 5.3, (7), Sec. 5.12, (28), Sec. 7.23*, Table 29*, Sec. 11.2, (120), (121), Sec. 11.3*, (122), (123), Sec. 11.5, (126), Sec. 11.7*, (127), (128), Sec. 11.8, (130)	Sec. 8.1.1, (5), Sec. 8.3.7, (31), Sec. 9.2.1, (35), Sec. 9.5.13, Sec. 9.6.2 (86)-(88), Sec. 15.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162), Sec. 15.2.9, (165), (166), Sec. 15.2.10, (168) Sec. R.2.2, (R.1), Sec. R.2.3, (R.2), Sec. R.2.4, (R.3), Sec. R.2.5, (R.4), (R.5), Sec. R.2.6, (R.6), (R.7)	Sec. 7.1.1, (5), Sec. 7.3.8, (37), Sec. 8.2.1, (41), Sec. 8.6.2 (101)-(103), Sec. 8.5.17, Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183), Sec. 14.2.9, (186), (187), (188), Sec. 14.2.10, (189), Sec. L.2.2, (L.1), Sec. L.2.3, (L.2), Sec. L.2.4, (L.3), Sec. L.2.5, (L.4), (L.5), Sec. L.2.6, (L.6), (L.7)	Sec. 1.4.1.1, (1.4.1), Sec. 1.5.2.1, (1.5.1), Sec. 1.5.5.13, Sec. 1.7.2, (1.7.1), Annex N, (N.1), (N.2), Table N.2, Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9), Sec. 1.12.2.9, (1.12.12), (1.12.13), (1.12.14), Sec. 1.12.2.10, (1.12.15)	Sec. 8.1.1, (8.1), Sec. 8.1.3, (8.3), Sec.8.3.7, (8.27), Sec. 9.2.1, (9.1), Sec. 9.5.13, Sec. 11.2, (11.1), Annex M, Sec.M.2, (M.1), (M.2), Annex U, Sec.U.1.2, (U.1), Sec.U.1.3, (U.2), Sec.U.1.4, (U.3), Sec.U.1.5, (U.4), (U.5), Sec.U.1.6, (U.6), (U.7), Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9), Sec. 16.2.9, (16.12), (16.13), (16.14), Sec. 16.2.10, (16.15)	Sec. 7.1, (1), Sec. 7.3, (3), Sec. 7.12, (24), Sec. 9.23, Sec. 13.2, (129), (130), Sec. 13.3, (131), (132), Sec. 13.5, (135), Sec. 13.7, (136), (137), Sec. 13.8, (139)

Mode	References to sections of standards or codes					
	SNiP II-23-81	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014	ShNK 2.03.05-13
Beam-To-Column Joints	Sec. 5.3, (7), Sec. 5.12, (28), (29), Sec. 5.13, (31), Sec. 5.14*, (33), Sec. 5.25*, (50), Sec. 7.2* (72), (73), Sec. 7.4, (74), Sec. 7.6*, (79), Sec. 7.24, Sec. 11.2*, (120), (121), Sec. 11.3*, (122), (123), Sec. 11.13*, (131)*, (132)*	Sec. 8.1.3, (7) Sec. 8.5.18, (97), Sec. 9.2.1, (35), (36), (38), (39) Sec. 9.2.2, (41), Sec. 9.5.2, (71), (72), Sec. 9.5.3, (73) Sec. 10.1.1, (91), Sec. 5.1.16, (155), (156), Sec. 15.1.17, (157), (158), Sec. 15.1.19, (161), (162), Sec. 15.3.3, (170), Sec. 15.3.4, (171)	Sec. 7.1.3 (7), Sec. 8.2.1, (41), (42), (44)-(46), Sec. 8.5.17, Sec. 9.1.1, (106), Sec. 9.4.2, (125), (126), Sec. 9.4.3, (131), Sec. 9.5.14 (82), Sec. 14.1.16, (176), (177), Sec. 14.1.17, (178), (179), Sec. 14.1.19, (182), (183) Sec. 14.3.3, (191), Sec. 14.3.4, (192)	Sec. 1.4.1.3, (1.4.3), Sec. 1.4.3.7, (1.4.27), Sec. 1.5.2.1, (1.5.2), (1.5.4), (1.5.5) Sec. 1.5.2.2, (1.5.6), Sec. 1.5.5.13, Sec. 1.6.1.1, (1.6.2), Sec. 1.6.4.2, (1.6.20), (1.6.21), (1.6.22), Sec. 1.12.1.16, (1.12.2), (1.12.3), Sec. 1.12.1.17, (1.12.4), (1.12.5), Sec. 1.12.1.19, (1.12.8), (1.12.9) Sec. 1.12.3.3, (1.12.17), Sec. 1.12.3.4, (1.12.18)	Sec. 8.1.1, (8.1), Sec. 8.1.3, (8.3), Sec. 9.2.1, (9.1), (9.2), (9.4), (9.5) Sec. 9.2.2, (9.6), (9.7), Sec. 9.5.13, Sec. 10.1.1, (10.2), Sec. 10.4.2, (10.20), (10.21), (10.22), Sec.12.1, Sec. 16.1.16, (16.2), (16.3), Sec. 16.1.17, (16.4), (16.5), Sec. 16.1.19, (16.8), (16.9) Sec. 16.3.3, (16.17), Sec. 16.3.4, (16.18)	Sec. 7.3, (3), Sec. 7.12, (24), (25), Sec. 7.13, (27), Sec. 7.14, (29), Sec. 7.25, (48), Sec. 9.2, 9.3 (79), (80), Sec. 9.5, (81), Sec. 9.7, (86), Sec. 9.24, Sec. 13.2*, (129), (130), Sec. 13.3, (131), (132), Sec. 13.13, (140), (141)

9.7 References

[1] V.T. Vasilchenko, A.N. Rutman, E.P. Lukianenko, A manual for steelwork designers, – 2-nd edition — Kiev, "Budivelnik", 1990 – 311 p.

[2] A.V.Perelmuter, L.A.Gildengorn. On classification of steel structures / Structural mechanics and analysis. —1990. —N3. — pp. 67–70.

[3] Guide to design of steel structures (to SNiP II-23-81*) / Kucherenko Centr. Res. Inst. for Building Structures, USSR State Committee for Construction, Moscow, 1989. – 148 p.

[4] Designer's manual. Masonry and reinforced masonry structures. — Moscow, "Stroyizdat" Publ., 1968. — 175 p.

10. COMET Eurocode 3

10.1 General Information

COMET is used to check and design the most common types of joints of steel bar structures used in civil and industrial engineering. This chapter provides a detailed description of the software branch which implements the requirements of EN 1993-1-1:2005, EN1993-1-5:2005, and EN 1993-1-8:2005.

Once a design is selected for the joint, the application enables to determine all its parameters, which must comply with the standard requirements, a number of structural constraints, and the assortment of steel members. Both the standard requirements and structural constraints provided in the codes are obligatory, and their violation is not allowed. However, there are also certain design constraints violating which would cause only a warning, and the application can generate a solution with such violations.

The initial data for computer-aided design of steel structural joints include a configuration or type of the joint, type and sizes of cross-sections of bearing members connected in this joint, and internal forces acting in these members for one or several design combinations of loadings.

The user can either accept the suggested solution or modify it according to his preferences, in order to take into account:

- the technology used to manufacture the structure;
- requirements of unification of the design within the framework of a project or other accepted limits of application (design organization, manufacturing plant etc.);
- the usage of standard designs commonly applied in the enterprise;
- the quality control system, the accepted marking system etc.

Then the software performs a check of the design of the joint and generates a drawing: a sketch of the joint design with all its parameters. In order to be able to modify the design thus generated, or to alter the format of representation of drawings (e.g., dimensioning, legends etc.), the graphical results can be exported as a DXF (AutoCAD) file.

Sec. 9.2 provides some additional comments describing a number of general decision making considerations which are incorporated in the concept of **COMET** development and have to be taken into account by the user.

10.2 Main Window

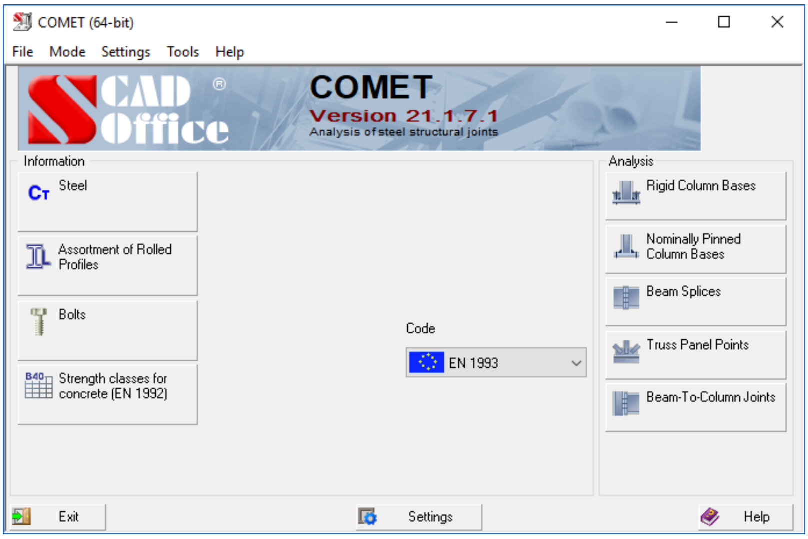


Figure 10.2-1. *The main window*

When the application is started, the first thing that appears on the screen is its main window (Fig. 10.2-1) with a set of buttons for selecting a working mode. These modes can be subdivided into two groups:

- *reference modes* which perform reference and auxiliary operations and are united in the **Information** group;
- *design modes* which enable to perform checks and design of steel structural joints and are united in the **Analysis** group.

A detailed description of each mode is given below. Their brief characteristic is given here.

The reference modes include:

- **Steel —** implements recommendations of EN 1993-1-1:2005 on selection of steel grades. This information mode provides reference data on the characteristic values of the yield strength and the ultimate strength for different steel grades according to EN 10025-2:2004, EN 10025-3:2004, EN 10025-4:2004, EN 10025-5:2004, EN 10025-6:2004, as well as EN 10210-1:2006 and EN 10219-1:2006;
- **Assortment of Rolled Profiles** — enables to browse through assortments of structural steel;
- **Bolts** — enables to browse through an assortment of bolts with reference data on the mechanical properties of bolts, and also on the proof loads, and minimum ultimate tensile loads according to EN ISO 898-1:1999-12;
- **Concrete Classes (EN 1992)** – enables to determine the characteristic cylindrical compressive strength of concrete after 28 days, and the design strength at the ultimate and serviceability limit states depending on the compressive strength class of concrete according to EN 1992-1-1:2004.

The *design modes* include:

- **Rigid Column Bases** — enables to perform a check and design of joints of the column bases which provide a rigid column-to-foundation connection;
- **Nominally Pinned Column Bases** — enables to perform a check and design of nominally pinned joints of column bases;
- **Beam Splices** — enables to perform a check and design of beam splices with plates or end-plates;
- **Truss Panel Points** — enables to perform a check and design of truss joints;
- **Beam-To-Column Joints** — enables to perform a check and design of pinned and rigid joints between beams and columns.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

The main window also contains a number of buttons which are common controls for all working modes. These include the **Exit**, **Settings**, and **Help** buttons. The **Help** and **Exit** buttons perform functions common for a Windows application: providing reference information and finishing the current session, respectively.

The **Settings** button invokes the **Application Settings** dialog box where you can customize the program (see Sec. 2.2).

The **Menu** button switches from any mode to the main window.

10.3 Information Modes

10.3.1 Steel

This mode is used to select a steel grade for the designed structure and its members and contains two tabs: **Structural Steels and Plates** (Fig. 10.3.1-1) and **Pipes** (Fig. 10.3.1-2), which contain a list of steel grades according to EN 10025-2:2004, EN 10025-3:2004, EN 10025-4:2004, EN 10025-5:2004, EN 10025-6:2004, as well as EN 10210-1:2006 and EN 10219-1:2006, and reference data on the characteristic values of the yield strength (f_y) and the ultimate strength (f_u), corresponding to the steel grade selected from the list.

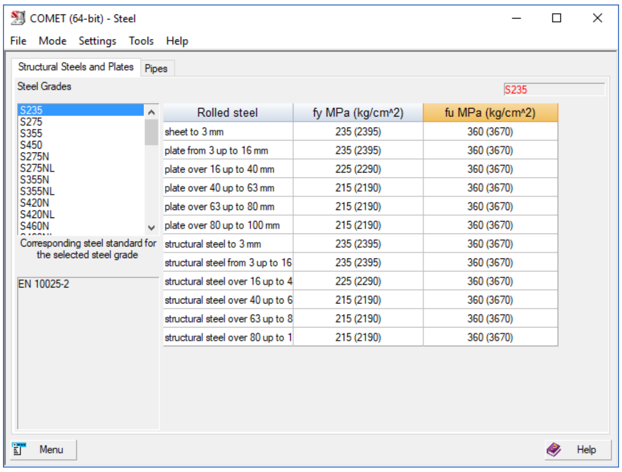


Figure 10.3.1-1. *The* **Structural Steels and Plates** *tab of the* **Steel** *dialog box*

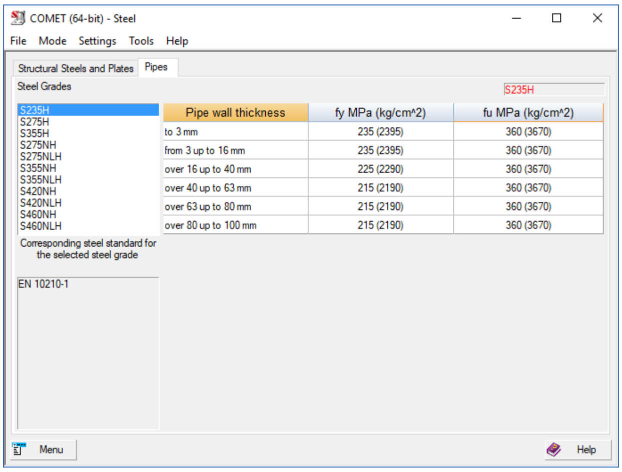


Figure 10.3.1-2. *The* **Pipes** *tab of the* **Steel** *dialog box*

10.3.2 Assortment of Rolled Profiles

This mode enables you to browse through steel profile assortments available in the database of **COMET**.

The dialog box of this mode contains a list of assortments, **Assortment of Rolled Profiles**, represented by a tree-like structure located on the left. When a profile type is selected, the information window displays a table with properties of the respective profiles (Fig. 10.3.2-1). Once you select a particular profile, this window will display the cross-section of this profile with its dimensions (Fig. 10.3.2-2).

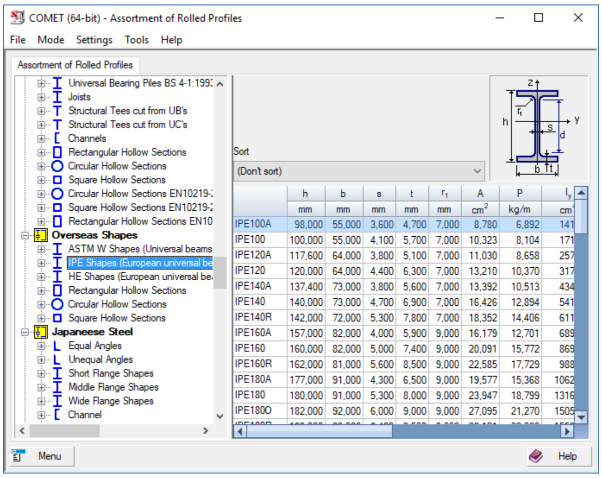


Figure 10.3.2-1. *The* **Assortment of Rolled Profiles** *dialog box*
(a profile type has been selected)

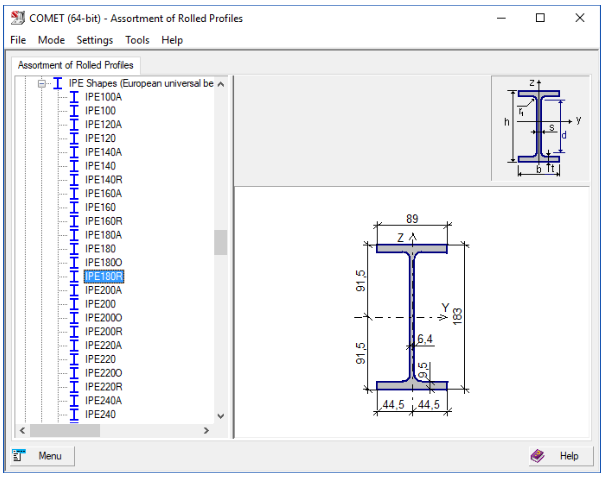


Figure 10.3.2-2. *The* **Assortment of Rolled Profiles** *dialog box*
(a particular profile has been selected)

10.3.3 Bolts

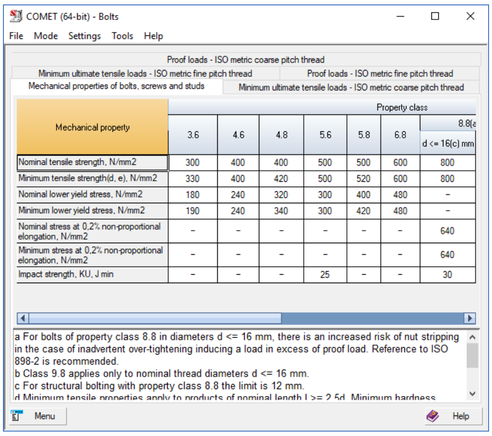


Figure 10.3.3-1. *The* **Bolts** *dialog box*

The **Bolts** mode enables to browse through assortments of bolts with the reference data on the mechanical properties of bolts, and also on the proof loads, and minimum ultimate tensile loads according to EN ISO 898-1:1999-12 (Fig. 10.3.3-1).

10.3.4 Concrete Class (EN 1992)

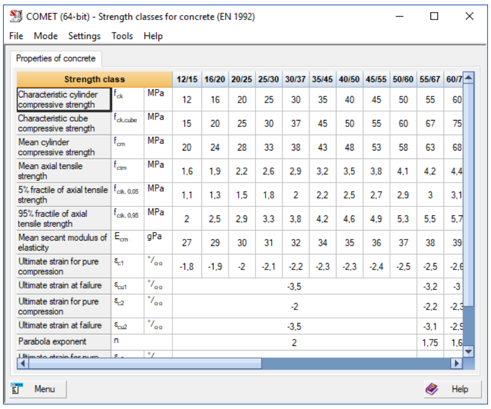


Figure 10.3.4-1. *The* **Concrete Class (EN 1992)** *dialog box*

The **Concrete Class (EN 1992)** mode (Fig. 10.3.4-1) enables to obtain information about the characteristic cylindrical compressive strength of concrete after 28 days, and the design strength at the ultimate and serviceability limit states depending on the compressive strength class of concrete according to EN 1992-1-1:2004.

10.4 Design Modes

10.4.1 Rigid Column Bases

The **Rigid Column Bases** mode enables to design and check the load-bearing capacity of joints of the column bases which provide a rigid column-to-foundation connection. This mode comprises a wide range of designs for this type of joints, such as:

- column bases without wing plates and cantilever stiffeners (Fig. 10.4.1-1);
- column bases with wing plates and cantilever stiffeners (Fig. 10.4.1-2).

This mode performs the following checks in compliance with EN 1993-1-1 and EN 1993-1-8:

- resistance of the structural members of the column base joint (base plate, anchor bolts, anchor plates, wing plates and cantilever stiffeners, foundation concrete in local bearing);
- resistance of welded connections (between the column and the base plate, between the wing plate and column flanges, between the wing plate and the base plate, between the cantilever stiffener and column flanges, between the cantilever stiffener and the wing plate);
- a number of structural and assortment constraints.

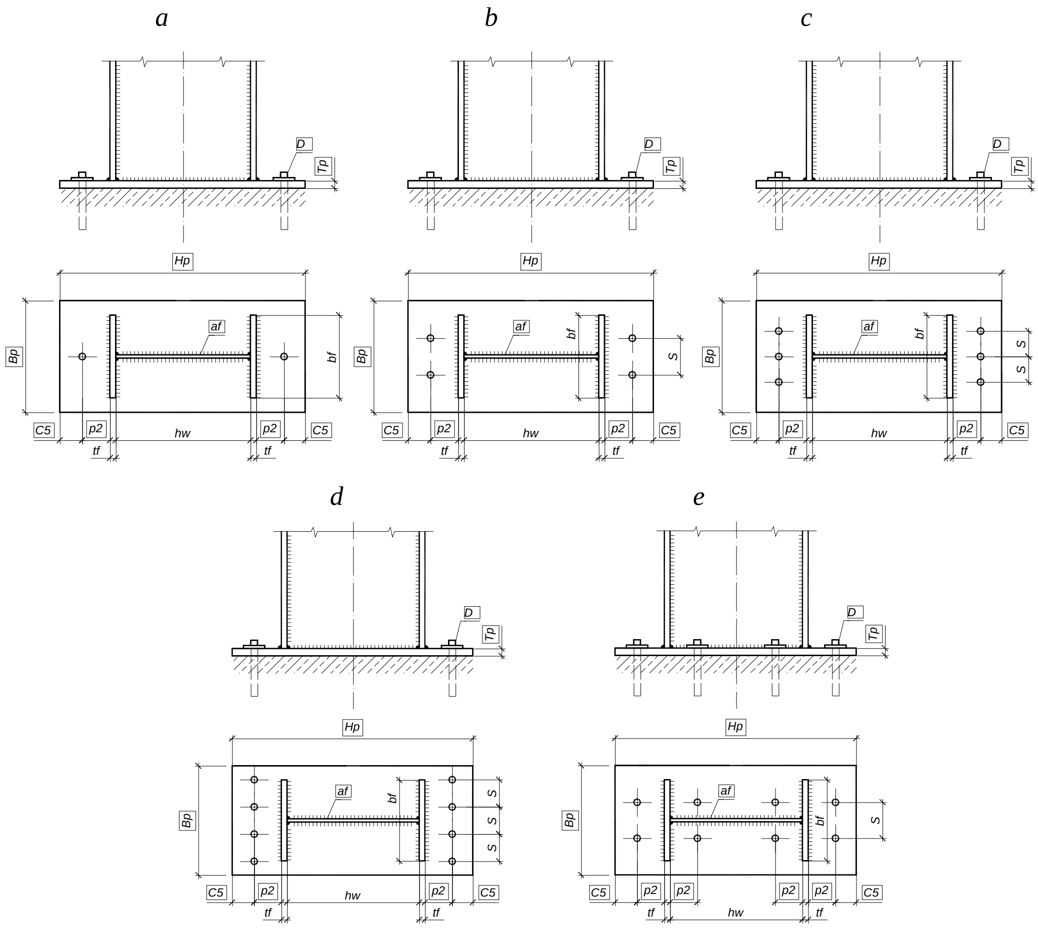


Figure 10.4.1-1. Types of designs for the joint of rigid column bases without wing plates and cantilever stiffeners

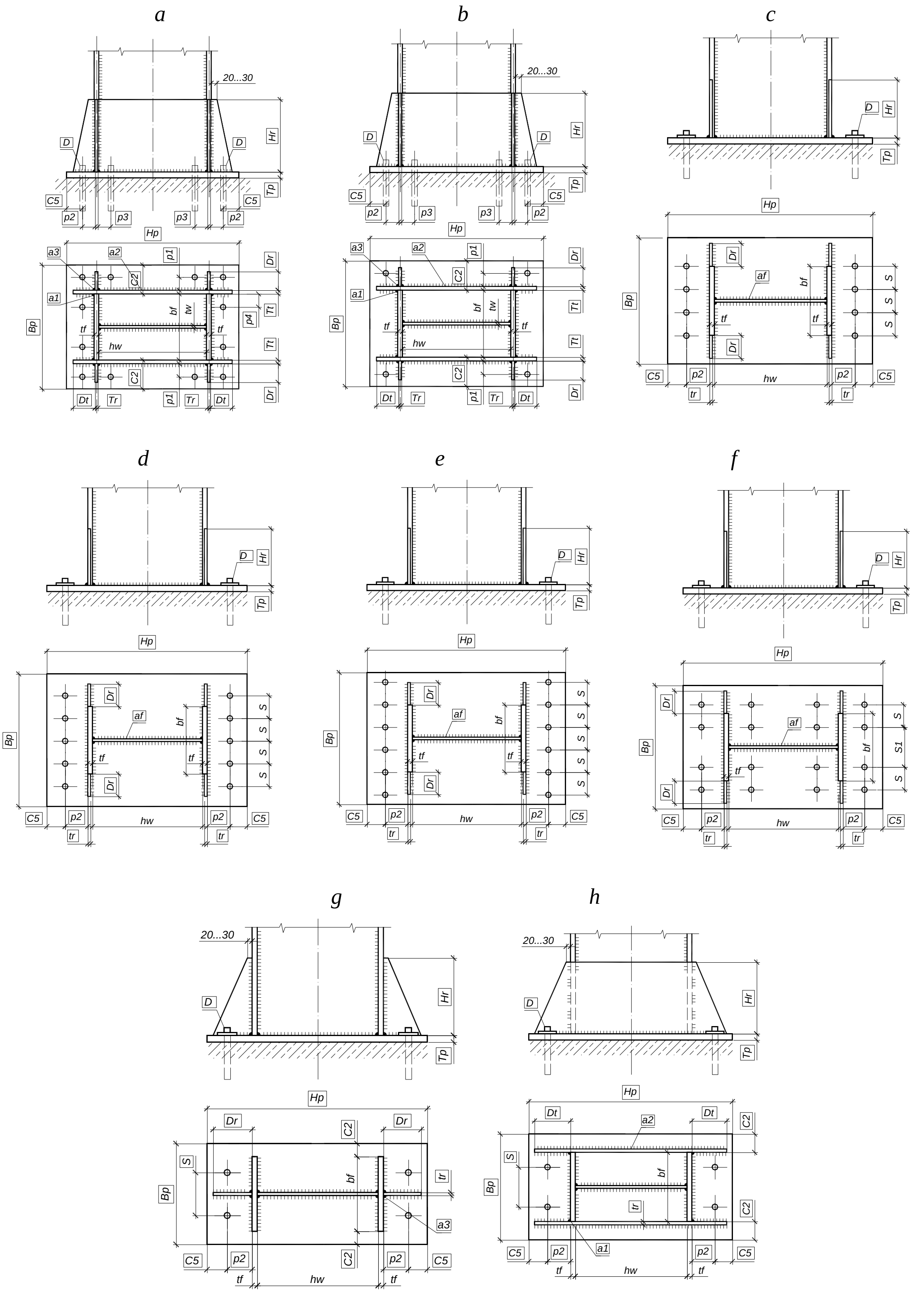


Figure 10.4.1-2. Types of designs for the joints of rigid column bases with wing plates and cantilever stiffeners

The **Rigid Column Bases** dialog box contains the following tabs: **Configuration** (Fig. 10.4.1-3, 10.4.1-4), **Connections** (Fig. 10.4.1-8), **Forces** (Fig. 10.4.1-9), **Structure** (Fig. 10.4.1-10), **Drawing** (Fig. 10.4.1-11) and **Interaction Curves** (Fig. 10.4.1-12).

First you have to select the type of the column section by clicking the respective button: **Rolled I-section** or **Welded I-section** in the **Configuration** tab. The interface of this tab depends on this choice (Fig. 10.4.1-3, Fig. 10.4.1-4).

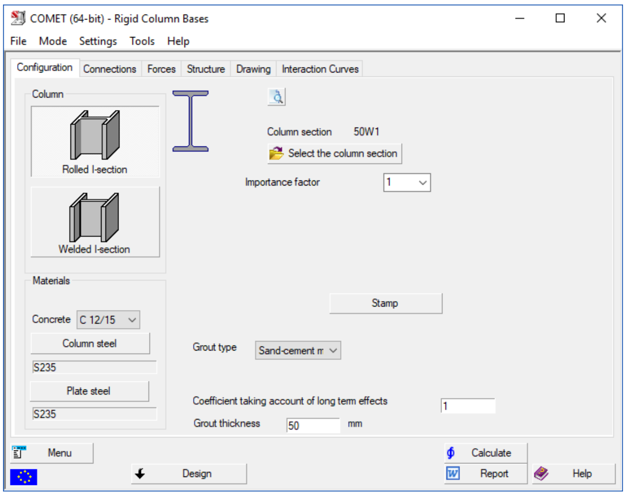


Figure 10.4.1-3. *The* **Configuration** *tab of the* **Rigid Column Bases** *dialog box* (a rolled I-section is selected as the column cross-section type)

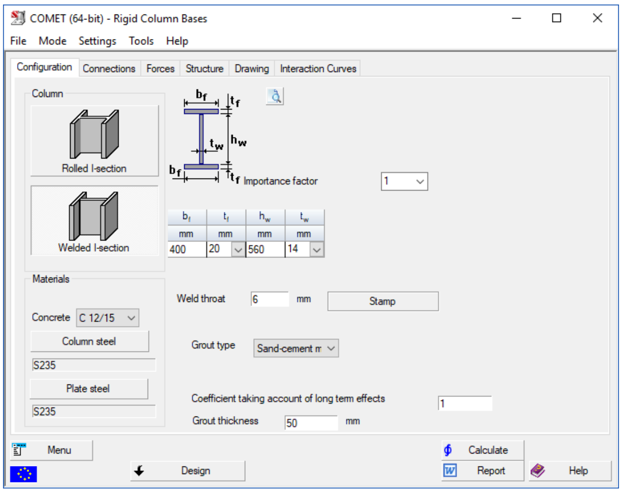


Figure 10.4.1-4. *The* **Configuration** *tab of the* **Rigid Column Bases** *dialog box* (a welded I-section is selected as the column cross-section type)

If a rolled I-section is selected as the column cross-section type, you then have to select an assortment and the profile number in this assortment in the **Select profile** dialog box (Fig. 10.4.1-5), which can be invoked by clicking the **Select the column section** button.

When a welded I-section is selected as the column cross-section type, you have to specify the sizes of the column cross-section: the height, h_w, and the thickness, t_w, of the web; the width, b_f, and the thickness, t_f, of the flanges. The sizes of the column cross-section should be entered in the table in millimeters (Fig. 10.4.1-4). It should be noted that the thickness of the flanges and of the web can be either entered manually or selected from the drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel. The column cross-section can be checked in the **Preview** window (Fig. 10.4.1-6), which can be invoked by clicking the **Preview** button ().

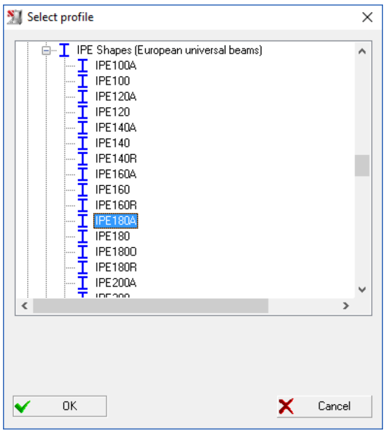


Figure 10.4.1-5. *The* **Select profile** *dialog box*

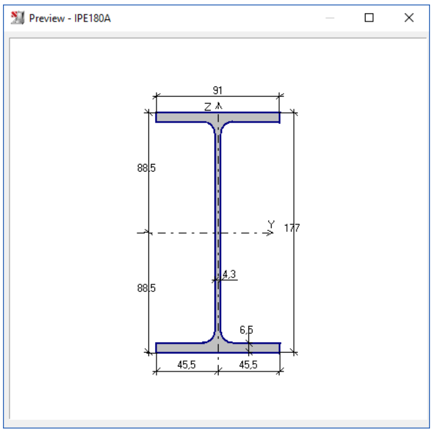


Figure 10.4.1-6. *The* **Preview** *window*

The material of the foundation is selected from the **Concrete** drop-down list which suggests concrete classes for the foundation of the column base according to EN 1992-1-1:2004. Clicking the **Column steel** and **Plate steel** buttons invokes the **Steel** information mode (Fig. 10.4.1-1), where you can select a steel grade for the column and for the support base plate respectively.

The **Configuration** tab also contains the **Grout type** drop-down list and the **Grout thickness** field, which are used to enter the respective information about the grout under the base plate of the column base joint (see Fig. 10.4.1-3, Fig. 10.4.1-4).

This tab also has the **Coefficient taking account of long term effects and of unfavourable effects** where you can specify the value of the coefficient (according to EN 1992-1-1:2004), which is by default taken as one.

The importance factor which will be further multiplied by the design values of all internal forces for all design combinations of loadings acting in the column base section has to be specified in the **Importance factor** drop-down list. If the values of the internal forces in the column base section have been obtained in the results of the analysis accounting for the importance factor (for example, when the design values of the loads were specified already multiplied by this factor), the value equal to one has to be selected in this drop-down list.

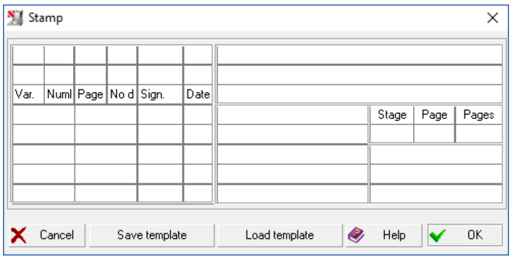


Figure 10.4.1-7. *The* **Stamp** *dialog box*

Clicking the **Stamp** button opens a **Stamp** dialog box, which enables to fill in the stamp of the drawing (Fig. 10.4.1-7), used in the draft of the design of the rigid column base joint. The **Save template** button enables to save the entered data as a template of the stamp for the current session of the application. The saved template can be used both in the current and in other modes of the application by clicking the **Load template** button.

The **Connections** tab (Fig. 10.4.1-8) of the **Rigid Column Bases** dialog box is used to specify information on the peculiarities of welded and bolted connections of the joints of rigid column bases. In particular, the class of bolts and the type of bolt hole are selected from **Class of bolts** and **Hole type** drop-down lists respectively. The type of washer for bolted connections is specified using the respective radio buttons (plain or chamfered). You can also take into account the peculiarities of bolted connections with countersunk bolts by checking the **Countersunk bolts** checkbox and specifying the height of the bolt head in the respective text field. The partial penetration of butt welds can be taken into account for welded connections of the joints of rigid column bases by checking the **Partial penetration butt welds** checkbox and specifying the depth of penetration in the respective text field. The deep penetration of fillet welds can be also taken into account for welded connections of the joints of rigid column bases by checking the **Deep penetration fillet welds** checkbox and specifying the additional throat thickness in the respective text field. Checkboxes of the **Peculiarities** group enable to specify the peculiarities of steels of the joint members, in particular in those cases when the steel is taken according to EN 10025-5 or is exposed to the weather or other corrosive influences. You can also take into account the peculiarities of bolted connections in the following cases: when the threads of bolts do not comply with EN 1090-1:2009, or when the coated fasteners are used in the bolted connection, or when the bolted connections are made in the structures of towers and masts.

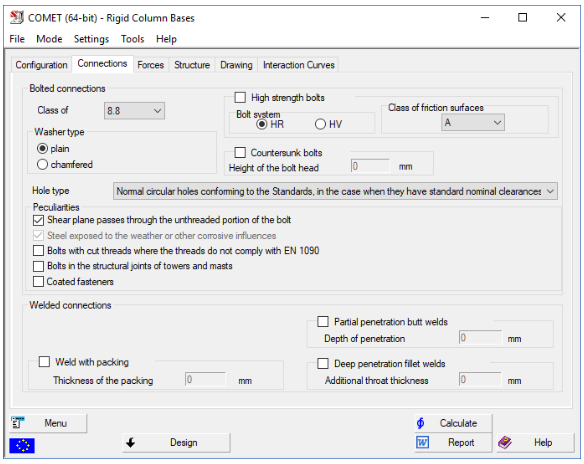


Figure 10.4.1-8. *The* **Connections** *tab of the* **Rigid Column Bases** *dialog box*

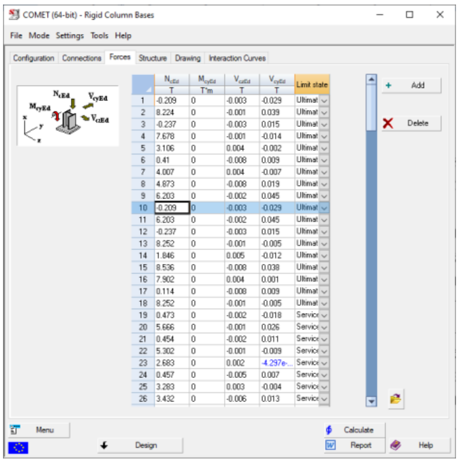


Figure 10.4.1-9. *The* **Forces** *tab of the* **Rigid Column Bases** *dialog box*

The **Forces** tab (Fig. 10.4.1-9) is used to specify the internal forces acting in the joint of the column base: an axial force, N_{cEd}; bending moments in the xOz plane, M_{cyEd}; shear forces with respect to the corresponding principal axes of inertia of the column section, V_{czEd} and V_{cyEd} [8]. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the column base members. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current combination of loads. There can be any number of design combinations of loads. Units of measurement for the internal forces acting in the joint are defined in the **Units of Measurement** tab of the **Application Settings** dialog box (see Sec. 2.2). The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter. The table for specifying the design values of internal forces has the **Limit state** column where you can select the limit state of the each combination of loads (ultimate or serviceability).

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

The **Structure** tab contains a group of buttons to select a design for the joint of the rigid column base (Fig. 10.4.1-10).

To perform a check of the load-bearing capacity of a known structural design of the column base according to EN 1993-1-1 and EN 1993-1-8, you have to specify all parameters of the joint: the sizes and thickness of structural members of the joint, diameters of anchor bolts, sizes which determine the mutual arrangement of structural

[8] To set a proper orientation of the specified internal forces with respect to the principal axes of inertia of the cross-sections that meet in a joint, each bar of the joint is referred to a local coordinate system, *xyz*. The application implements the following orientation of the local coordinate systems of bars: the $x-x$ axis goes from the beginning of a bar (its start node) to its end (its end node), the $y-y$ and $z-z$ axes (the principal central axes of inertia of the bar cross-section) make up a right-hand Cartesian coordinate system together with the $x-x$ axis. The $y-y$ axis is parallel to the XOY plane of the global coordinate system, and the $z-z$ axis goes to the upper half-space.

members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table on the right. The default units of linear measurement are millimeters. The diameter and the number of anchor bolts (for some types of bases) are selected from the special drop-down lists of the **Anchor bolts** group.

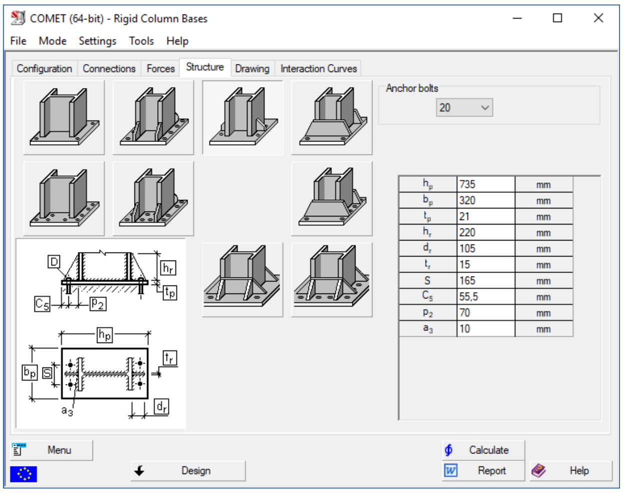


Figure 10.4.1-10. *The* **Structure** *tab of the* **Rigid Column Bases** *dialog box*

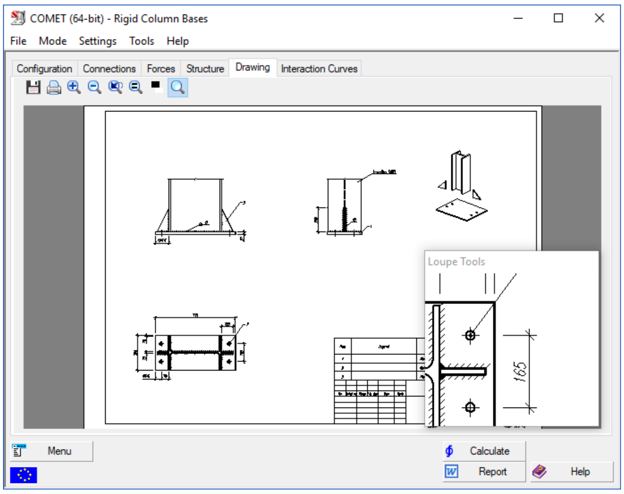


Figure 10.4.1-11. *The* **Drawing** *tab of the* **Rigid Column Bases** *dialog box*

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the column base design was performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 9.2). The diameter of the anchor bolts and the thickness of the base plate, as well as the dimensions of the support base plate were taken as the controlled parameters.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the column base section, structural elements of the joint (base plate, wing plates, cantilever stiffeners etc.) and connections between them (welded connections and anchor bolts) at the specified (or previously selected) values of the parameters of the joint according to EN 1993-1-1 and EN 1993-1-8.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the column base joint design of the MS stage will be generated.

A complete list of the performed checks can be obtained by clicking the **Factors** button. It will be displayed in the special **Factors Diagram** dialog box, where you can browse the values of all utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the rigid column bases performed by the application is given in Table 10.4.1-1.

Clicking the **Report** button generates a report document which contains the initial data and the results of analysis (see Sec. 2.5).

Once you switch to the **Drawing** tab (Fig. 10.4.1-11), the application performs a check and design of the joint, similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

The upper part of the **Drawing** tab contains a toolbar with buttons ( ),which enable to zoom the image in or out, save the drawing as DWG (DXF) for AutoCAD, or print it out.

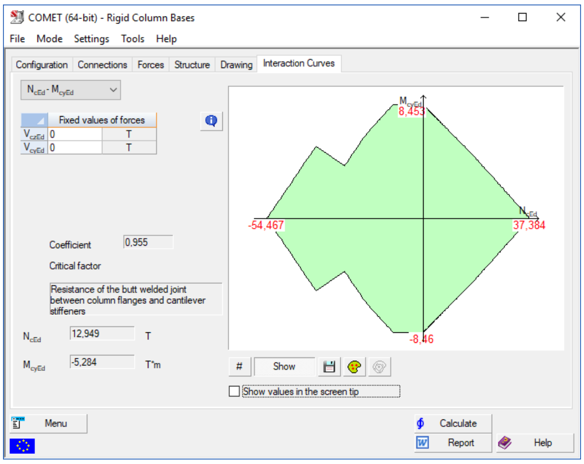


Figure 10.4.1-12. *The* **Interaction Curves** *tab of the* **Rigid Column Bases** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the rigid column base joint under various pairs of internal forces which can arise in the column base section are plotted in the **Interaction Curves** tab (Fig. 10.4.1-12).

Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the rigid base joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of internal forces corresponding to the current position of the pointer in the plot area of the interaction curve.

The maximum value of the utilization factor of restrictions K_{max}, that corresponds to the current values of the internal forces will be displayed in the **Factor** field, and the name of the type of check in which it takes place will be output in the **Critical factor** field. When the pointer is placed outside the area of the load-bearing capacity where $K_{max} > 1$, a warning sign is displayed next to the name of the type of check .

Table 10.4.1-1. A list of the load-bearing capacity checks of the members and connections of the joints of the rigid column bases according to EN 1993-1-1:2005 and EN 1993-1-8:2005

Check	Type of base	EN 1993-1-8	EN 1993-1-1
Resistance of the column base subject to a compression force	Fig. 10.4.1-1; 10.4.1-2	Sec. 6.2.3, 6.2.6.1, 6.2.6.2	
Resistance of the column base subject to a bending moment and an axial force	Fig. 10.4.1-1; 10.4.1-2	Sec. 6.2.6.1, 6.2.6.3, Table 6.7	
Tensile strength of anchor bolts	Fig. 10.4.1-1; 10.4.1-2	Sec. 3.4.2 (2), Table 3.4, 3.6.1 (1), 3.6.1 (3), 6.2.4.12	
Shear strength of anchor bolts	Fig. 10.4.1-1; 10.4.1-2	Sec. 3.4.1 (2), Table 3.4, 3.6.1 (1), 3.6.1 (3)	
Strength of anchor bolts in tension and shear	Fig. 10.4.1-1; 10.4.1-2	Sec. 3.4.1 (2), Table 3.4, 3.6.1 (1), 3.6.1 (3)	
Resistance of the fillet welded connection between the support column section and the support base plate	Fig. 10.4.1-1; 10.4.1-2	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the column flanges	Fig. 10.4.1-2, *a*, *b*, *h*, *i*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the support base plate	Fig. 10.4.1-2, *a*, *b*, *h*, *i*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the cantilever stiffeners	Fig. 10.4.1-2, *a*, *b*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	

Check	Type of base	EN 1993-1-8	EN 1993-1-1
Resistance of the butt welded joint between column flanges and cantilever stiffeners	Fig. 10.4.1-2, *c, d, e, f, g*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the support column section subject to a shear force parallel to the web plane	Fig. 10.4.1-1; 8.5.1-2		Sec. 6.2.6, (6.12)
Resistance of the support column section subject to a shear force perpendicular to the web plane	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.6, (6.12)
Resistance of the support column section subject to an axial tensile force	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.3, (6.5)
Resistance of the support column section subject to an axial compressive force	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.4, (6.9)
Resistance of the support column section subject to a bending moment	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.5, (6.10)
Resistance of the support column section subject to a bending moment and a shear force	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.8, (6.25)
Resistance of the support column section subject to a bending moment and an axial force	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.9.1, (6.26)
Resistance of the support column section subject to a bending moment, an axial force and a shear force	Fig. 10.4.1-1; 10.4.1-2		Sec. 6.2.10
Bending resistance of wing plates	Fig. 10.4.1-2, *a, b, h, i*		Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25)
Bending resistance of cantilever stiffeners	Fig. 10.4.1-2, *a, b, c, d, e, f, g*		Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25)

Notes:

1. The end and edge distances and spacing of fasteners in the bolted connections of joints were taken according to Table 3.3 of EN 1993-1-8:2005.
2. The effective throat thickness of a fillet weld is not less than 3 mm, as defined in Sec. 4.5.2 (2) EN 1993-1-8:2005.
3. The effective length of fillet welds is taken as the overall length of the weld reduced by twice the effective throat thickness in compliance with Sec. 4.5.1 (1) EN 1993-1-8:2005 taking into account the minimum length of a fillet weld defined in Sec. 4.5.1 (2) EN 1993-1-8:2005.

10.4.2 Nominally Pinned Column Bases

The **Nominally Pinned Column Bases** mode enables to design and check the joints of the column bases which provide a pinned column-to-foundation connection. This mode comprises a wide range of designs for this type of joints, such as:

- column bases without wing plates and cantilever stiffeners (Fig. 10.4.2-1);
- column bases with wing plates and cantilever stiffeners (Fig. 10.4.2-2).

This mode performs the following checks in compliance with EN 1993-1-1 and EN 1993-1-8:

- resistance of the structural members of the column base joint (base plate, wing plates and cantilever stiffeners, foundation concrete in local bearing);
- resistance of welded connections (between the column and the base plate, between the wing plate and column flanges, between the wing plate and the base plate, between the cantilever stiffener and column flanges, between the cantilever stiffener and the wing plate);
- a number of structural and assortment constraints.

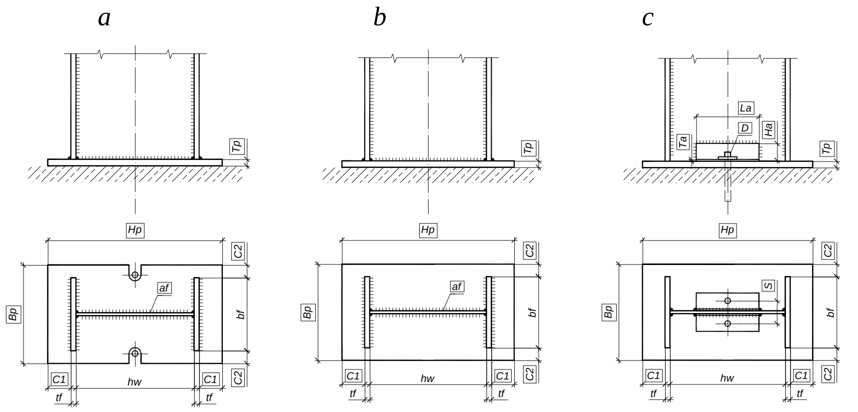


Figure 10.4.2-1. Types of designs for nominally pinned column bases without wing plates and cantilever stiffeners

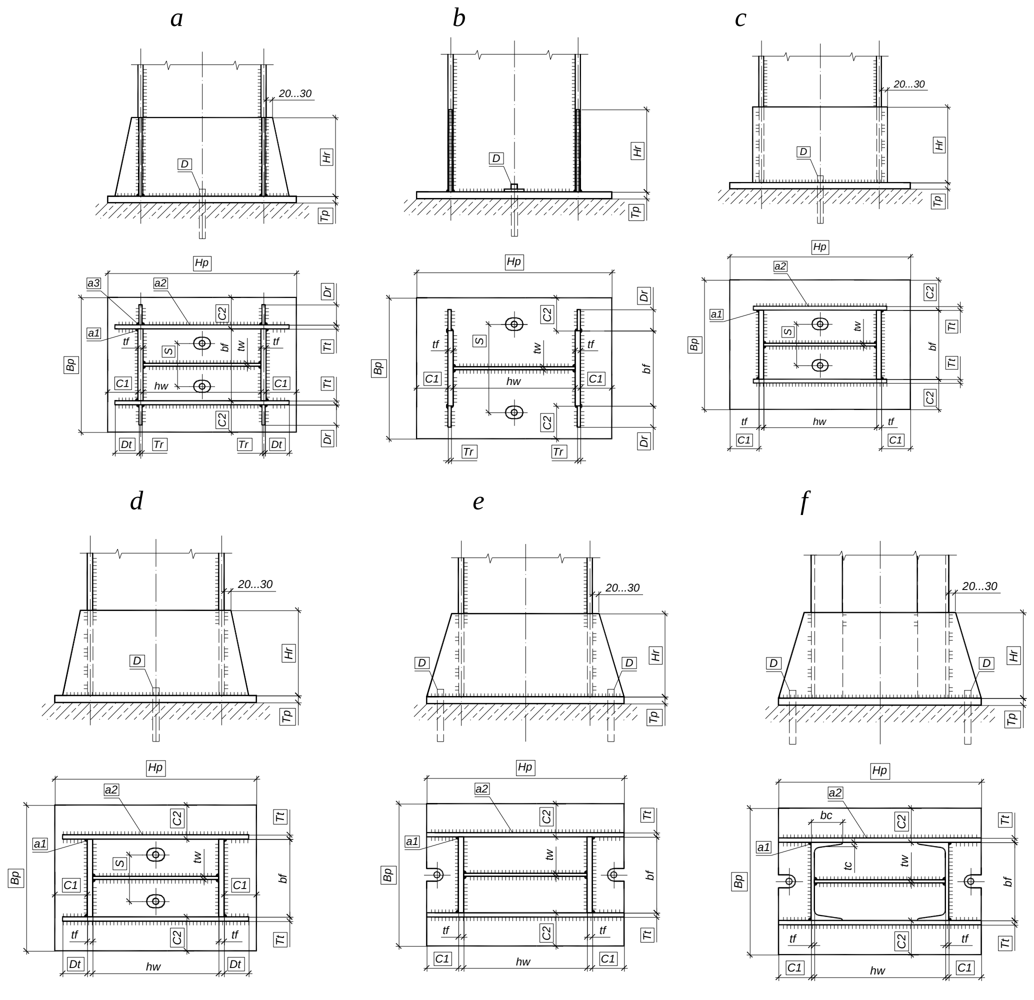


Figure 10.4.2-2. Types of designs for nominally pinned column bases with wing plates and cantilever stiffeners

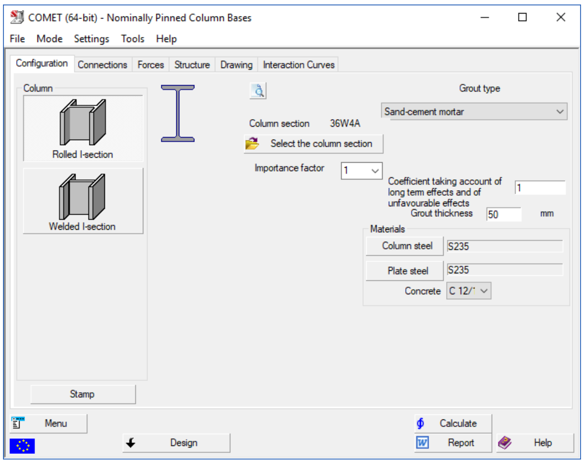


Figure 10.4.2-3. *The* **Configuration** *tab of the* **Nominally Pinned Column Bases** *dialog box*

The **Nominally Pinned Column Bases** dialog box contains five tabs: **Configuration** (Fig. 10.4.2-3), **Connections** (Fig. 10.4.2-6), **Forces** (Fig. 10.4.2-7), **Structure** (Fig. 10.4.2-8), **Drawing** (Fig. 10.4.2-9) and **Interaction Curves** (Fig. 10.4.2-10).

First you have to select the type of the column section by clicking the respective button in the **Configuration** tab (Fig. 10.4.2-3). The mode provides two types of column cross-sections: a rolled I-section and a welded I-section. The sizes of the cross-section of the welded I-section and the profile number of the rolled cross-section are specified in the same way as in the **Rigid Column Bases** mode.

If a rolled I-section is selected as the column cross-section type, you then have to select an assortment and the I-section number in this assortment in the **Select profile** dialog box (Fig. 10.4.2-4), which can be invoked by clicking the **Select the column section** button. The column cross-section can be checked in the **Preview** window (Fig. 10.4.2-5), which can be invoked by clicking the **Preview** button ().

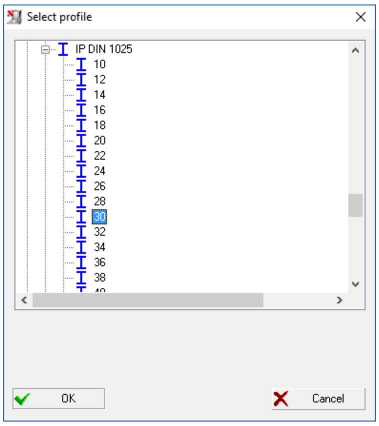


Figure 10.4.2-4. *The* **Select profile** *dialog box*

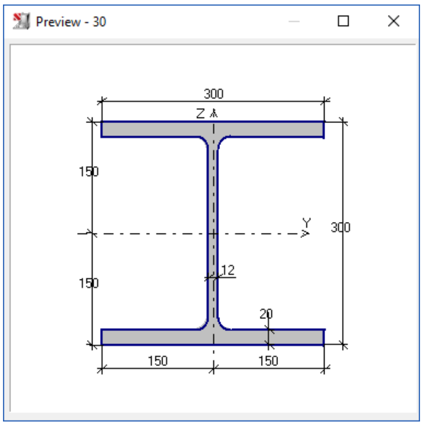


Figure 10.4.2-5. *The* **Preview** *window*

The **Configuration** tab also contains the **Grout type** drop-down list and the **Grout thickness** field, which are used to enter the respective information about the grout under the base plate of the column base joint (see Fig. 10.4.2-3).

Materials used for design and analysis of a nominally pinned column base joint are specified using the **Concrete** drop-down list and the **Column steel** and **Plate steel** buttons. The **Concrete** drop-down list suggests concrete classes for the foundation of the column base according to EN 1992-1-1:2004 Clicking the **Column steel** and **Plate steel** buttons invokes the **Steel** information mode (Fig. 10.3.1-1), where you can select a steel grade for the column and for the support base plate respectively.

The value of the coefficient taking account of long term effects and of unfavourable effects according to EN 1992-1-1:2004, which is by default taken as one, has to be specified in the respective text field (see Fig. 10.4.2-3). There is also the **Importance factor** drop-down list where you have to select or enter the value of the respective factor.

The functionality of the **Stamp** button is the same as that in the **Rigid Column Bases** mode.

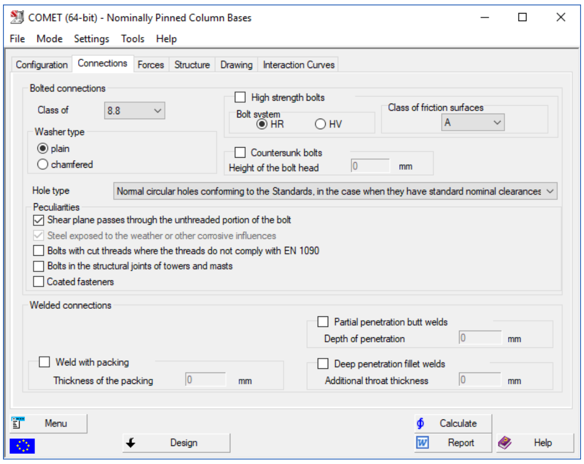


Figure 10.4.2-6. *The* **Connections** tab *of the* **Nominally Pinned Column Bases** *dialog box*

The **Connections** tab (Fig. 10.4.2-6) of the **Nominally Pinned Column Bases** dialog box is used to specify information on the peculiarities of welded and bolted connections of the joints of nominally pinned column bases. In particular, the class of bolts and the type of bolt hole are selected from **Class of bolts** and **Hole type** drop-down lists respectively. The type of washer for bolted connections is specified using the respective radio buttons (plain or chamfered). You can also take into account the peculiarities of bolted connections with countersunk bolts by checking the **Countersunk bolts** checkbox and specifying the height of the bolt head in the respective text field. The partial penetration of butt welds can be taken into account for welded connections of the joints of nominally pinned column bases by checking the **Partial penetration butt welds** checkbox and specifying the depth of penetration in the respective text field.

The deep penetration of fillet welds can be also taken into account for welded connections of the joints of nominally pinned column bases by checking the **Deep penetration fillet welds** checkbox and specifying the additional throat thickness in the respective text field. Checkboxes of the **Peculiarities** group enable to specify the peculiarities of steels of the joint members, in particular in those cases when the steel is taken according to EN 10025-5 or is exposed to the weather or other corrosive influences. You can also take into account the peculiarities of bolted connections in the following cases: when the threads of bolts do not comply with EN 1090-1:2009, or when the coated fasteners are used in the bolted connection, or when the bolted connections are made in the structures of towers and masts.

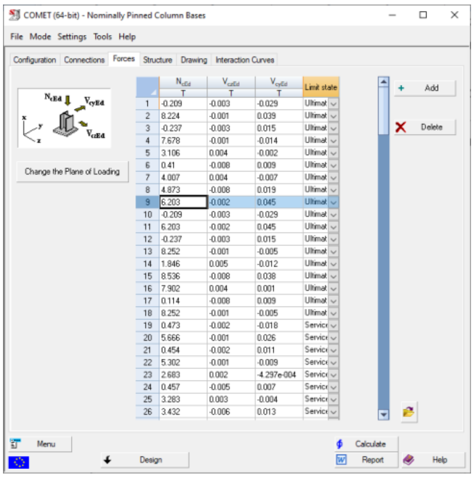


Figure 10.4.2-7. *The* **Forces** tab *of the* **Nominally Pinned Column Bases** *dialog box*

The **Forces** tab (Fig. 10.4.2-7) is used to specify the internal forces acting in the joint of the column base: an axial force, N_{cEd}; shear forces in two planes, V_{czEd} and V_{cyEd}. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of the axial force and the shear forces for the current combination of loads. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the column base members. There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes. Other units of measurement for the internal forces acting in the joint can be defined in the **Units of Measurement** tab of the **Application Settings** dialog box (see Sec. 2.2).

When the analysis and design are performed according to EN 1993-1-1:2005 and EN 1993-1-8:2005, the table in the **Forces** tab (Fig. 10.4.2-7) used for specifying the values of internal forces for one or several design combinations of loads has an additional column, **Limit state**, where you have to select the limit state for the current combination of loads (ultimate or serviceability). To change the load plane, use the respective button. This will transfer the values of M_y and Q_y to the respective columns of the table for M_z and Q_z, and vice versa.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

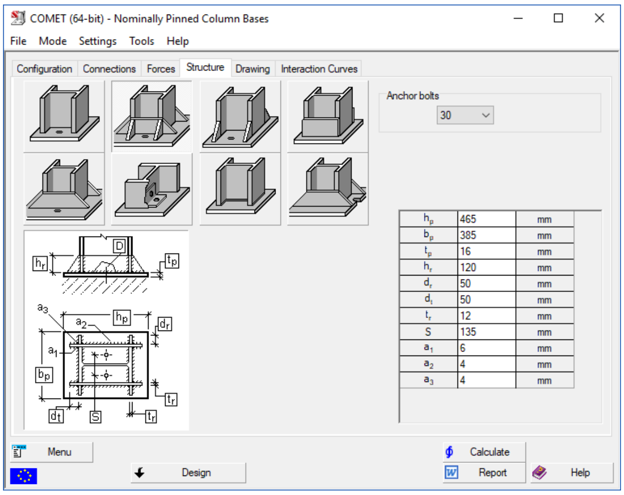


Figure 10.4.2-8. *The* **Structure** *tab of the* **Nominally Pinned Column Bases** *dialog box*

The **Structure** tab contains a group of buttons to select a design for the joint of the nominally pinned column base (Fig. 10.4.2-8).

To perform a check of the load-bearing capacity of the specified structural design of the column base joint, you have to enter the design parameters of the joint in the table in the **Structure** tab. The diameter and the steel grade of anchor bolts are selected from the special drop-down lists of the **Anchor bolts** group. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the nominally pinned column base design was performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards. The diameter of the anchor bolts and the thickness of the base plate, as well as the dimensions of the support base plate were taken as the controlled parameters.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the column base section, structural elements of the joint (base plates, wing plates, cantilever stiffeners etc.) and connections between them (welded connections and anchor bolts) at the specified (or previously selected) values of the parameters of the joint according to EN 1993-1-1:2005 and EN 1993-1-8:2005.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the nominally pinned column base joint design of the MS stage will be generated.

A complete list of the performed checks can be obtained by clicking the **Factors** button. It will be displayed in the special **Factors Diagram** dialog box, where you can browse the values of all utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the nominally pinned column bases performed by the application is given in Table 10.4.2-1.

Once you switch to the **Drawing** tab, the application performs a check of the joint similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the nominally pinned column base joint under various pairs of internal forces which can arise in the column base section are plotted in the **Interaction Curves** tab (Fig. 10.4.2-10).

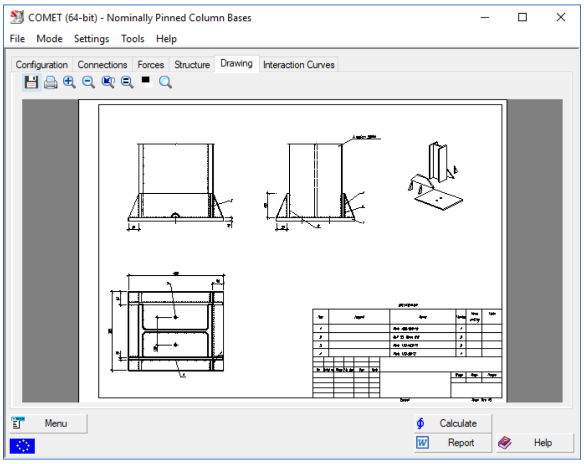


Figure 10.4.2-9. *The* **Drawing** *tab of the* **Nominally Pinned Column Bases** *dialog box*

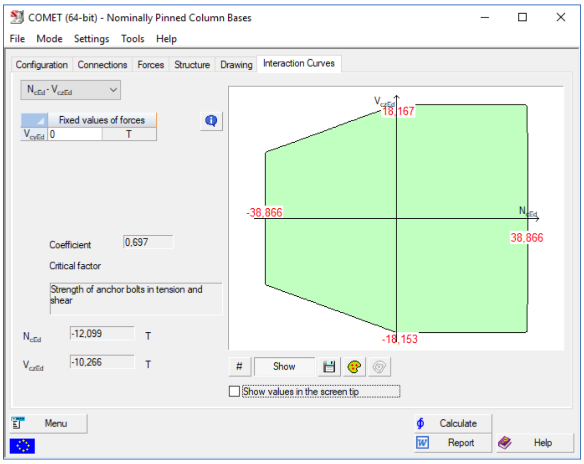


Figure 10.4.2-10. *The* **Interaction Curves** *tab of the* **Nominally Pinned Column Bases** *dialog box*

Click the **Show** button to generate such a curve. A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the nominally pinned base joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

Table 10.4.2-1. A list of the load-bearing capacity checks of the members and connections of the joints of the nominally pinned column bases according to EN 1993-1-1:2005 and EN 1993-1-8:2005

Check	Type of base	EN 1993-1-8	EN 1993-1-1
Resistance of the column base subject to a compression force	Fig. 10.4.2-1; 10.4.2-2	Sec. 6.2.3, (6.4)-(6.6), Sec. 6.2.6.2	
Tensile strength of anchor bolts	Fig. 10.4.2-1; 10.4.2-2	Sec. 3.4.2 (2), Table 3.4, Sec. 3.6.1 (1), Sec. 3.6.1 (3), Sec. 6.2.4.12	
Shear strength of anchor bolts	Fig. 10.4.2-1; 10.4.2-2	Sec. 3.4.1 (2), Table 3.4, Sec. 3.6.1 (1), Sec. 3.6.1 (3)	
Strength of anchor bolts in tension and shear	Fig. 10.4.2-1; 10.4.2-2	Sec. 3.4.1 (2), Table 3.4, Sec. 3.6.1 (1), Sec. 3.6.1 (3)	
Resistance of the fillet welded connection between the support column section and the support base plate	Fig. 10.4.2-1; 10.4.2-2	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the column flanges	Fig. 10.4.2-2, *a*, *b*, *h*, *i*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the support base plate	Fig. 10.4.2-2, *a*, *b*, *h*, *i*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the fillet welded connection between the wing plate and the cantilever stiffeners	Fig. 10.4.2-2, *a*, *b*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	

Check	Type of base	EN 1993-1-8	EN 1993-1-1
Resistance of the butt welded joint between column flanges and cantilever stiffeners	Fig. 10.4.2-2, *c, d, e, f, g*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the support column section subject to a shear force parallel to the web plane	Fig. 10.4.2-1; 10.4.2-2		Sec. 6.2.6, (6.12)
Resistance of the support column section subject to a shear force perpendicular to the web plane	Fig. 10.4.2-1; 10.4.2-2		Sec. 6.2.6, (6.12)
Resistance of the support column section subject to an axial tensile force	Fig. 10.4.2-1; 10.4.2-2		Sec. 6.2.3, (6.5)
Resistance of the support column section subject to an axial compressive force	Fig. 10.4.2-1; 10.4.2-2		Sec. 6.2.4, (6.9)
Bending resistance of wing plates	Fig. 10.4.2-2, *a, b, h, i*		Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25)
Bending resistance of cantilever stiffeners	Fig. 10.4.2-2, *a, b, c, d, e, f, g*		Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25)
Notes: see Table 10.4.1-1.			

10.4.3 Beam Splices

The **Beam Splices** mode enables to design and check the load-bearing capacity of erection joints between I-beams with high strength bolts or ordinary bolts using plates or end-plates.

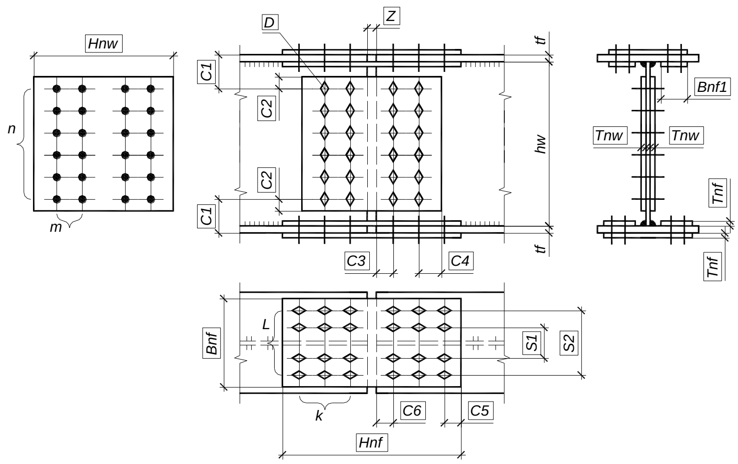


Figure 10.4.3-1. Design of a beam splice with plates

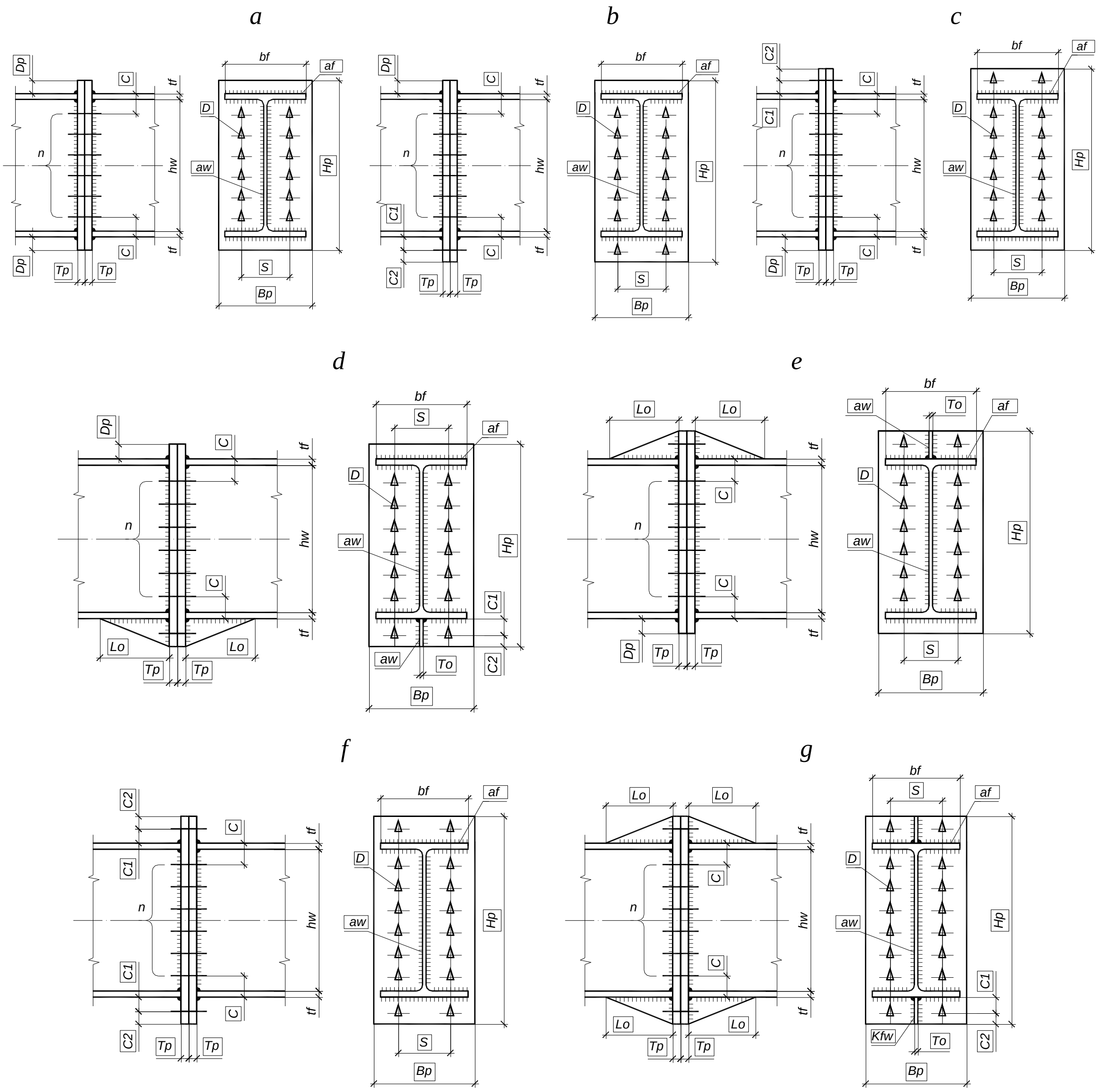


Figure 10.4.3-2. Types of designs for end-plate beam splices

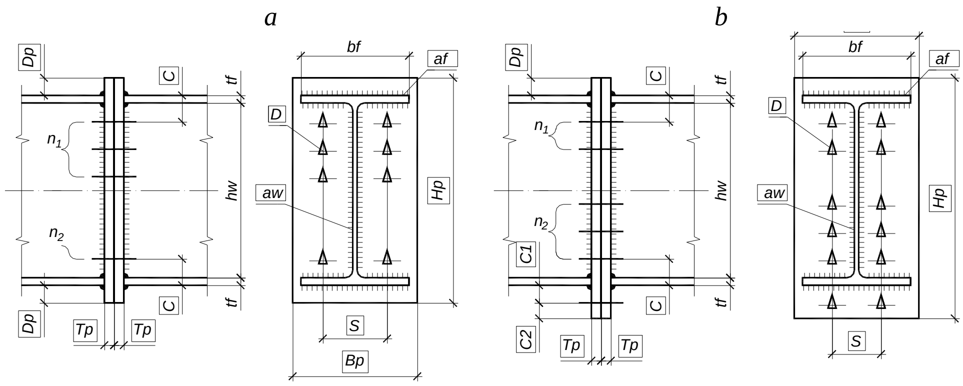


Figure 10.4.3-3. Examples of types of end-plate beam splices with the irregular placement of high strength bolts along the beam web.

This mode comprises a wide range of designs for erection joints between beams:

- beam splices with plates using ordinary bolted connections with bolts of normal and improved strength and connections with high strength bolts (Fig. 10.4.3-1);
- end-plate beam splices of various configurations with high strength bolts (Fig. 10.4.3-2);
- end-plate beam splices of various configurations with the irregular placement of high strength bolts along the beam webs and flanges (Fig. 10.4.3-3).

The erection beam splice with plates has an advantage over the end-plate beam splices in that it does not require the members to be manufactured with a high accuracy. However, this kind of joint usually requires a much greater number of bolts in comparison to end-plate joints, therefore it takes more effort to mount the structures. Moreover, the joint with plates entails the weakening of the cross-sections of connected members by holes, which may in some cases require using more steel for the main structural members.

It should be noted that the computer-aided calculation of beam splices with plates assumes that the cross-sectional dimensions of the plates covering the beam splice are taken as close as possible to the cross-sectional dimensions of the beam. It refers to both the thickness of the plates, and to their linear dimensions (in particular, to the height of the plate on the beam webs and the width of the plates on the beam flanges). The program does not take into account additional stresses from the stress concentrations that can occur if the height of the plate on the beam webs is assumed to be less than the effective height of the beam web.

End-plate joints are usually designed in such a way so that the height of the end-plate corresponds to that of the beam (Fig. 10.4.3-2, *a*). If the bending moment acting in the beam splice can hardly be resisted by bolts placed between the beam flanges, it becomes necessary to use the designs that involve external bolt rows. The latter expand the end-plate dimension downward (Fig. 10.4.3-2, *b, d*) or upward (Fig. 10.4.3-2, *c, e*), depending on the prevailing sign of the bending moment. If there are significant alternating-sign bending moments, you should use end-plate joints with external bolts on both sides of the beam (Fig. 10.4.3-2, *f, g*).

This mode performs the following checks in compliance with EN 1993-1-1:2005 and EN 1993-1-8:2005:

- resistance of plates and end-plates;
- resistance of bolted and welded connections of the joint;
- a number of structural and assortment constraints.

The **Beam Splices** dialog box contains five tabs: **Materials** (Fig. 10.4.3-4, Fig. 10.4.3-5), **Connections** (Fig. 10.4.3-6), **Forces** (Fig. 10.4.3-7), **Structure** (Fig. 10.4.3-8, 10.4.3-9), **Drawing** (Fig. 10.4.3-10) and **Interaction Curves** (Fig. 10.4.3-11).

First you have to specify the materials used in the beam splice. A steel grade for the connected beams can be selected in the **Steel** dialog box (see Sec. 10.3.1), which is invoked by clicking the **Beam steel** button in the **Materials** tab (Fig. 10.4.3-4). This mode also enables to specify different steel grades for end-plates and plates in the considered joints by clicking the **End-plate steel** and **Web plate and flange plate steel** buttons.

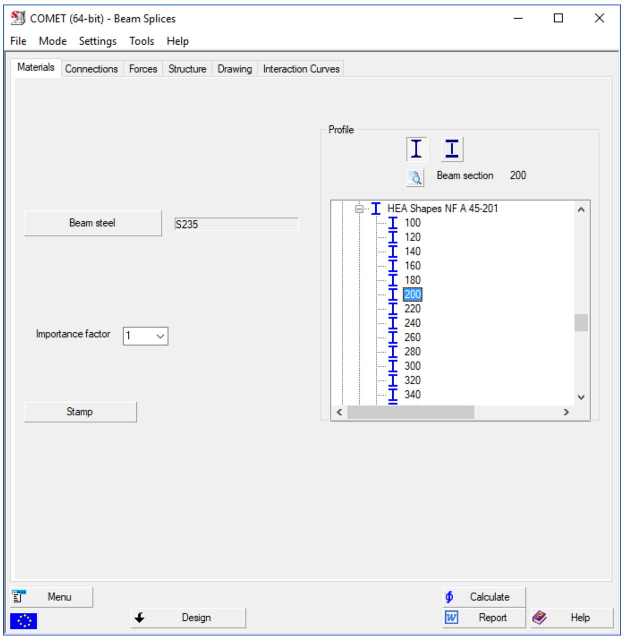


Figure 10.4.3-4. *The* **Materials** *tab of the* **Beam Splices** *dialog box* (a rolled I-section is selected as the beam cross-section type)

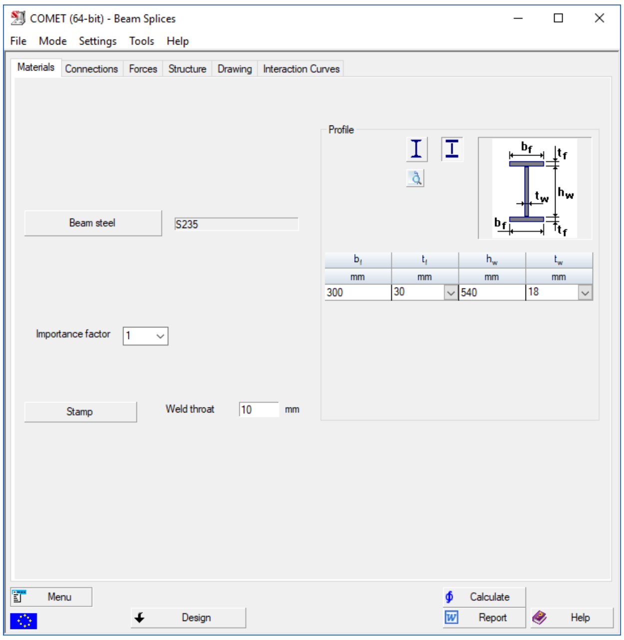


Figure 10.4.3-5. *The* **Materials** *tab of the* **Beam Splices** *dialog box* (a welded I-section is selected as the beam cross-section type)

Controls of the **Profile** group are used to define the type and sizes of the cross-sections of spliced beams. The **Beam Splices** mode provides two types of beam cross-sections: rolled or welded I-section. The section type is selected by clicking the respective button (or). The interface of the right part of the **Materials** tab depends on this choice (Fig. 10.4.3-4, Fig. 10.4.3-5). If a rolled I-section is selected as the beam cross-section type, you then have to select an assortment and the profile number in this assortment from the tree-like list. When a welded I-section is selected as the beam cross-section type, you have to specify the sizes of the beam cross-section: the height, h_w, and the thickness, t_w, of the beam web; the width, b_f, and the thickness, t_f, of the beam flanges. The thickness of the flanges and of the web can be either entered manually or selected from drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel.

The beam cross-section can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ().

Clicking the **Stamp** button opens a dialog box which enables to fill in the stamp of the drawing, which will be generated automatically once the structural design of the beam splice is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

The **Connections** tab (Fig. 10.4.3-6) is used to specify information on the peculiarities of welded and bolted connections of the beam splices. In particular, the class of bolts and the type of bolt hole are selected from **Class of bolts** and **Hole type** drop-down lists respectively. When the **High strength bolts** checkbox is checked, the bolted connections with high strength bolts of classes 8.8 and 10.9 with controlled tightening can be used in the **Beam Splices** mode. The **Class of bolts** drop-down list will contain only two classes of bolts – 8.8 and 10.9.

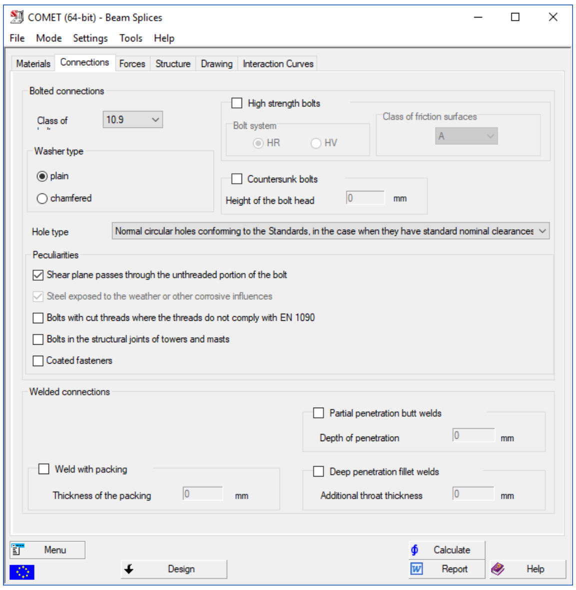


Figure 10.4.3-6. *The* **Connections** *tab of the* **Beam Splices** *dialog box*

The class of friction surfaces has to be selected from the respective drop-down list. The type of washer for bolted connections is specified using the respective radio buttons (plain or chamfered). You can also take into account the peculiarities of bolted connections with countersunk bolts by checking the **Countersunk bolts** checkbox and specifying the height of the bolt head in the respective text field (Fig. 10.4.3-6).

The partial penetration of butt welds can be taken into account for the welded connections of beam splices by checking the **Partial penetration butt welds** checkbox and specifying the depth of penetration in the respective text field. The deep penetration of fillet welds can be also taken into account for the welded connections of beam splices by checking the **Deep penetration fillet welds** checkbox and specifying the additional throat thickness in the respective text field (Fig. 10.4.3-5).

Checkboxes of the **Peculiarities** group enable to specify the peculiarities of steels of the joint members, in particular in those cases when the steel is taken according to EN 10025-5 or is exposed to the weather or other corrosive influences. You can also take into account the peculiarities of bolted connections in the following cases: when the threads of bolts do not comply with EN 1090-1:2009, or when the coated fasteners are used in the bolted connection, or when the bolted connections are made in the structures of towers and masts.

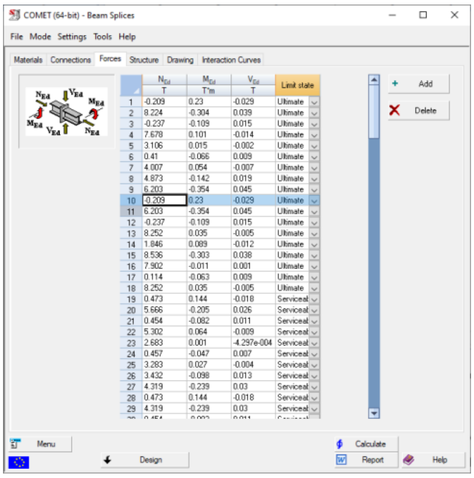


Figure 10.4.3-7. *The* **Forces** *tab of the* **Beam Splices** *dialog box*

The **Forces** tab (Fig. 10.4.3-7) is used to specify the internal forces acting in the beam splice: the axial force, N_{Ed}, the bending moment, M_{Ed}, and its respective shear force, V_{Ed}. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current design combination of loads.

The table used for specifying the values of internal forces for one or several combinations of loads has an additional column, **Limit state**, where you have to select the limit state for the current combination of loads (ultimate or serviceability).

There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter. The positive direction of internal forces is defined by the picture to the left from the table of internal forces.

The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported by clicking the button . It should be noted that when creating an **.rsu2** file in **SCAD**, the table of design combinations should include only those combinations that correspond to the section of the bar element adjacent to the node.

The **Structure** tab contains buttons of the **Type of joint** group, which are used to select a design for the beam splice (Fig. 10.4.3-8, 10.4.3-9). In the case when the user has to perform the analysis or design of end-plate beam splices with the irregular placement of bolts, he can use the **Placement of bolts with respect to the beam** group of interface elements which enables to specify the regular or irregular placement of bolts along the web (Fig. 10.4.3-9), and to select the design of a splice with an additional bolt row along the beam flanges (usually in tension).

To perform a check of the load-bearing capacity of a known (specified) structural design of the beam splice, you have to specify all design parameters of the joint. The parameters include the sizes and thickness of structural members of the joint, diameters of bolts, sizes which determine the mutual arrangement of members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The design parameters of the joint are entered in the table on the right. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

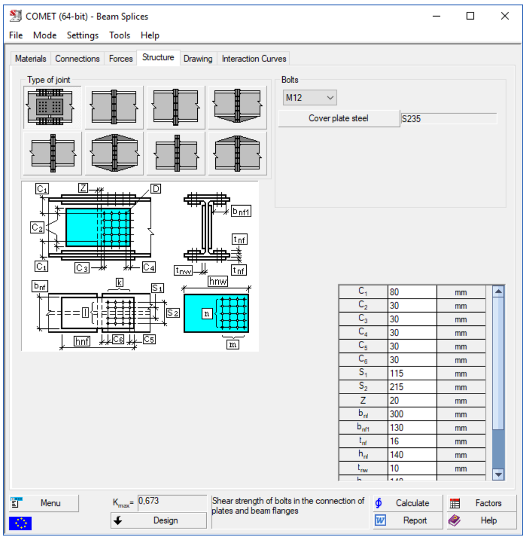


Figure 10.4.3-8. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (a joint with plates is selected)

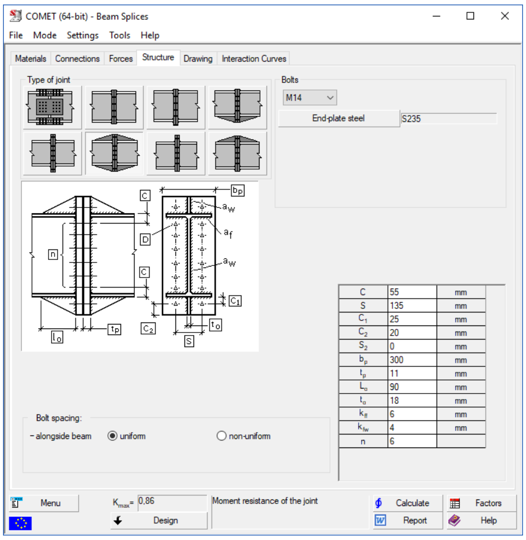


Figure 10.4.3-9. *The* **Structure** *tab of the* **Beam Splices** *dialog box* (an end-plate joint is selected)

The automatic selection of the beam splice design is performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 8.2). The diameter of bolts and the number of bolt rows have been selected as the controlled parameters of the beam splice joint structures for an erection beam splice with plates, and the diameter of bolts, thickness of the end-plate and the number of bolt rows – for the erection end-plate beam splices.

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the considered joint (beam), structural elements of the splice (end-plates, plates, stiffeners etc.), welded and bolted connections in the splice at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the K_{max} field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the beam splice joint design of the MS stage will be generated.

Clicking the **Factors** button invokes the **Factors Diagram** dialog box, where you can browse the values of all other utilization factors of restrictions. The list of the load-bearing capacity checks of the members and connections of the joints of the beam splices performed by the application is given in Table 10.4.3-1.

Table 10.4.3-1. A list of the load-bearing capacity checks of the members and connections of the joints of the beam splices according to EN 1993-1-1:2005 and EN 1993-1-8:2005

Check	Type of splice	EN 1993-1-8	EN 1993-1-1
Tension resistance of plates on beam flanges	Fig. 10.4.3-1		Sec. 6.2.3, (6.5)
Compression resistance of plates on beam flanges	Fig. 10.4.3-1		Sec. 6.2.4, (6.9)
Shear strength of bolts in the connection of plates and beam flanges	Fig. 10.4.3-1	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of high strength bolts in the connection of plates and beam flanges when there are friction forces	Fig. 10.4.3-1	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Bearing strength of bolts in the connection of plates and beam flanges	Fig. 10.4.3-1	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Shear strength of bolts in the connection of plates and beam webs	Fig. 10.4.3-1	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of high strength bolts in the connection of plates and beam webs when there are friction forces	Fig. 10.4.3-1	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Bearing strength of bolts in the connection of plates and beam webs	Fig. 10.4.3-1	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Shear resistance of plates on beam webs	Fig. 10.4.3-1		Sec. 6.2.6, (6.12)
Moment resistance of the joint	Fig. 10.4.3-2	Sec. 6.2.5.1, (6.23)	
Resistance of the welded connection between the beam web and the end-plate	Fig. 10.4.3-2	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the welded connection between the beam flange and the end-plate	Fig. 10.4.3-2	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Shear resistance of the bolted connection between end-plates	Fig. 10.4.3-2	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection with high strength bolts between end-plates when there are friction forces	Fig. 10.4.3-2	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Bearing resistance of the bolted connection between end-plates	Fig. 10.4.3-2	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the beam section weakened by holes subject to a shear force	Fig. 10.4.3-1, 10.4.3-2		Sec. 6.2.6, (6.12)
Resistance of the beam section weakened by holes subject to a bending moment	Fig. 10.4.3-1, 10.4.3-2		Sec. 6.2.5, (6.10)
Resistance of the beam section weakened by holes subject to a bending moment and a shear force	Fig. 10.4.3-1, 10.4.3-2		Sec. 6.2.8, (6.25)
Resistance of the beam section weakened by holes subject to a bending moment and an axial force	Fig. 10.4.3-1, 10.4.3-2		Sec. 6.2.9.1, (6.26)
Resistance of the beam section weakened by holes subject to a bending moment, shear and axial forces	Fig. 10.4.3-1, 10.4.3-2		Sec. 6.2.10

Notes: see Table 10.4.1-1.

Once you switch to the **Drawing** tab (Fig. 10.4.3-10), the application performs a check of the joint similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

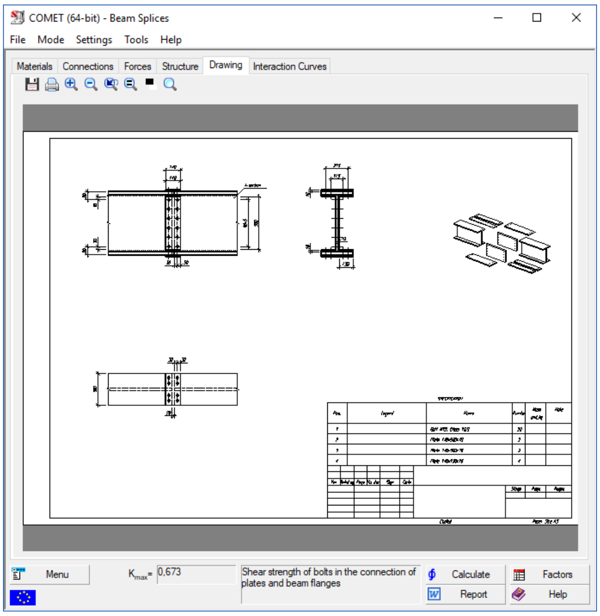


Figure 10.4.3-10. *The* **Drawing** *tab of the* **Beam Splices** *dialog box*

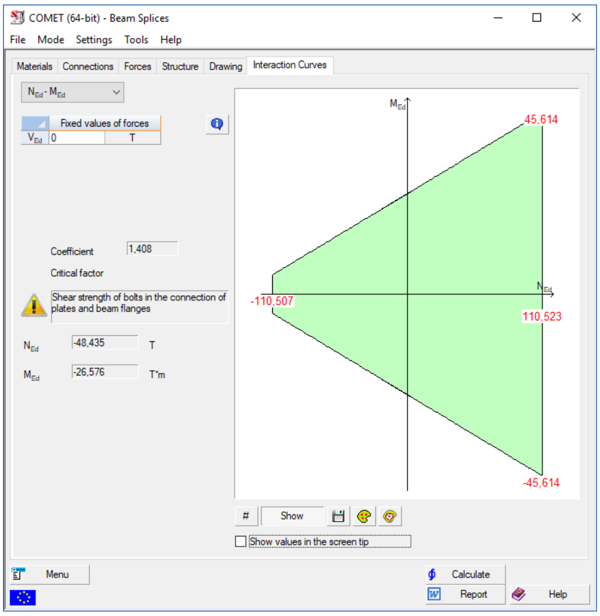


Figure 10.4.3-11. *The* **Interaction Curves** *tab of the* **Beam Splices** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the beam splice joint under various pairs of internal forces which can arise in the beam section adjacent to the splice are plotted in the **Interaction Curves** tab (Fig. 10.4.3-11). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the beam splice joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

10.4.4 Truss Panel Points

The **Truss Panel Points** mode enables to design and check the joints of the truss with the bars made of double angles or rectangular (square) hollow sections. The mode implements a wide range of the types of joints:

- joint between a brace member and the truss chord (regular joint) (Fig. 10.4.4-1, 10.4.4-2);
- joint where the chord changes its cross-section (Fig. 10.4.4-4);
- erection joints (Fig. 10.4.4-5);
- support joints (Fig. 10.4.4-3, 10.4.4-6).

This mode performs the following checks in compliance with EN 1993-1-1:2005 and EN 1993-1-8:2005:

- resistance of structural members (plates, bearing stiffeners (end-plates), gusset plates);
- resistance of welded connections (connections between the brace members and the gusset plates, connections between the chord members and the gusset plates, connection between the plates and the chord in the joints where the chord section changes and in erection joints, connections between the support gusset plate and the bearing stiffener (end plate) in support joints of the truss);

- resistance of bolted connections (connection between the bearing stiffener (end-plate) and the support structure);
- a number of structural and assortment constraints.

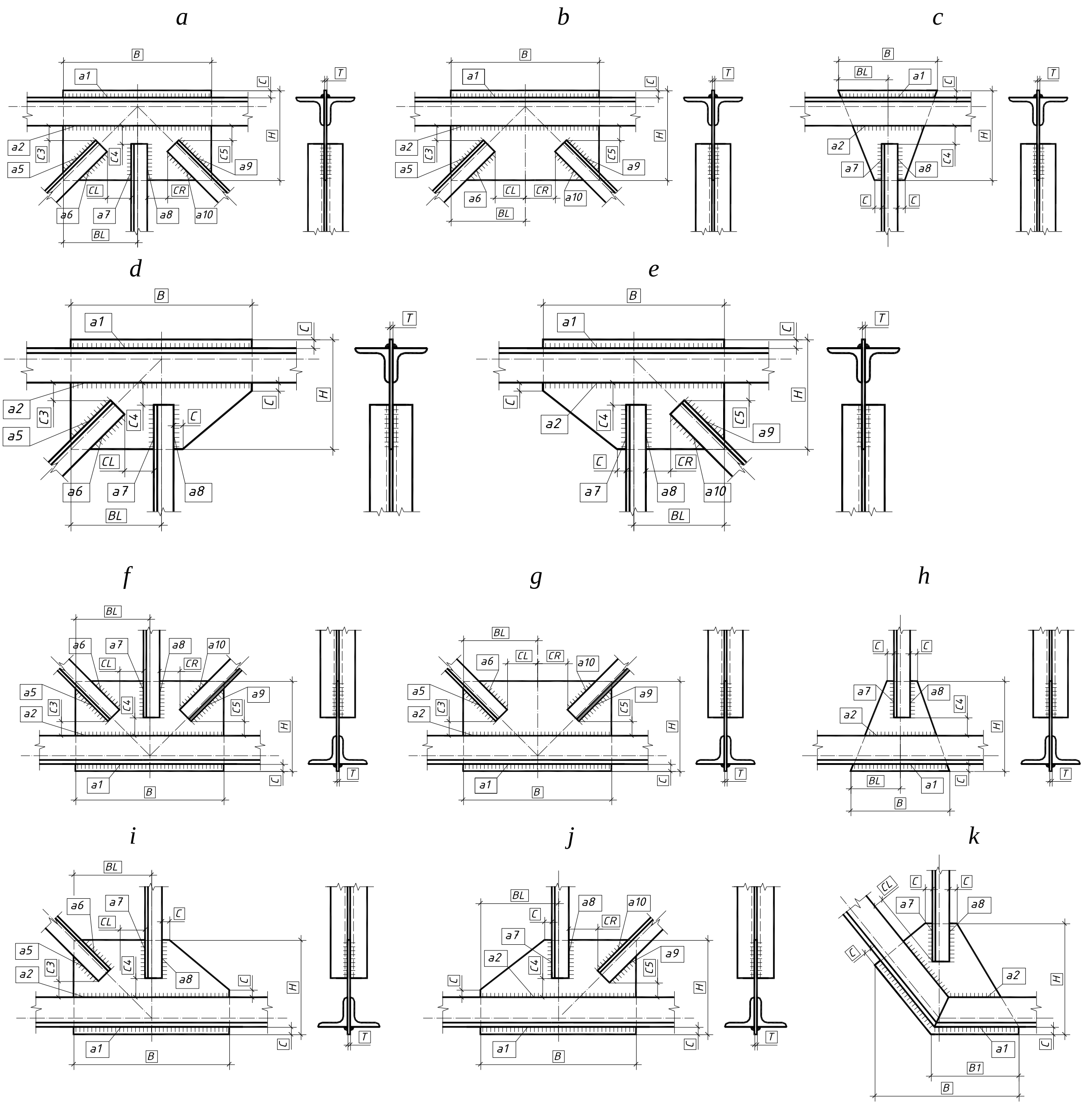


Figure 10.4.4-1. Regular joints in trusses with elements made of double angles: *a* - *e* – top chord joints; *f* - *k* – bottom chord joints

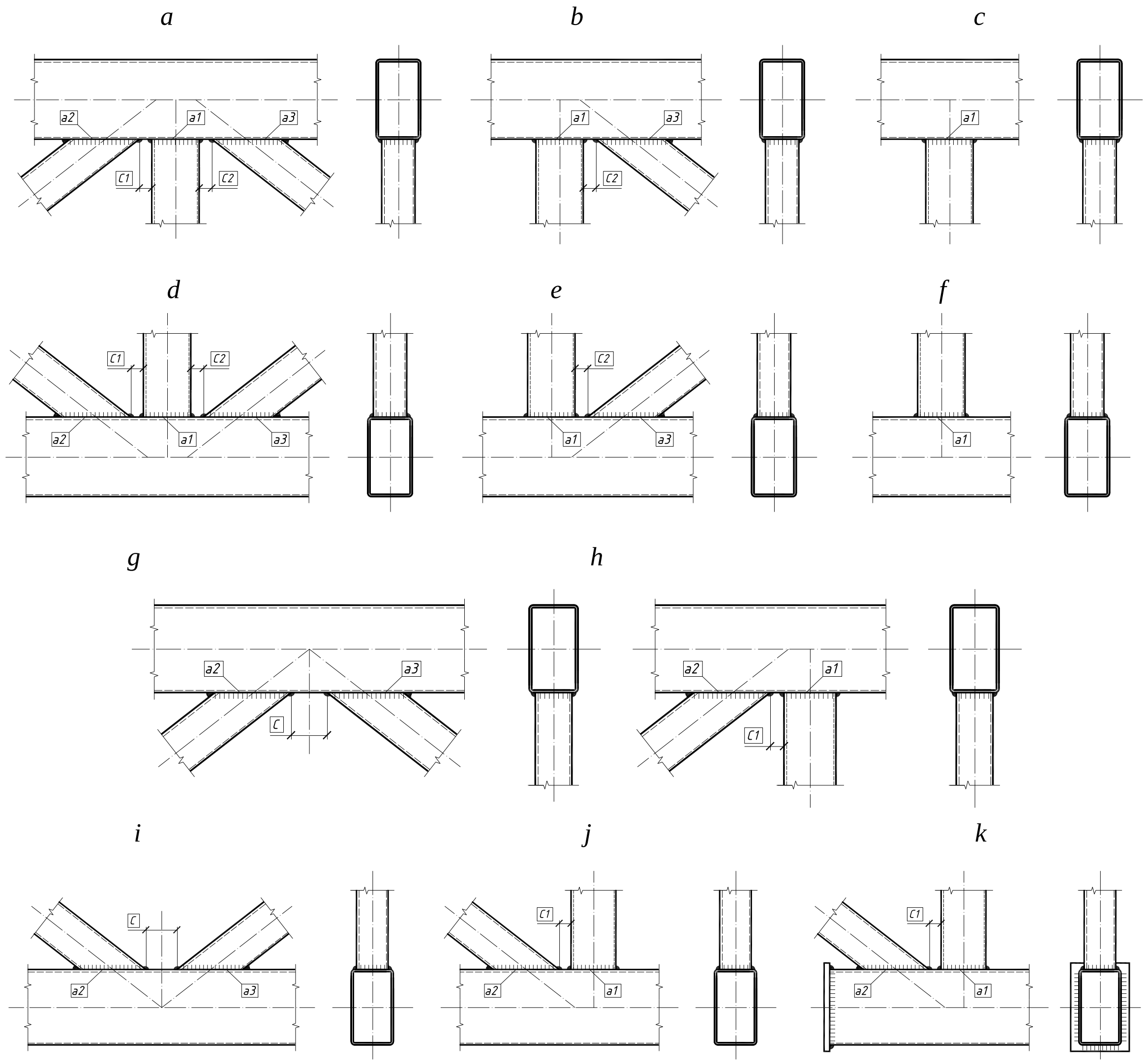


Figure 10.4.4-2. Regular joints in trusses with elements made of rectangular (square) hollow sections: *a*, *b*, *c*, *g*, *h* – top chord joints; *d*, *e*, *f*, *i*, *j*, *k* – bottom chord joints

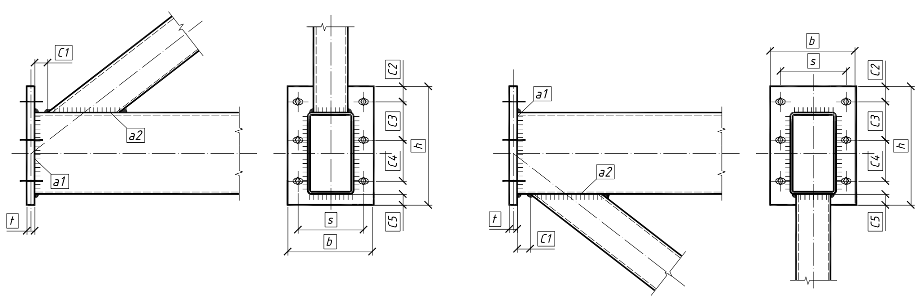


Figure 10.4.4-3. Support joints in trusses with elements made of rectangular (square) hollow sections

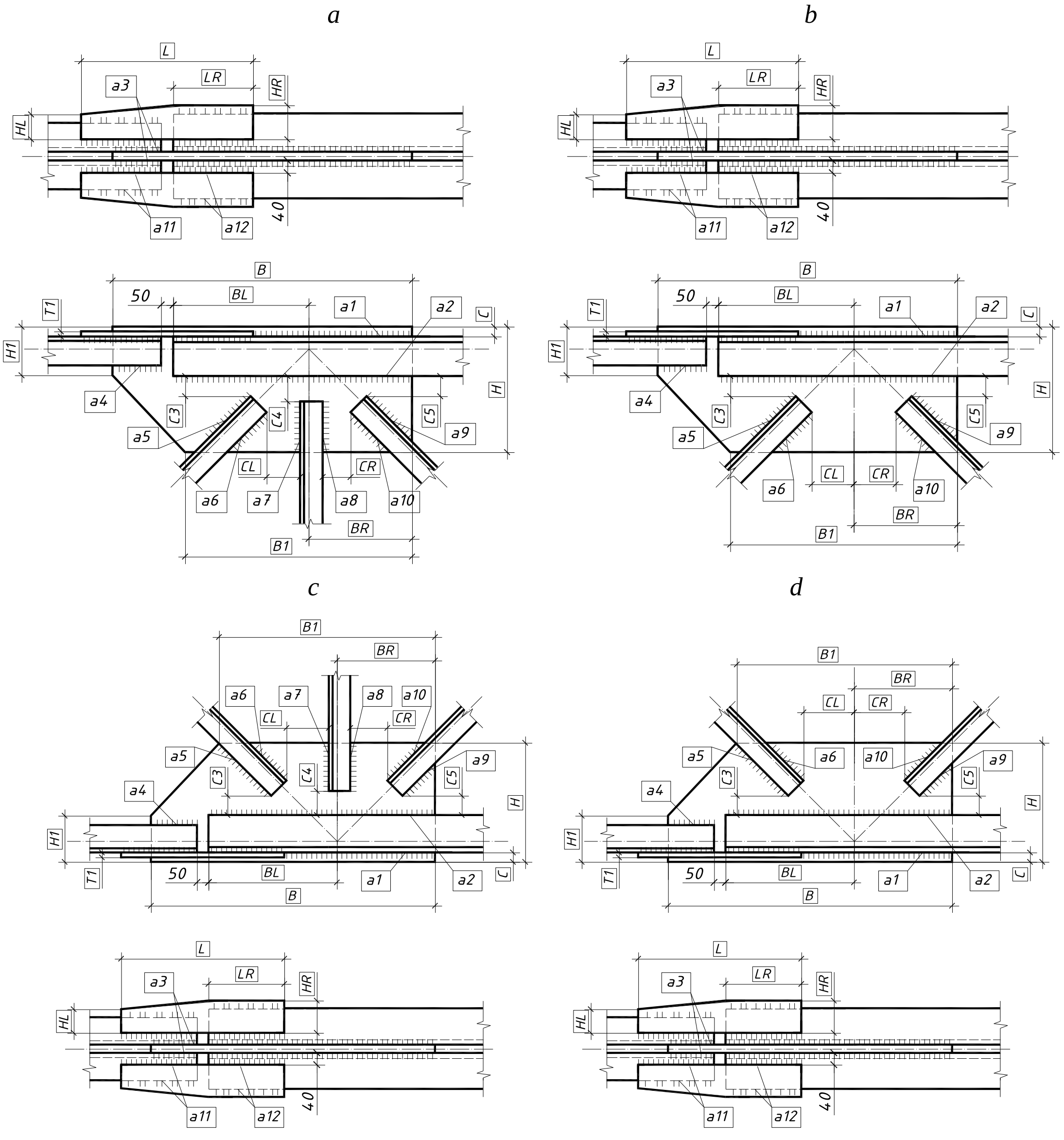
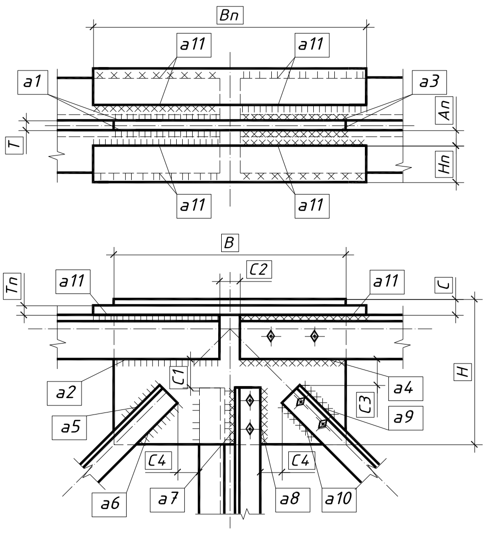
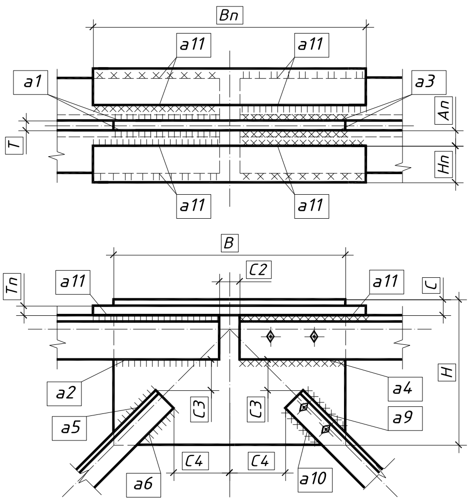


Figure 10.4.4-4. Joints where the chord changes its cross-section in trusses with elements made of double angles

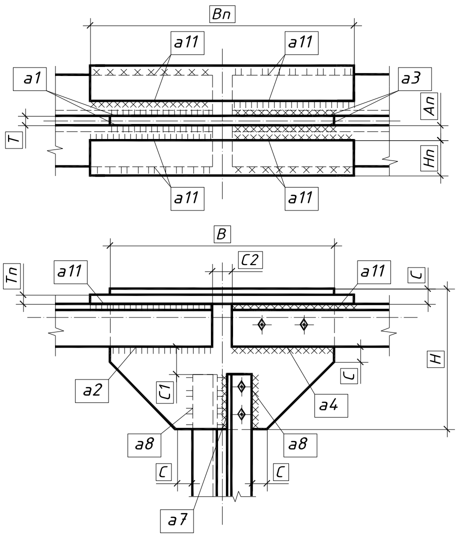
a


Bn
a11
a11
a1
a3
An
T
Hn
a11
a11
B
C2
Tn
a11
a11
C
H
a2
C1
a4
C3
a5
a9
C4
C4
a6
a7
a8
a10

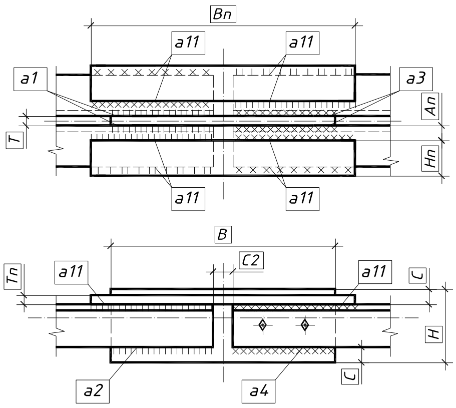
b


Bn
a11
a11
a1
a3
An
T
Hn
a11
a11
B
C2
Tn
a11
a11
C
H
a2
a4
C3
C3
a5
a9
C4
C4
a6
a10

c


Bn
a11
a11
a1
a3
An
T
Hn
a11
a11
B
C2
Tn
a11
a11
C
H
a2
C1
C
a4
a8
a8
C
C
a7

d


Bn
a11
a11
a1
a3
An
T
Hn
a11
a11
B
C2
Tn
a11
a11
C
H
C
a2
a4

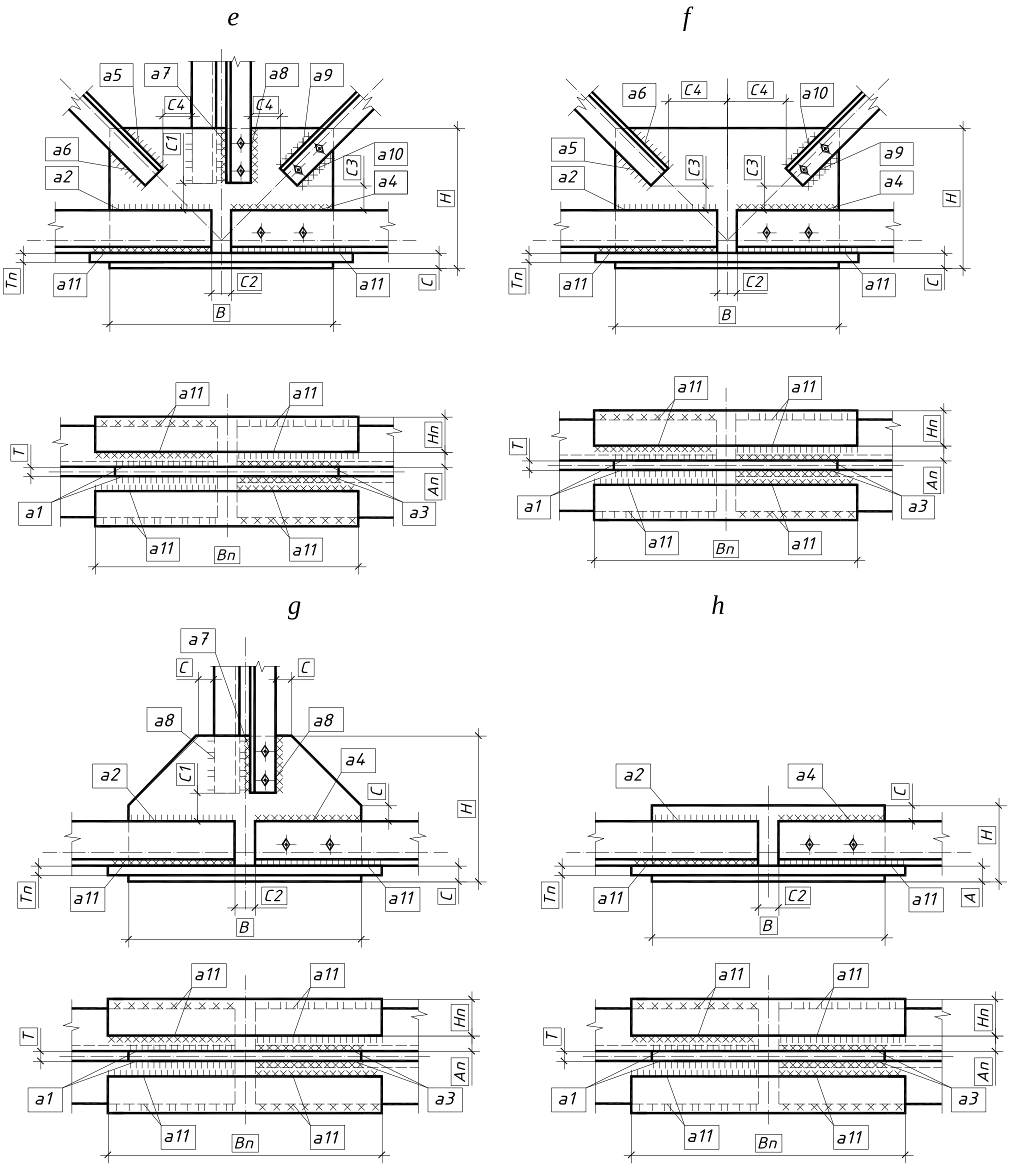


Figure 10.4.4-5. Erection joints in trusses with elements made of double angles

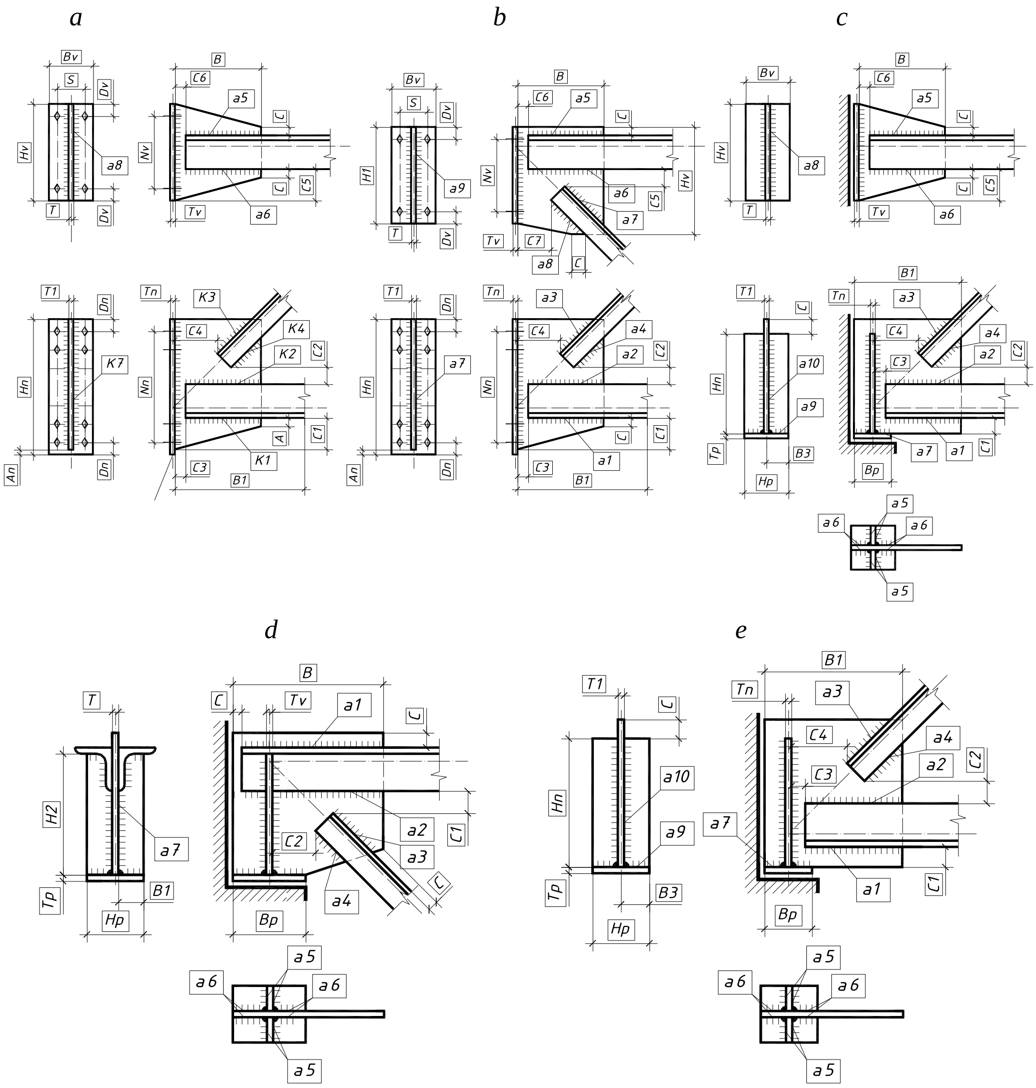


Figure 10.4.4-6. Support joints in trusses with elements made of double angles

The **Truss Panel Points** dialog box contains the following tabs: **Type of Joint** (Fig. 10.4.4-7), **Materials** (Fig. 10.4.4-8), **Forces** (Fig. 10.4.4-9), **Joint Members** (Fig. 10.4.4-10), **Structure** (Fig. 10.4.4-11), **Drawing** (Fig. 10.4.4-12) and **Interaction Curves** (Fig. 10.4.4-13).

The **Type of Joint** tab (Fig. 10.4.4-7) contains buttons which are used to select the type of a truss joint (**Regular Joint**, **Section Change**, **Erection Joint**, or **Support Joint**) and to specify its configuration. You can define the type of cross-section for the bar members of the truss by using the respective buttons in the **Structure** group.

This tab also enables to specify the materials used in a joint of a truss with the bars made of double angles or hollow sections. A steel grade for these members can be selected in the **Steel** dialog box (see Sec. 10.3.1) which is invoked by clicking the **Angle steel** and **Gusset plate steel** buttons.

The **Type of Joint** tab (Fig. 10.4.4-7) also contains the **Stamp** button used to fill in the stamp of the drawing, which will be generated automatically once the structural design of the truss joint is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

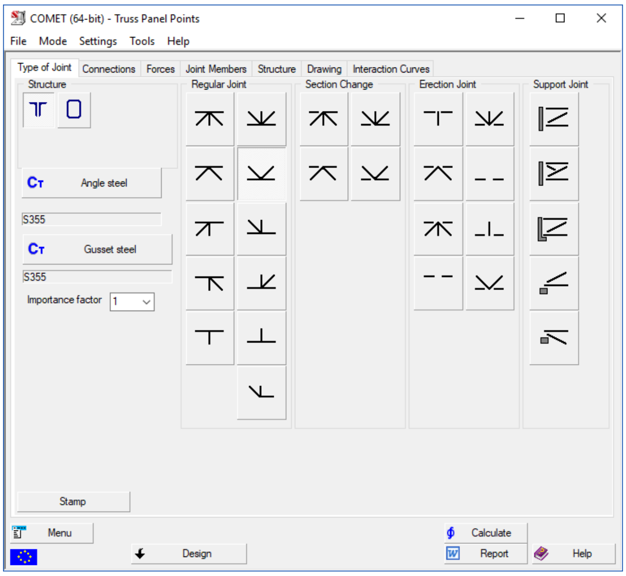


Figure 10.4.4-7. *The* **Type of Joint** *ta of the* **Truss Panel Points** *dialog box*

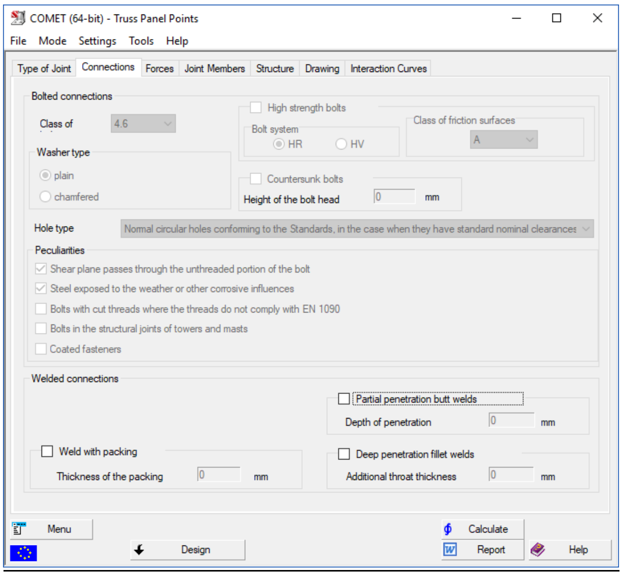


Figure 10.4.4-8. *The* **Connections** *tab of the* **Truss Panel Points** *dialog box*

The **Forces** tab (Fig. 10.4.4-9) is used to specify axial forces $N_{Ed,i}$ acting in the bar members of the truss joint. Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the values of internal forces for the current design combination of loads. There can be any number of design combinations of loads. The default units of measurement for axial forces are tonnes. The drawing next to the table of internal forces defines the positive directions of internal forces in the sections of the truss members.

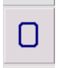
The **Joint Members** tab (Fig. 10.4.4-10) is used to specify the dimensions (width and height) of panels adjacent to the considered truss panel point (parameters *a*, *b*, *c*, and *d*). The default units of measurement for the dimensions of truss panels are meters. The **Section** group is used to specify the cross-sections of members connected in the considered joint and their orientation with respect to the truss plane.

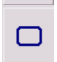
The **Type of section** buttons enable you to select:

 a section of double equal angles or of double unequal angles with the longer leg perpendicular to the truss plane;

 a section of double equal angles or of double unequal angles with the longer leg parallel to the truss plane;

 a section of equal angles arranged as a cross, which is used for the verticals of an erection joint;

 a rectangular hollow section with the longer side parallel to the truss plane;

 a rectangular hollow section with the longer side perpendicular to the truss plane.

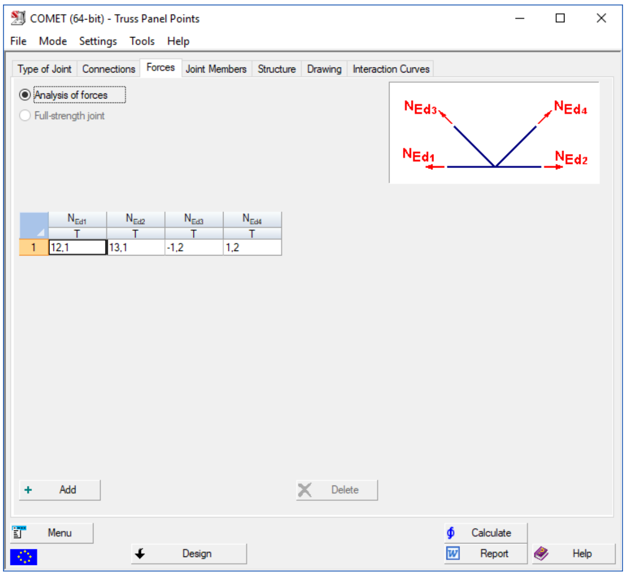


Figure 10.4.4-9. *The* **Forces** *tab of the* **Truss Panel Points** *dialog box*

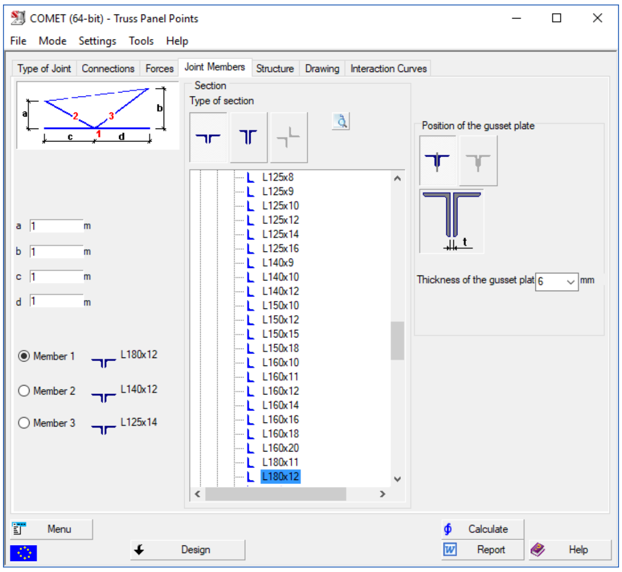


Figure 10.4.4-10. *The* **Joint Members** *tab of the* **Truss Panel Points** *dialog box*

It should be noted that the set of **Type of section** buttons depends on the design of the joint specified in the **Structure** group of the **Type of Joint** tab.

Each member of the truss joint is assigned a number (e.g., 

). To assign profiles to the truss members, you have to select the respective radio button of the member and select a profile from the list of assortments of rolled profiles. The specified cross-sections of the members of the truss joint can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ().

You have to specify the position of the gusset plate by clicking the respective buttons in the **Position of the gusset plate** group. The thickness of the gusset plate can be selected from the respective drop-down list, which provides a set of thickness values according to the assortment of sheet and plate steel.

The **Structure** tab (Fig. 10.4.4-11) provides a draft of the design for the truss joint.

To perform a check of the load-bearing capacity of the specified structural design of the truss joint, you have to specify all design parameters of the joint. The parameters include the sizes and thickness of structural members of the joint, leg lengths of welds, sizes which determine the mutual arrangement of members, diameters of bolts, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table on the left in the **Structure** tab. Leg lengths of the fillet welds are entered in the table at the bottom of the dialog. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

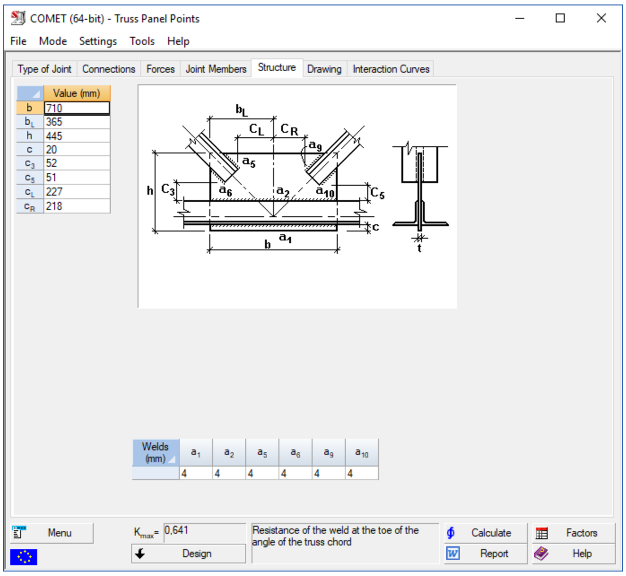


Figure 10.4.4-11. *The* **Structure** *tab of the* **Truss Panel Points** *dialog box*

Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the truss members adjacent to the considered joint, structural members of the joint (plates, end-plates, bearing stiffeners, bearing end-plates, etc.), welded and bolted connections in the joint at the specified (or previously selected) values of the parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the truss joint design of the MS stage will be generated.

Once you switch to the **Drawing** tab (Fig. 10.4.4-12), the application performs a check of the truss joint similarly to the **Calculate** mode. If the results of the check of the parameters of the joint elements do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the truss joint under various pairs of internal forces which can arise in the truss members adjacent to the joint are plotted in the **Interaction Curves** tab (Fig. 10.4.4-13). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of truss joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

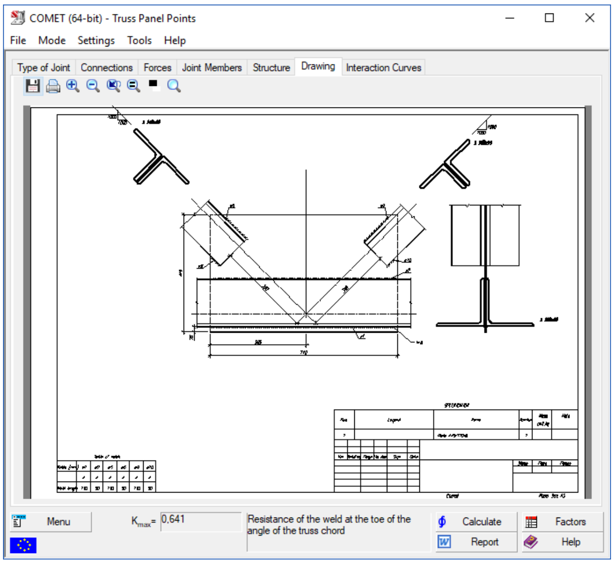


Figure 10.4.4-12. *The* **Drawing** *tab of the* **Truss Panel Points** *dialog box*

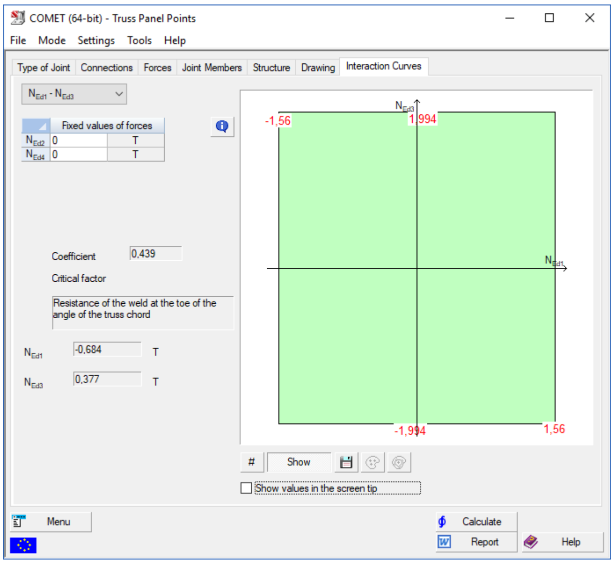


Figure 10.4.4-13. *The* **Interaction Curves** *tab of the* **Truss Panel Points** *dialog box*

A complete list of checks and values of the respective utilization factors of restrictions can be obtained by clicking the **Factors** button. The list of the load-bearing capacity checks of the members and connections of the truss joints performed by the application is given in Table 10.4.4-1.

Table 10.4.4-1. A list of the load-bearing capacity checks of the members and connections of the truss joints

Check	EN 1993-1-8:2005	EN 1993-1-1:2005
Truss joints from double angles:		
Resistance of the weld at the toe of the angle of the left diagonal of the truss	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the toe of the angle of the right diagonal of the truss	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the toe of the angle of the truss vertical	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the toe of the angle of the truss chord	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the heel of the angle of the left diagonal of the truss	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the heel of the angle of the right diagonal of the truss	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the heel of the angle of the truss vertical	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the weld at the heel of the angle of the truss chord	Sec. 4.5.3.2, 4.5.3.3	
Resistance of the gusset plate (assuming a load distribution at 30°)		Sec. 6.2.4
Punching resistance of the gusset plate using the model based on plastic deformations		Sec. 6.2.3(2)
Truss joints from rectangular (square) hollow sections:		
Joint resistance based on the chord face failure (plastic failure of the chord face) or chord plastification (plastic failure of the chord cross-section) under the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord side wall failure (or chord web failure) by yielding, crushing or instability (crippling or buckling of the chord side wall or chord web) under the compression left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	

Check	EN 1993-1-8:2005	EN 1993-1-1:2005
Joint resistance based on the brace failure with reduced effective width (cracking in the welds or in the brace members) under the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the punching shear failure of a hollow section chord wall (crack initiation leading to rupture of the brace members from the chord member) under the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under reduced stresses for the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under normal stresses for the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the first design criterion for special types of welded joints between hollow section members according to Table 7.6 or 7.15 EC3	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the second design criterion for special types of welded joints between hollow section members according to Table 7.6 or 7.15 EC3	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord face failure (plastic failure of the chord face) or chord plastification (plastic failure of the chord cross-section) under the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord side wall failure (or chord web failure) by yielding, crushing or instability (crippling or buckling of the chord side wall or chord web) under the compression right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the brace failure with reduced effective width (cracking in the welds or in the brace members) under the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the punching shear failure of a hollow section chord wall (crack initiation leading to rupture of the brace members from the chord member) under the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under reduced stresses for the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under normal stresses for the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord face failure (plastic failure of the chord face) or chord plastification (plastic failure of the chord cross-section) under the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	

Check	EN 1993-1-8:2005	EN 1993-1-1:2005
Joint resistance based on the chord side wall failure (or chord web failure) by yielding, crushing or instability (crippling or buckling of the chord side wall or chord web) under the compression vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the brace failure with reduced effective width (cracking in the welds or in the brace members) under the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the punching shear failure of a hollow section chord wall (crack initiation leading to rupture of the brace members from the chord member) under the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under reduced stresses for the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Resistance of the welded connection under normal stresses for the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord shear failure under the left diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord shear failure under the right diagonal	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord shear failure under the vertical	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	
Joint resistance based on the chord shear failure	Sec. 7.4, (7.3), Table 7.1…7.6 Sec. 7.5, (7.4) Table 7.8…7.15	

10.4.5 Beam-To-Column Joints

The **Beam-To-Column Joints** mode enables to design and check the load-bearing capacity of beam-to-column joints. The considered joints can be classified into the following types by the conditions of resistance to internal forces acting in the joint and by the possibility of the mutual rotation of the beam with respect to the column:

- rigid ones, which nearly immobilize the beam with respect to the column (Fig. 10.4.5-1);
- pinned ones, which can hardly resist the rotation of the beam with respect to the column (Fig. 10.4.5-2).

Fig. 10.4.5-1 presents the following types of structural designs for rigid beam-to-column joints implemented in the application: a welded joint (Fig. 10.4.5-1, *a*) and joints with high strength bolts (Fig. 10.4.5-1, *b…g*). Structural designs of the beam-to-column joints which use a bearing end-plate without an angle cleat (Fig. 10.4.5-1, *c…g*) are usually developed as friction joints with high strength bolts. In cases when a significant bending moment acts in the joint, and its value exceeds the resistance of the beam, the application provides the designs with haunches (Fig. 10.4.5-1, *f, g*).

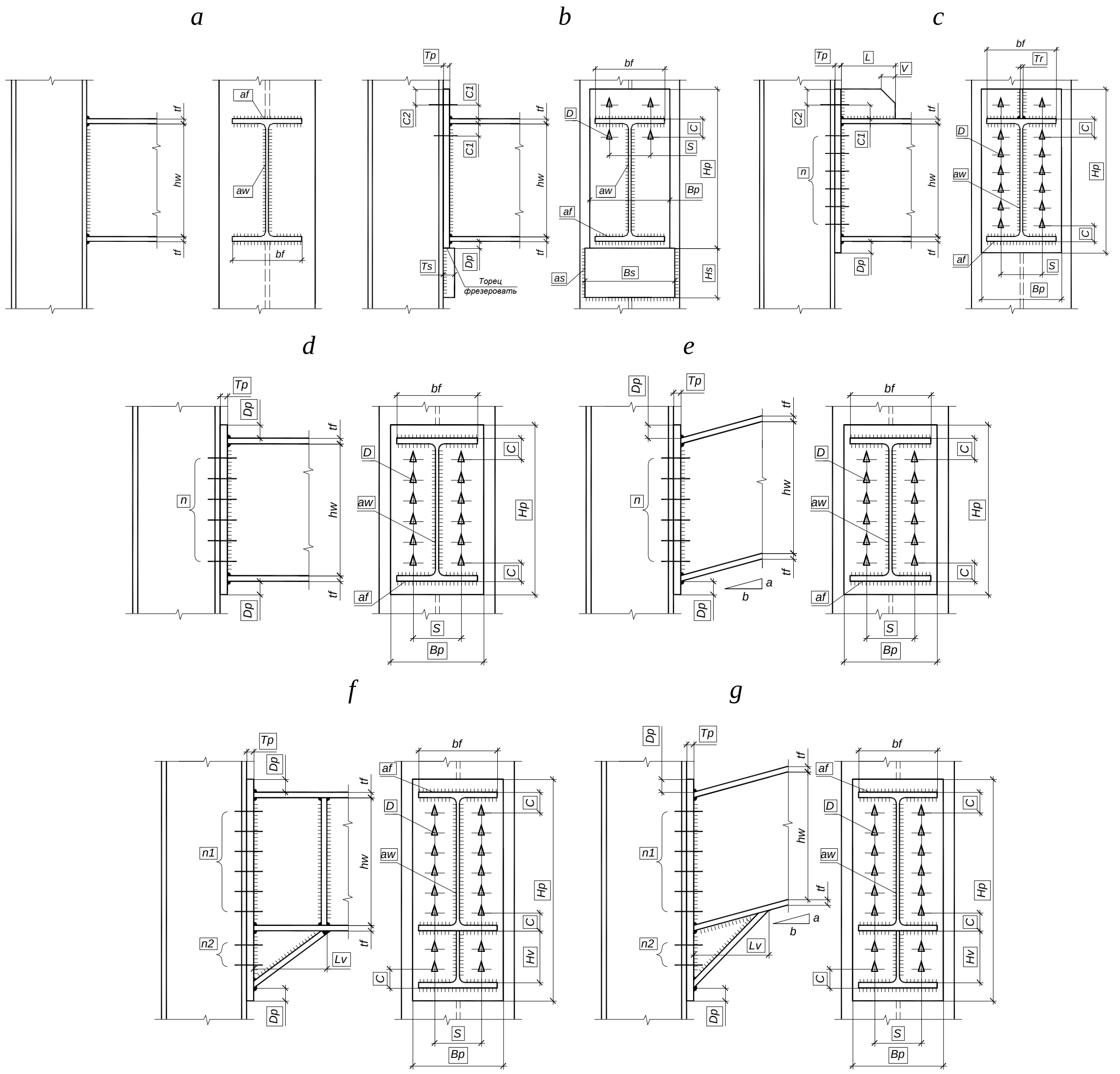


Figure 10.4.5-1. Types of designs for the rigid beam-to-column joints

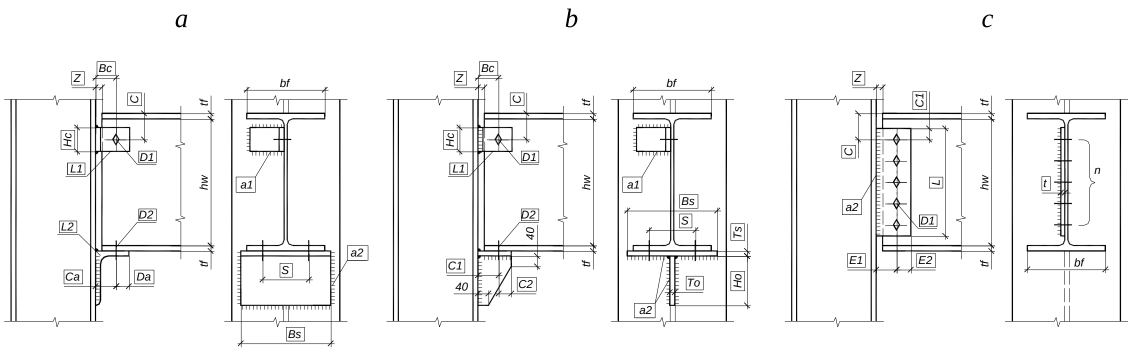


Figure 10.4.5-2. Types of designs for the pinned beam-to-column joints

This mode performs the following checks in compliance with EN 1993-1-8:2005:

- resistance of end-plates, stiffeners and haunches;
- resistance of bolted and welded connections included in the joint;
- a number of structural and assortment constraints.

The **Beam-To-Column Joints** dialog box contains the following tabs: **Configuration** (Fig. 10.4.5-3), **Connections**, **Forces** (Fig. 10.4.5-6), **Beam 1** (Fig. 10.4.5-7), **Beam 2** (Fig. 10.4.5-8), **Drawing** (Fig. 10.4.5-9) and **Interaction Curves** (Fig. 10.4.5-10).

First you have to define the configuration of your joint in the **Configuration** tab (Fig. 10.4.5-3). If the beam is connected to the column on the left, you have to check the checkbox (☑) **Beam 1**, and if it is connected to the column on the right, check the **Beam 2** checkbox. If there are 2 beams in the joint (both on the left and on the right), both checkboxes (**Beam 1** and **Beam 2**) have to be checked. The group of buttons **Beam position** is used to define the position of beams in their connection to the column. The upper position of the beams corresponds to a joint between the roof beams and the head of the column, and the middle position corresponds to the joint between the floor beams and the column.

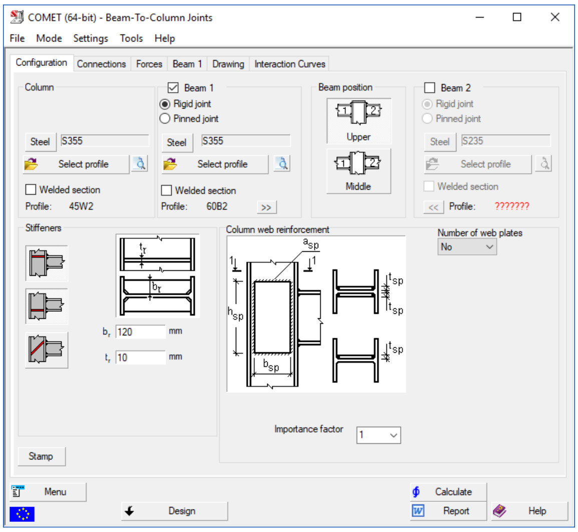


Figure 10.4.5-3. *The* **Configuration** *tab of the* **Beam-To-Column Joints** dialog box

The **Column** group of controls is used to define the cross-section and the steel grade for the column. A steel grade can be selected from the **Steel** drop-down list of the **Column** group (or by clicking the **Steel** button if the analysis and design are performed according to EN 1993-1-8:2005). If the column cross-section type is a welded I-section, check the **Welded section** checkbox and click the **Select profile** button. The **Sizes of section** dialog box will appear (Fig. 10.4.5-4) where you have to specify the sizes of the column cross-section: height h_w and thickness t_w of the column web, width b_f and thickness t_f of the column flange; and then click the **OK** button. The default units of measurement for the sizes of the column cross-section are millimeters.

The thickness of the flanges and of the web can be either entered manually or selected from the drop-down lists, which contain the set of thickness values according to the assortment of sheet and plate steel. If the column cross-section type is a rolled one, click the **Select profile** button (the **Welded section** checkbox should be unchecked). Select an assortment and the profile number in this assortment from the tree-like list in the **Select profile** dialog box (Fig. 10.4.5-5), and then click the **OK** button.

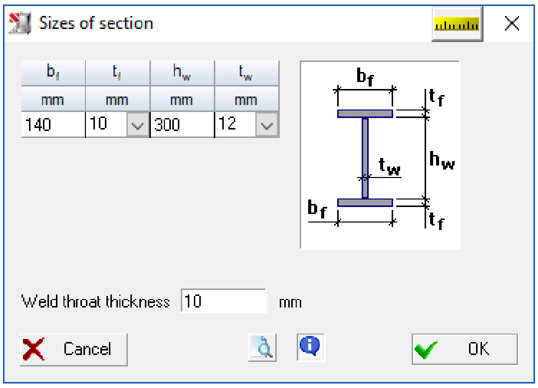


Figure 10.4.5-4. *The* **Sizes of section** *dialog box*

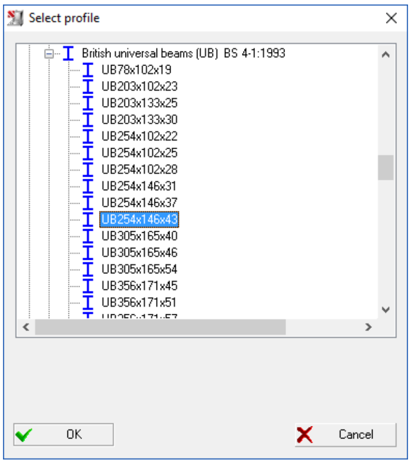
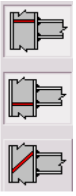


Figure 10.4.5-5. *The* **Select profile** *dialog box*

The specified column cross-section can be checked in the **Preview** window, which can be invoked by clicking the **Preview** button ().

In the same way you can define the steel grade and the cross-section for the beams of the joint using the controls of the respective groups (**Beam 1** and **Beam 2**). You also have to specify the type of connection for each beam using the respective radio buttons: **Rigid joint** or **Pinned joint**. Clicking the button >> in the **Beam 1** group will assign the type and sizes of the cross-section selected for beam 1 (on the left) to beam 2 (on the right). Clicking the button << in the **Beam 2** group will assign the type and sizes of the cross-section specified for beam 2 to beam 1.

The importance factor has to be specified in the **Importance factor** drop-down list.



When there are significant bending moments acting in the beam-to-column joints, it often becomes necessary to additionally reinforce the column web by transverse stiffeners. The arrangement of the stiffeners of the column web is specified by clicking one of the buttons of the **Stiffeners** group, which define their arrangement in the level of the upper beam flange, in the level of the lower beam flange, or obliquely.

When the analysis and design are performed according to EN 1993-1-8:2005, the sizes of the stiffeners b_r and t_r are selected by the application. Moreover, the column web can be reinforced by plates which are welded to the column web along the contour. The number of welded plates (on one side of the column web or on two sides – one or two plates respectively) is selected from the **Number of web plates** drop-down list. The sizes of welded plates (height h_{sp}, width b_{sp} and thickness t_{sp}), and the weld throat thickness a_{sp} are specified by the user in the respective text fields, or they can be automatically selected by the application if this reinforcement has been selected and all the sizes are specified as zero.

Clicking the **Stamp** button opens a dialog box which enables to fill in the stamp of the drawing, which will be generated automatically once the structural design of the beam-to-column joint is completed. The **Stamp** dialog box is described in Sec. 9.5.1.

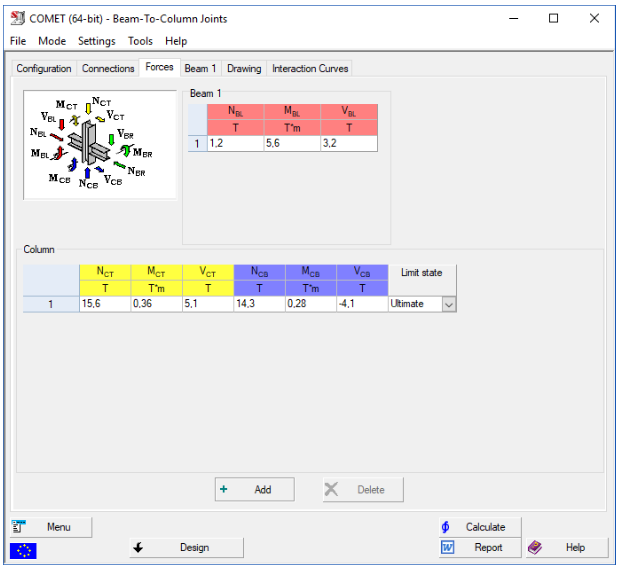


Figure 10.4.5-6. *The* **Forces** tab
of the **Beam-To-Column Joints** dialog box

The **Forces** tab (Fig. 10.4.5-6) is used to specify the internal forces acting in the beam-to-column joint. In the general case, any joint is subjected to an axial force N, bending moment M and its respective shear force Q transferred to it from the beam. It should be noted that in a pinned beam-to-column joint the bending moment, M, must be equal to zero. Moreover, the following forces are acting in the design column sections: axial force N, bending moments in both planes M_x, M_y, and their respective shear forces Q_y and Q_x. Forces acting in the design column sections are specified for the cross-sections above and below the considered joint.

The table used for specifying the values of internal forces for one or several design combinations of loads has an additional column, **Limit state**, where you have to select the limit state for the current combination of loads (ultimate or serviceability).

Clicking the **Add** button adds a new row to the table of internal forces, where you have to enter the design values of internal forces for the current combination of loads. There can be any number of design combinations of loads. The default units of measurement for axial and shear forces are tonnes, and for bending moments – tonne×meter. The drawing in top left corner of the tab defines the positive directions of internal forces.

The **Beam 1** tab (Fig. 10.4.5-7) contains a group of buttons to select a design for the beam-to-column joint (the beam is on the left of the column). If a rigid joint has been specified for beam 1 in the **Configuration** tab, the **Beam 1** tab will display structural designs for rigid beam-to-column joints.

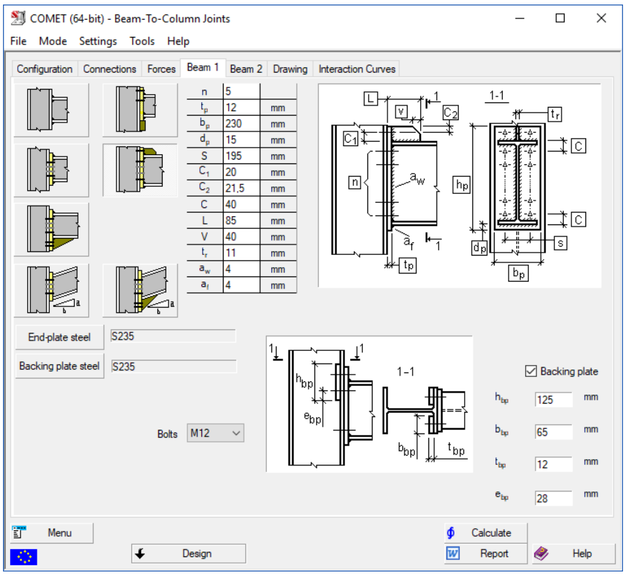


Figure 10.4.5-7. *The* **Beam 1** *tab*
of the **Beam-To-Column Joints** *dialog box*
(rigid joint between beam 1 and the column)

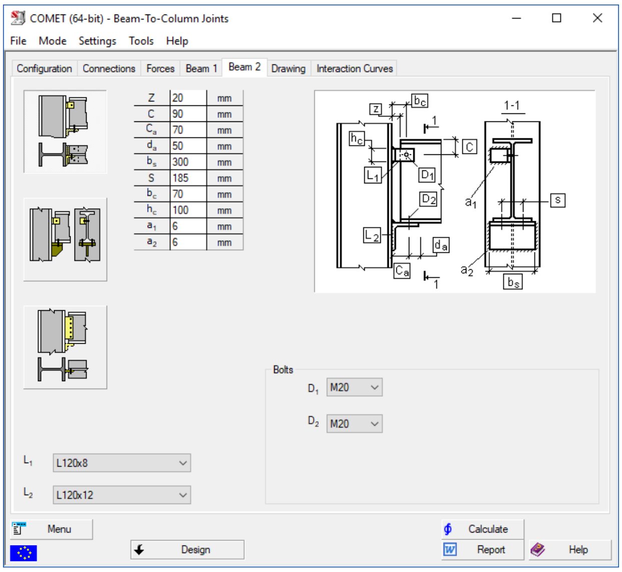


Figure 10.4.5-8. *The* **Beam 2** *tab*
of the **Beam-To-Column Joints** *dialog box*
(pinned joint between beam 2 and the column)

A slope of the beam can be specified for some types of beam-to-column joints (Fig. 10.4.5-1, *e, g*) by entering the values of dimensionless parameters *a* and *b* (Fig. 10.4.5-1, *e, g*) in the **Slope** group.

The **Connections** tab is used to specify information on the peculiarities of welded and bolted connections of the beam-to-column joints, i.e.: the class of bolts, the type of bolt hole, the system of high strength bolts, etc. The functionality of the **Connections** tab is described in detail in the **Beam Splices** section (see Sec. 10.4.3). In addition to the steel grades of the bearing joint members (columns and beams) specified earlier in the **Configuration** tab (Fig. 10.4.5-3), you can also specify the steel grades for the additional joint members (plates and end-plates) by clicking the **Plate steel** and **End-plate steel** buttons in the **Beam 1** tab (Fig. 10.4.5-7).

Moreover, you can specify the reinforcement of the column flange by a backing plate by checking the **Backing plate** checkbox. The sizes of the backing plate (height h_{bp}, width b_{bp}, thickness t_{bp}), and the distance from the center of the bolt hole to the edge of the backing plate e_{bp} can be specified in the respective text fields. If the user does not specify the values of these parameters, the application will select them automatically.

The **Beam 2** tab (Fig. 10.4.5-8) contains a group of buttons to select a design for the beam-to-column joint (the beam is on the right of the column). If a pinned joint has been specified for beam 2 in the **Configuration** tab, the **Beam 2** tab will display structural designs for pinned beam-to-column joints. The joints are made with bolts of normal strength (for joints with an angle cleat, Fig. 10.4.5-8-2, *a, b*) and with high strength bolts (for joints without an angle cleat, Fig. 10.4.5-8-2, *c*).

Equal angles according to GOST 8509-86 are used to fix the position of the beam with respect to the column and also as angle cleats in some types of pinned joints (Fig. 10.4.5-2, *a, b*). If the user knows the sizes of these angles, they can be specified in the drop-down lists L_1 and L_2.

It should be noted that the computer-aided calculation of a pinned beam-to-column joint with a plate assumes that the cross-sectional dimensions of the plate are taken as close as possible to the cross-sectional dimensions of the beam (in particular, the height of the plate on the beam web). In the case when the height of the plate is assumed to be less than the effective height of the beam web, the stress concentrations should be calculated and taken into account in the check, which is not implemented in the program due to the absence of any standard methods for performing such a calculation.

To perform a check of the load-bearing capacity of the specified structural design of the beam-to-column joint, you have to specify all design parameters of the joint. The parameters include the sizes and thickness of structural members of the joint, diameters of bolts, sizes which determine the mutual arrangement of members, leg lengths of welds, the number of bolts, the number of bolt rows, etc. The parameters of the joint are entered in the table of the **Beam 1** or/and **Beam 2** tab. The default units of linear measurement are millimeters.

Clicking the **Design** button drops down a menu. If the first item, ***All parameters are not specified***, is selected, the automatic selection of all parameters of the joint design is performed and it is assumed that the parameters of the joint design are not specified (are equal to zero), and their previously specified values are ignored. If the **Some parameters are specified** menu item is selected, the program will automatically determine the values of the undefined (are equal to zero) parameters with fixed values of the specified parameters.

The automatic selection of the beam-to-column joint design is performed on the basis of the analysis of its sensitivity with respect to the variation of the controlled parameters of the joint taking into account the conditions of the adequate resistance and structural constraints defined by the standards (see Sec. 8.2). Clicking the **Calculate** button will perform the check of the load-bearing capacity of the sections of the elements adjacent to the joint (column and beams), structural elements of the joint (end-plates, plates, angle cleats etc.), and their connections (welded and bolted) at the specified (or previously selected) values of all parameters of the joint according to the codes.

After clicking the **Design** or **Calculate** button the maximum utilization factor of restrictions (the most dangerous) will be displayed in the $\mathbf{K_{max}}$ field located in the lower part of the dialog box, and the type of the standard check (strength, stability, local stability, etc.) in which this maximum took place will be indicated, and a drawing of the beam-to-column joint design of the MS stage will be generated.

A complete list of checks and values of the respective utilization factors of restrictions can be obtained by clicking the **Factors** button. The list of the load-bearing capacity checks of the members and connections of the rigid and pinned beam-to-column joints performed by the application is given in Tables 10.4.5-1 and 10.4.5-2.

Once you switch to the **Drawing** tab (Fig. 10.4.5-9), the application performs a check and design of the joint, similarly to the **Calculate** mode. If the results of analysis of the parameters of the joint members do not contradict the structural and standard requirements, a drawing of the joint design of the MS stage will be generated.

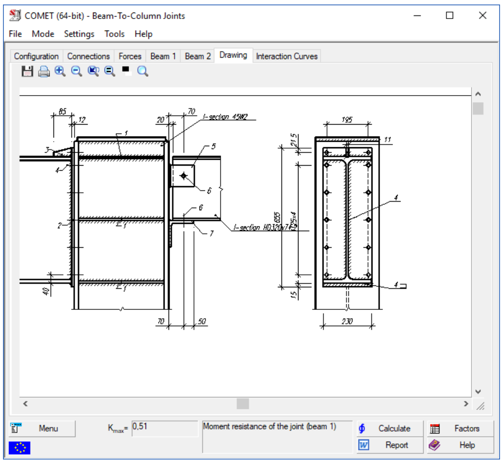


Figure 10.4.5-9. *The* **Drawing** *tab of the* **Beam-To-Column Joints** *dialog box*

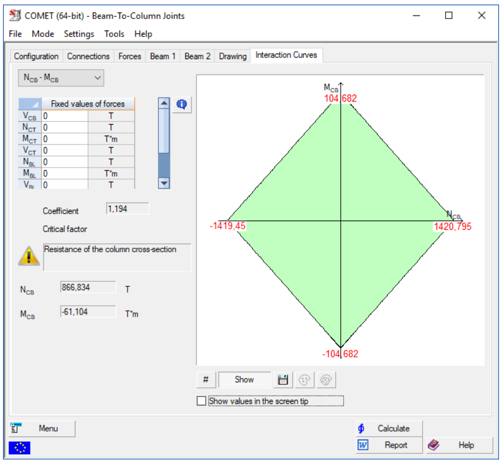


Figure 10.4.5-10. *The* **Interaction Curves** *tab of the* **Beam-To-Column Joints** *dialog box*

The curves enclosing an area of the load-bearing capacity of the specified (or selected) design of the beam-to-column joint under various pairs of internal forces which can arise in the members adjacent to the joint (in the column and in the girders) are plotted in the **Interaction Curves** tab (Fig. 10.4.5-10). Click the **Show** button to generate such a curve.

A drop-down list serves to select a pair of variable internal forces, and all other forces are taken as values specified in the **Fixed values** group.

Using your mouse pointer, you can explore the area of the load-bearing capacity of the beam-to-column joint shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the variable forces; their values are displayed in the respective fields. Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the current position of the pointer in the plot area of the interaction curve.

Table 10.4.5-1. A list of the load-bearing capacity checks of the members and connections of the rigid beam-to-column joints according to EN 1993-1-1:2005 and EN 1993-1-8:2005

Check	Type of joint	EN 1993-1-8	EN 1993-1-1
Moment resistance of the joint	Fig. 10.4.5-1	Sec. 6.2.5.1, (6.23), (6.24), (6.25)	
Resistance of the weld of the beam web taking into account the weld reinforcement in the case of a plastic analysis of the system	Fig. 10.4.5-1, *a*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the weld of the beam flange taking into account the weld reinforcement in the case of a plastic analysis of the system	Fig. 10.4.5-1, *a*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the welded connection between the beam web and the end-plate	Fig. 10.4.5-1, *b, c, d, e, f, g*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	

Check	Type of joint	EN 1993-1-8	EN 1993-1-1
Resistance of the welded connection between the beam flange and the end-plate	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Shear resistance of the column web	Fig. 10.4.5-1	Sec. 6.2.4.1, (6.7), (6.8)	
Resistance of the column web subject to transverse compression	Fig. 10.4.5-1	Sec. 6.2.4.2, (6.9)-(6.14)	
Resistance of the column web subject to transverse tension	Fig. 10.4.5-1	Sec. 6.2.4.3, (6.15)-(6.19a), (6.19b)	
Resistance of the column web subject to transverse bending	Fig. 10.4.5-1	Sec. 6.2.4.4, Table 6.4, Table 6.5, (6.20)	
Resistance of the welded connection between the angle cleat and the column flange	Fig. 10.4.5-1, *b*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the bearing stiffener (end-plate) subject to local end bearing	Fig. 10.4.5-1, *b*		
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the shear resistance of a bearing-type connection	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the bearing resistance of a bearing-type connection	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the slip resistance at the serviceability limit state of a slip-resistant connection at serviceability limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the shear resistance at the ultimate limit state of a slip-resistant connection at serviceability limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the bearing resistance at the ultimate limit state of a slip-resistant connection at serviceability limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the slip resistance at the ultimate limit state of a slip-resistant connection at ultimate limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the bearing resistance at the ultimate limit state of a slip-resistant connection at ultimate limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the plastic resistance of the net cross-section allowing for the weakening by holes at the ultimate limit state of a slip-resistant connection at ultimate limit state	Fig. 10.4.5-1, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the tension resistance of a connection with non-preloaded bolts	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.2 (2), Table 3.2, Sec. 3.6.1, Table 3.4	

Check	Type of joint	EN 1993-1-8	EN 1993-1-1
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the punching shear resistance of a connection with non-preloaded bolts	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.2 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the tension resistance of a connection with preloaded bolts	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.2 (3), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the punching shear resistance of a connection with preloaded bolts	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Sec. 3.4.2 (3), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the end-plate and the column flange calculated on the basis of the combined shear and tension resistance of a connection	Fig. 10.4.5-1, *b*, *c*, *d*, *e*, *f*, *g*	Table 3.4	
Resistance of the beam section weakened by holes subject to a shear force	Fig. 10.4.5-1		Sec. 6.2.6, (6.12)
Resistance of the beam section weakened by holes subject to a bending moment	Fig. 10.4.5-1		Sec. 6.2.5, (6.10)
Resistance of the beam section weakened by holes subject to a tensile force	Fig. 10.4.5-1		Sec. 6.2.3, (6.5)
Resistance of the beam section weakened by holes subject to a compressive force	Fig. 10.4.5-1		Sec. 6.2.4, (6.9)
Resistance of the beam section weakened by holes subject to a bending moment and a shear force	Fig. 10.4.5-1		Sec. 6.2.8, (6.25)
Resistance of the beam section weakened by holes subject to a bending moment and an axial force	Fig. 10.4.5-1		Sec. 6.2.9.1, (6.26)
Resistance of the beam section weakened by holes subject to a bending moment, shear and axial forces	Fig. 10.4.5-1		Sec. 6.2.10
Notes: see Table 10.4.1-1.			

Table 10.4.5-2. A list of the load-bearing capacity checks of the members and connections of the pinned beam-to-column joints according to EN 1993-1-1:2005 and EN 1993-1-8:2005

Check	Type of joint	EN 1993-1-8	EN 1993-1-1
Shear resistance of the column web	Fig. 10.4.5-2	Sec. 6.2.4.1, (6.7), (6.8)	
Resistance of the bolted connection between the plate and the beam web calculated on the basis of the shear resistance of a bearing-type connection	Fig. 10.4.5-2, *c*	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the plate and the beam web calculated on the basis of the bearing resistance of a bearing-type connection	Fig. 10.4.5-2, *c*	Sec. 3.4.1 (2), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the plate and the beam web for a slip-resistant connection at serviceability limit state	Fig. 10.4.5-2, *c*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	
Resistance of the bolted connection between the plate and the beam web at ultimate limit state	Fig. 10.4.5-2, *c*	Sec. 3.4.1 (3), 3.4.1 (4), Table 3.2, Sec. 3.6.1, Table 3.4	

Check	Type of joint	EN 1993-1-8	EN 1993-1-1
Resistance of the welded connection between the plate and the column flange under reduced stresses	Fig. 10.4.5-2, *c*	Sec. 4.5.3, (4.1)	
Resistance of the welded connection between the plate and the column flange under normal stresses	Fig. 10.4.5-2, *c*	Sec. 4.5.3, (4.1)	
Resistance of the welded connection between the plate and the column flange calculated conservatively without taking into account the orientation of the welds	Fig. 10.4.5-2, *c*	Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the welded connection between the cantilever angle and the column web under reduced stresses	Fig. 10.4.5-2, *a*, *b*	Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4)	
Resistance of the plate section weakened by holes subject to a shear force	Fig. 10.4.5-2, *c*		Sec. 6.2.6, (6.12)
Resistance of the plate section weakened by holes subject to a bending moment	Fig. 10.4.5-2, *c*		Sec. 6.2.5, (6.10)
Resistance of the plate section weakened by holes subject to a tensile force	Fig. 10.4.5-2, *c*		Sec. 6.2.3, (6.5)
Resistance of the plate section weakened by holes subject to a compressive force	Fig. 10.4.5-2, *c*		Sec. 6.2.4, (6.9)
Resistance of the plate section weakened by holes subject to a bending moment and a shear force	Fig. 10.4.5-2, *c*		Sec. 6.2.8, (6.25)
Resistance of the plate section weakened by holes subject to a bending moment and an axial force	Fig. 10.4.5-2, *c*		Sec. 6.2.9.1, (6.26)
Resistance of the plate section weakened by holes subject to a bending moment, shear and axial forces	Fig. 10.4.5-2, *c*		Sec. 6.2.10
Shear resistance of the plate in the section weakened by a set of holes	Fig. 10.4.5-2, *c*	Sec. 3.10.2, (3.9), (3.10)	
Shear resistance of the beam in the section weakened by a set of holes	Fig. 10.4.5-2, *c*	Sec. 3.10.2, (3.9), (3.10)	
Resistance of the beam section weakened by holes subject to a shear force	Fig. 10.4.5-2		Sec. 6.2.6, (6.12)
Resistance of the beam section weakened by holes subject to a bending moment	Fig. 10.4.5-2		Sec. 6.2.5, (6.10)
Resistance of the beam section weakened by holes subject to a tensile force	Fig. 10.4.5-2		Sec. 6.2.3, (6.5)
Resistance of the beam section weakened by holes subject to a compressive force	Fig. 10.4.5-2		Sec. 6.2.4, (6.9)
Resistance of the beam section weakened by holes subject to a bending moment and a shear force	Fig. 10.4.5-2		Sec. 6.2.8, (6.25)
Resistance of the beam section weakened by holes subject to a bending moment and an axial force	Fig. 10.4.5-2		Sec. 6.2.9.1, (6.26)
Resistance of the beam section weakened by holes subject to a bending moment, shear and axial forces	Fig. 10.4.5-2		Sec. 6.2.10
Notes: see Table 10.4.1-1.			

10.5 Design Codes the Requirements of Which are Implemented in COMET Eurocode 3

Mode	References to sections of standards or codes at the calculations according to EN 1993-1-1:2005 and EN 1993-1-8:2005
Steel	EN 10025-2:2004, EN 10025-3:2004, EN 10025-4:2004, EN 10025-5:2004, EN 10025-6:2004, EN 10210-1:2006, EN 10219-1:2006
Assortment of Rolled Profiles	
Bolts	EN ISO 898-1:1999-12
Concrete Classes (EN 1992)	EN 1992-1-1:2004
Rigid Column Bases	Sec. 6.2.3, Sec. 6.2.6.1, Sec. 6.2.6.2, Sec. 6.2.6.3, Table 6.7, Sec. 6.2.4.12, Sec. 3.4.1 (2), Sec. 3.4.2 (2), Table 3.4, Sec. 3.6.1 (1), Sec. 3.6.1 (3), Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4) **EN 1993-1-8:2005** Sec. 6.2.3, (6.5), Sec. 6.2.4, (6.9), Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25), Sec. 6.2.9.1, (6.26), Sec. 6.2.10 **EN 1993-1-1:2005**
Nominally Pinned Column Bases	Sec. 3.4.1 (2), Sec. 3.4.2 (2), Table 3.4, Sec. 3.6.1 (1), Sec. 3.6.1 (3), Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4), Sec. 6.2.3, (6.4)-(6.6), Sec. 6.2.4.12, Sec. 6.2.6.2 **EN 1993-1-8:2005** Sec. 6.2.3, (6.5), Sec. 6.2.4, (6.9), Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25), Sec. 6.2.9.1, (6.26), Sec. 6.2.10 **EN 1993-1-1:2005**
Beam Splices	Sec. 3.4.1 (2)-(4), Table 3.2, Sec. 3.6.1, Table 3.4, Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4), Sec. 6.2.5.1, (6.23) **EN 1993-1-8:2005** Sec. 6.2.3, (6.5), Sec. 6.2.4, (6.9), Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25), Sec. 6.2.9.1, (6.26), Sec. 6.2.10 **EN 1993-1-1:2005**
Truss Panel Points	
Beam-To-Column Joints	Sec. 3.4.1 (2)-(4), Table 3.2, Sec. 3.6.1, Table 3.4, Sec. 3.10.2, (3.9), (3.10), Sec. 4.5.3, (4.1), Sec. 4.5.4, (4.2)-(4.4), Sec. 6.2.4.1, (6.7), (6.8), Sec. 6.2.4.2, (6.9)- (6.14), Sec. 6.2.4.3, (6.15)- (6.19a), (6.19b), Sec. 6.2.4.4, Table 6.4, Table 6.5, (6.20), Sec. 6.2.5.1, (6.23), (6.24), (6.25) **EN 1993-1-8:2005** Sec. 6.2.3, (6.5), Sec. 6.2.4, (6.9), Sec. 6.2.5, (6.10), Sec. 6.2.6, (6.12), Sec. 6.2.8, (6.25), Sec. 6.2.9.1, (6.26), Sec. 6.2.10 **EN 1993-1-1:2005**

11. Monolit

11.1 General Information

Monolit enables you to design reinforced concrete monolithic ribbed and flat floor slabs. As the initial data, this program can use the results of selection of the reinforcement in the elements of reinforced concrete structures, obtained using either SCAD or other programs. The system has been developed in compliance with the current standards:

SNiP 2.03.01-84*. Concrete and reinforced concrete structures. Construction standards and regulations;

SNiP 52-01-2003 (SP 52-101-03). Concrete and reinforced concrete structures;

SP 63.13330. Concrete and reinforced concrete structures;

GOST 21.201-2011 and GOST 21.501-2011 (DSTU B A.2.4-7-95). System of design documents for construction. Rules for execution of architectural and construction working drawings;

GOST 21.1101-2013 ((DSTU B A.2.4-4-99). System of design documents for construction. Main requirements for design and working documents;

GOST 7473-94 (DSTU B V.2.7-96-2000). Ready-mixed concrete. Specifications;

GOST 14098-91. Welded joints of reinforcement and inserts for reinforced concrete structures. Types, constructions and dimensions.

Moreover, the following publications were used when developing **Monolit**:

Guide on designing concrete and reinforced concrete structures from heavy-weight and light-weight concrete without prestressing the reinforcements [1];

Manual to designing concrete and reinforced concrete structures from heavy-weight concrete (without prestressing) [3];

Reference manual “Designing of reinforced concrete structures” [2].

11.1.1 Designing of Ribbed Floors

Ribbed floors are formed by a system of slabs and beams bearing on columns and/or walls. The program enables to reinforce the structures by individual bars (without bends), welded cages and meshes manufactured with the help of spot welding of bar connections. The maximum size of the reinforcement items (meshes) can be specified by the user. If there are no such specifications, the maximum width of the welded meshes is taken by default as 3.0 m.

The program outputs the required set of floor working drawings, which includes:

- formwork layout with characteristic cross-sections;
- layouts of the upper and lower reinforcement of a slab (arrangement of the reinforcement meshes);
- beam and column reinforcement drawings;
- drawings of welded cages and meshes used for the reinforcement of slabs and beams;
- specification of details;
- steel consumption per slab and the overall specification;
- specifications for slabs and the overall specification.

All output documents are prepared in formats standard for the majority of printers – A4 and A3. However, they can be printed out using devices of other format, as well as a Plotter. The output documents are printed with the specified orientation: Portrait or Landscape (printer property). For some of them the orientation is fixed and does not depend on the printer settings (this is usually done for the convenience of the information display).

11.1.2 General Floor Layout

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

The required cross-section of longitudinal As (cm^2) and transverse Asw (cm^2/m) reinforcement in the main and secondary beams is specified at individual segments (not less than three and not more than five) along the length of the beam span. The reinforcement is assumed to be uniform within a segment. Reinforcement in supports is determined by the reinforcement cross-section specified at the near-support segments. When specifying five segments along the span length, the designing is performed with the account of rational distribution of the material (variable spacing between transverse bars and break of the span reinforcement at the segments where its full cross-section is not required).

11.1.3 Beams

Floor beams are reinforced in the spans by flat welded cages united into three-dimensional modules by welding transverse bars or by individual bars which form a spatial cage with the help of stirrups. If the beam width is up to 150 mm, one cage can be installed in a span. In case of a larger width, two and more welded cages are installed in the beam span, depending on the reinforcement cross-section and the conditions of its arrangement required by the design. Cages have one or two longitudinal main bars at the top and bottom cross-section edges. In those cases when there are two longitudinal bars, they are located in two levels with a gap. Depending on the beam cross-section dimensions, the diameter of the longitudinal bars being used is limited by the following values:

Beam height (mm)	<= 400	<=600	> 600
Maximum diameter of the cage longitudinal bars	22	30	40

Beam height is the distance from the bottom surface of the beam to its top surface that aligns with the top of the floor slab.

11.1.4 Slabs

The floor slabs are reinforced in the spans and on the supports by flat welded meshes or tied meshes with separate reinforcement bars without bends with the main bars running in one or two directions depending on the performance conditions of the given slab segment.

In all cases, the main reinforcement cross-section (cm^2/m) is taken to be not less than $0.0005*h_s$, where h_s is the slab thickness in cm. If the reinforcement has the design cross-section, the rebars are placed with a spacing of not more than 200 mm when the slab thickness is up to 150 mm, and not more than $1.5*h_s$ if h_s exceeds 150 mm. The bars of the distribution reinforcement of meshes (provided the design does not require reinforcement in this direction) are specified according to the welding conditions. The span slab reinforcement, which is installed at the bottom within the segment enclosed by the adjacent beams or walls, is specified in the form of one or several welded meshes depending on the size of the segment and the limit width of the meshes. If the latter is the case (several meshes), the mesh joints must have an overlap in the direction of the smaller cross-section of the mesh reinforcement. Besides, if there are less than two transverse bars within the joint, additional bars are added to the meshes to provide the required conditions.

The embedment length of slabs in brick walls can be specified by the user, but it must be not less than 120 mm. If the embedment length is not specified, it is taken as 120 mm by default. Wherever the slabs are simply supported on brick walls, the span welded meshes are extended beyond the edge of the support. If the embedment length does not allow to perform the required anchorage of longitudinal bars (10 diameters), additional transverse bars are welded to the meshes during the reinforcement placement.

The slabs near the supporting area are reinforced by flat welded (tied) meshes with the unidirectional main reinforcing bars, which are placed in the upper part of the slab. Meshes are installed with “staggered” joints, the gap between the extreme main bars not exceeding the spacing between these bars in the mesh. The cross-section of the main reinforcement of the supporting meshes is taken to be constant over the entire length of the support line and is determined by the maximum required cross-section of the reinforcement at the slab segments adjacent to the given support. Meshes located at the areas where the upper reinforcement is specified on the basis of the calculation (monolithic connection between the slab and the adjacent structures) are extended into the span by the distance equal to 0.25 of the span length, whereas at the bearing areas on brick walls (where according to the calculation reinforcement is not required) they are extended by 0.1 of the span length. At the monolithic connections between slabs and the adjacent structures of walls or beams on the extreme supports (one-sided abutting joint of the slab), the supporting meshes are extended beyond the edge of the support by the distance equal to the anchorage length. In this case, if necessary, the ends of meshes are bent up.

11.1.5 Columns

Only columns (posts) of rectangular or circular solid cross-section are considered here. Reinforcement is performed for one floor. A column is reinforced by a spatial tied cage consisting of longitudinal bars and stirrups (studs can be added if necessary), or by flat welded cages which form a spatial cage with the help of splice bars. The longitudinal reinforcement is placed symmetrically with respect to the cross-section axes along its facets. The longitudinal reinforcement has weldless lapped joints. Flat welded cages are taken with a one-sided arrangement of reinforcement. Main bars of not more than two different diameters are selected for the flat cage. The transverse reinforcement along the entire length of the column must have one diameter but different spacing between the bars. The transverse reinforcement is installed with a smaller spacing at the joint of the longitudinal reinforcement with the starter bars of the foundation or the ground floor than in the other part of the column.

11.2 MDI Interface

MONOLIT control is based on the multi-window interface which in the Windows environment is given the abbreviation MDI (Multiple Document Interface). This enables the user to work simultaneously with several projects, compare the initial data and the outputs of different projects, and to copy tabular data from one project to another. The restriction accepted for this program is that each project can have only one open window.

Windows of the following four types are distinguished in the program:

- main window;
- project tree window;
- initial data visualization window;
- analysis results visualization window.

Moreover, different dialog boxes are used to enter the initial data and to customize general parameters of the program.

11.2.1 Main Window

The main window of the program (Fig. 11.2.1-1) includes the menu, the toolbar and the work area where the project windows are opened.

The menu contents and the set of toolbars depend on the project status.

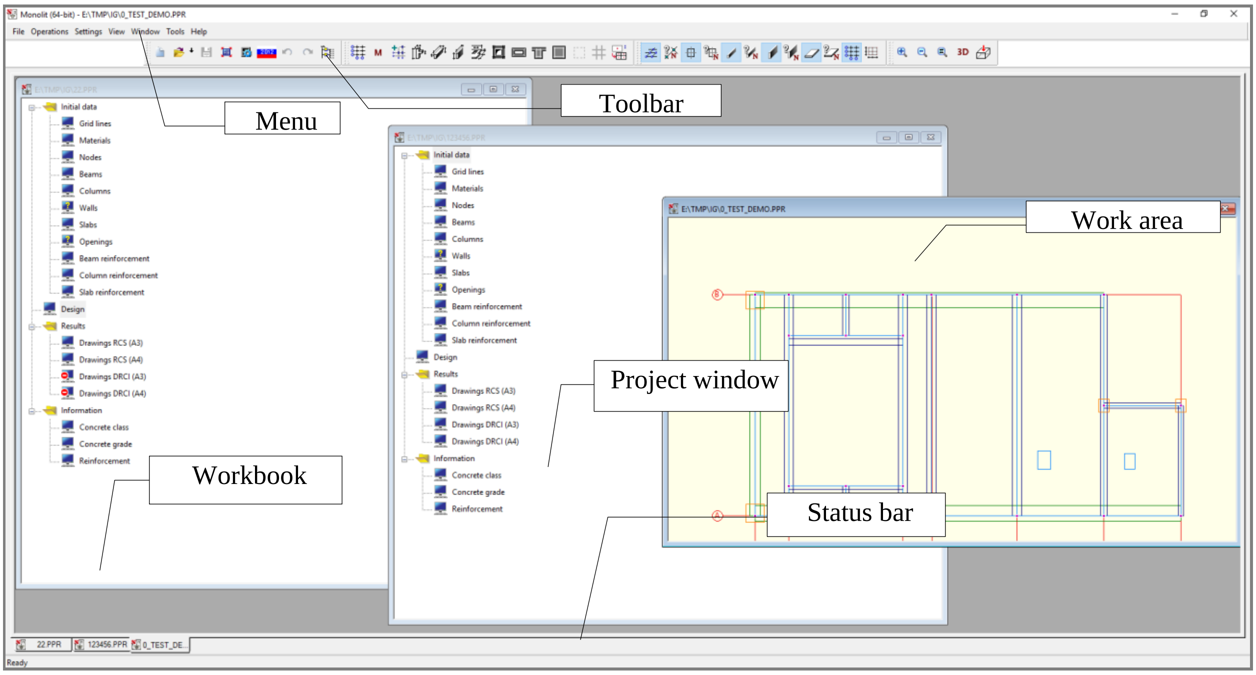


Figure 11.2.1-1. Main window of the program

Menus

The number of the menus and their content depend on the status of the project. For example, in the original state (there are no active projects), there are six menus: **File, Settings, View, Window, Tools,** and **Help,** while in the data preparation mode there are seven menus (the **Operations** menu is added). Table 11.2.1-1 lists the menu items and available operations for different statuses of the project.

Table 11.2.1-1.

Menu	**Complete set of operations in the menu**	**Available operations**				
		Initial state (there are no open projects or no changes in the project)	**Project tree is open**		**Project is open in the initial data entry mode**	**Project is open in the results analysis mode**
			changes in the project were not saved	**changes in the project were saved**		
File	New	*	*	*	*	*
	Open	*	*	*	*	*
	Close				*	*
	Save				*	*
	Save as…				*	*
	Output to DXF					*
	Print					*
	Send					
	Exit	*	*	*	*	
Operations	Undo				*	
	Redo				*	
	Grid lines				*	
	Materials				*	
	Nodes				*	

Menu	Complete set of operations in the menu	Initial state (there are no open projects or no changes in the project)	Project tree is open: changes in the project were not saved	Project tree is open: changes in the project were saved	Project is open in the initial data entry mode	Project is open in the results analysis mode
	Columns				*	
	Beams				*	
	Walls				*	
	Slabs				*	
	Openings				*	
	Slab reinforcement				*	
	Beam reinforcement				*	
	Column reinforcement				*	
	Additional reinforcement				*	
	Joint reinforcement details				*	
	Dividing slabs by beams				*	
Settings	Settings	*	*		*	
	Project Properties		*		*	
	Stamp (form 3)		*		*	*
	Stamp (form 4)		*		*	*
	Stamp (form 5)		*		*	*
	Stamp (form 6)		*		*	*
	Project Parameters		*		*	*
	Specifying Working Directories	*				
	Workbook	*	*		*	*
View	Zoom				*	*
	Toolbars	*	*		*	*
	Status Bar	*	*		*	*
Window	Cascade		*		*	*
	Tile		*		*	*
	Arrange Icons		*		*	*
Tools	Windows Calculator	*	*		*	*
	Formula Calculator	*	*		*	*
	Converting Units of Measurement	*	*		*	*
Help	Help Topics	*	*		*	*
	Check for Update	*	*		*	*
	About…	*	*		*	*

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

- **New (**project**)** — creates a new project;
- **Open … (**project**)** — opens a previously created project;
- **Save** — saves the current changes in the project under the same name;
- **Save as…** — saves the current changes in the project in a file under a new name;
- **Close** — ends the work with the current project;
- **Output to DXF** — creates files in the DXF format (AutoCAD) for the drawings selected in the results tree;

- **Print** — print out the drawings selected in the results tree;
- **Send** — sends a project file by e-mail;
- **Exit** — finishes the current session.

The bottom lines of the menu contain the file names of the latest projects (up to ten) the user has worked with.

The **Operations** menu is accessible only in the initial data preparation mode; it duplicates the corresponding buttons of the **Operations** toolbar and includes the following items:

- **Undo** — undo the last action;
- **Redo** — redo the previously undone action;
- **Grid lines** — enter the data required for plotting the grid lines;
- **Materials** — specification of the lists of the used materials (concrete, reinforcement);
- **Nodes** — specification of the snap nodes of structural members;
- **Columns** — setting the snaps and dimensions of columns;
- **Beams** — setting the snaps and dimensions of beams;
- **Walls** — setting the snaps and dimensions of walls;
- **Slabs** — setting the snaps and dimensions of floor slabs;
- **Openings** — setting the snaps and dimensions of openings;
- **Slab reinforcement** — specification of characteristics of the slab reinforcement;
- **Beam reinforcement** — specification of characteristics of the beam reinforcement;
- **Column reinforcement** — specification of characteristics of the column reinforcement;
- **Additional reinforcement** — specification of characteristics of the additional reinforcement (used only for flat slabs);
- **Joint reinforcement details** — description of the reinforcement details of the structural joints (used only for flat slabs);
- **Dividing slabs by beams** —create new slabs with their edges resting on the extreme and supporting nodes of the beams.

The **Settings** menu includes the following items:

- **Settings** — invokes the **Settings** dialog box, which enables to select the units of measurement, format and language of the report, and colors for the visualization of the structural members on layouts;
- **Project Properties**;
- **Stamp (form 3), (form 4), (form 5), (form 6)** — templates for filling the stamps of different forms;
- **Project Parameters** — invokes the **Project Parameters** dialog box, which enables to set the design system of the program;
- **Specifying Working Directories** — the **Specifying Working Directories** dialog box invoked by this operation enables to specify the names of catalogues from which the project file should be taken and where the working files and the result files should be put. You can change the purpose of catalogues only if the projects are not active;
- **Workbook** — checking this checkbox displays the tabs of the active projects.

The **View** menu is used for showing/hiding the toolbars and status bar, as well as for zooming the image in the design results analysis mode.

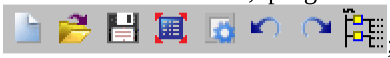
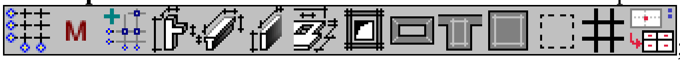
The **Window** menu includes the operations for arranging the windows when working simultaneously with several projects. It contains the following items: **Cascade, Tile** and **Arrange Icons** (the latter operation enables to arrange the list of icons belonging to the minimized windows of projects in the main window).

The **Tools** menu contains items for invoking a standard calculator of the Windows environment, a formula calculator, and a converter of units of measurement.

The **Help** menu provides the help information on using the application and on its functionality, information about the application, and it also contains an item that enables you to check for update www.scadsoft.com.

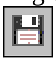
11.2.2 Toolbars in the Initial Data Entry Mode

The toolbar contains the following strips with the control buttons:

- **General operations** — includes the operations of creating, reading and saving a project, as well as a number of service operations (switching to the full-screen mode, program setting, undo and redo the previous action, switching to the project tree) ;
- **Operations** — this strip duplicates the **Operations** menu and enables to invoke different operations on the data preparation from the toolbar ;
- **Display filters** — contains a set of buttons which control displaying of the entered data on the structural layout in the process of the initial data preparation ;
- **Visualization** — this strip includes a set of controls for the visualization of the structural layout .

General Operations Toolbar

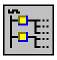
The toolbar contains the icons for invoking the following commands:

 — **New** (project),  — **Open**,  — **Save** — duplicate the respective items of the **File** menu.

 — **Full-screen mode** — switches to the maximum possible size of the **Monolit** window;

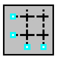
 — **Settings** — invokes the **Settings** dialog box, which enables to select the units of measurement, format and language of the report, and colors for the visualization of the structural members on layouts;

 — **Undo**,  — **Redo** — duplicate the respective items of the **Operations** menu;

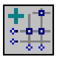
 — **Exit to project control screen** — switches to the **Project tree** window.

Operations Toolbar

The toolbar contains the following icons for invoking the commands which duplicate those of the **Operations** menu:

 — setting the grid lines parameters;

 — specification of the characteristics of concrete and selection of the reinforcement;

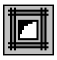
 — entry of nodes;

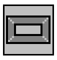
 — specification of the dimensions and position of columns;

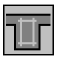
 — specification of the dimensions and position of beams;

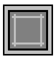
 — specification of the thickness and position of walls;

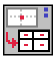
 — specification of the thickness and position of slabs;

 — specification of the dimensions and position of openings;

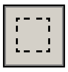
 — specification of the slab reinforcement parameters;

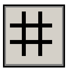
 — specification of the beam reinforcement parameters;

 — specification of the column reinforcement parameters;

 — dividing slabs by beams.

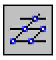
In the case of designing a flat floor slab, the following icons appear in the toolbar:

 — specification of the position and characteristics of additional reinforcement;

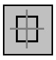
 — description of the joint reinforcement details.

Display Filters Toolbar

This toolbar contains the icons for invoking the following commands:

 — displays nodes;

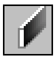
 — displays the node numbers;

 — displays columns;

 — displays the column numbers;

 — displays beams;

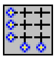
 — displays the beam numbers;

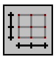
 — displays walls;

 — displays the wall numbers;

 — displays slabs;

 — displays the slab numbers;

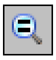
 — displays the grid lines;

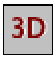
 — displays the dimension lines.

Visualization Toolbar

 — zooms in the layout;

 — zooms out the layout;

 — displays the layout in the initial scale;



— three-dimensional realistic visualization;

 — top view of the three-dimensional realistic visualization.

Arrangement of the Toolbar Strips

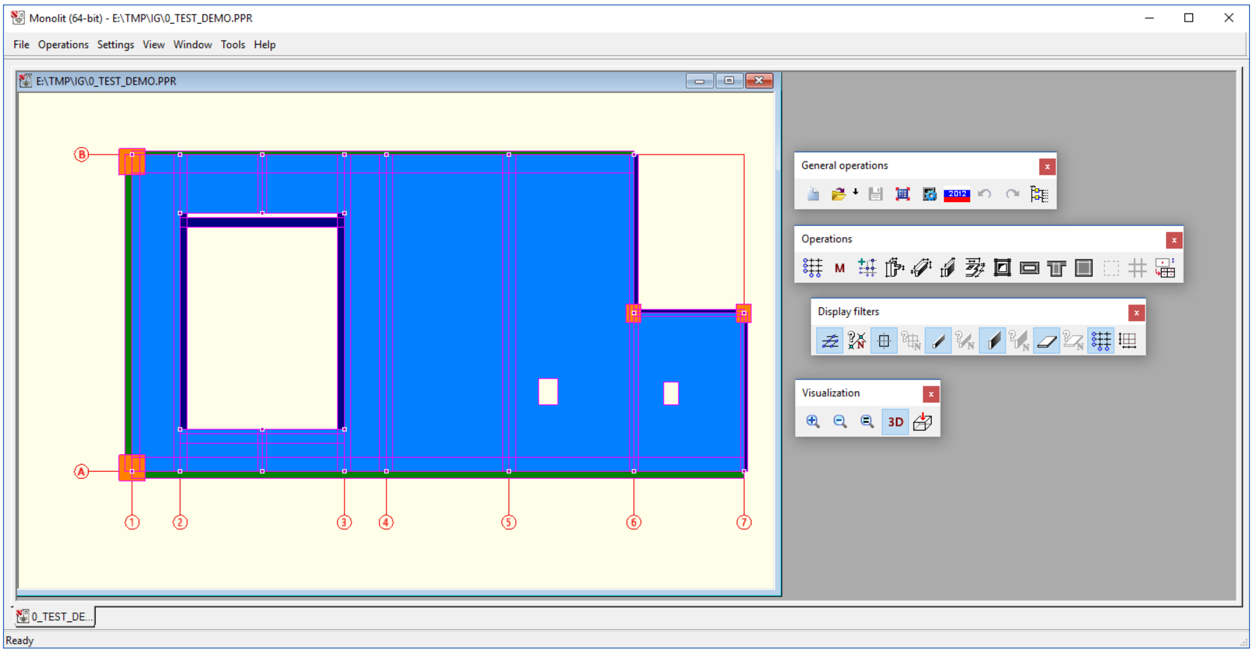


Figure 11.2.2-1. Arrangement of the toolbar strips

After the installation of the program the toolbar strips are located by default in the upper part of the window under the menu. If it is necessary for the convenience to change their position on the screen, they can be moved to a different place. To do it place the pointer over any point of the toolbar strip, save the button, click and hold the left mouse button and then drag the pointer and the strip to any place of the screen (even outside the program window). The toolbar strips can be returned to their initial position in the same way. When the working session is finished, the last arrangement of the toolbar strips is saved, and during the next loading of the program the strips will be arranged in the same way as in the end of the previous session (Fig. 11.2.2-1).

11.3 Creating a New Project

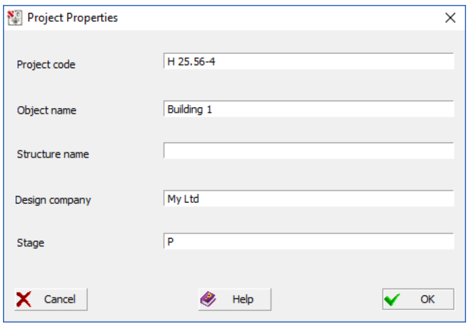


Figure 11.3-1. *The* **Project Properties** *dialog box*

To create a new project, place the pointer over the **New** button of the **General operations** toolbar strip and left-click. The screen will display the **Project Properties** dialog box (Fig. 11.3-1), where you can specify the information on the project, its code, the name of the design object, the name of the structure, the design company, and the design stage.

After clicking the **OK** button, the screen will display the **Save as…** standard window where you can specify the name of the new project file and allocate the folder where this file is going to be stored. The project is a file with the extension **.ppr**, containing information about the project and the design results.

Once the file creation process is completed, the main window field will display the window with the project tree (the title bar of this window contains the project file name).

11.3.1 Project Parameters

The **Project Parameters** dialog box (Fig. 11.3.1-1) has five checkboxes. The options they control enable to take into account the design features of the design object and include:

Structural reinforcement — enables to create a floor with the standard minimum reinforcement ratio. In this case, you do not have to specify the reinforcement of slabs, beams and columns. The calculated values of the reinforcement areas are saved, and you can look through them on the sheets of the reinforcement of slabs, beams and columns. You can correct these values if necessary and repeat the analysis;

Diameter of longitudinal rebar is not less than 12 mm — the checkbox is checked by default (for the main longitudinal reinforcement of a beam, it is recommended to use bars with a diameter of not less than 12 mm). If the user believes that in his case the rebar can have a smaller diameter, he has to uncheck this option;

Analysis and design with ribs upwards — this option enables to design a floor with the ribs directed upwards (and its flange facing downwards), i.e. the bottom surface of the beam aligns with the bottom of the floor slab;

Additional columns to the main stamp — displays the additional columns of the main stamp of drawings according to DSTU B A.2.4-4-99 (GOST 21.101-97);

Seismic load — this option is checked in those cases when the design has to take into account the design regulations for concrete-frame buildings erected in seismic regions.

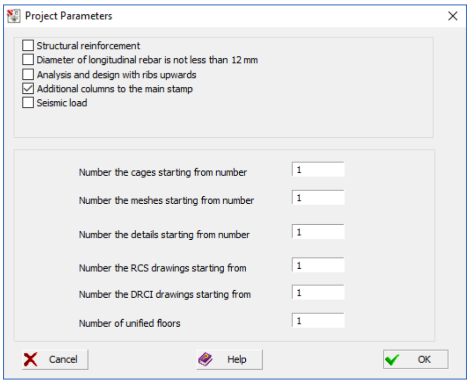


Figure 11.3.1-1. *The* **Project Parameters** *dialog box*

Besides, the **Project Parameters** dialog box has the text fields for assigning a start number when numbering the cages, meshes and details, as well as the RCS and DRCI drawings (1 by default), which enables to arrange continuous numbering within the project which comprises several floors.

In order to obtain the overall specifications for several identical floors, the user has to specify their number in the **Number of unified floors** text field. In this case the user will obtain the total number of items and total consumption of reinforcement and concrete for all indicated floors.

11.3.2 Project Tree

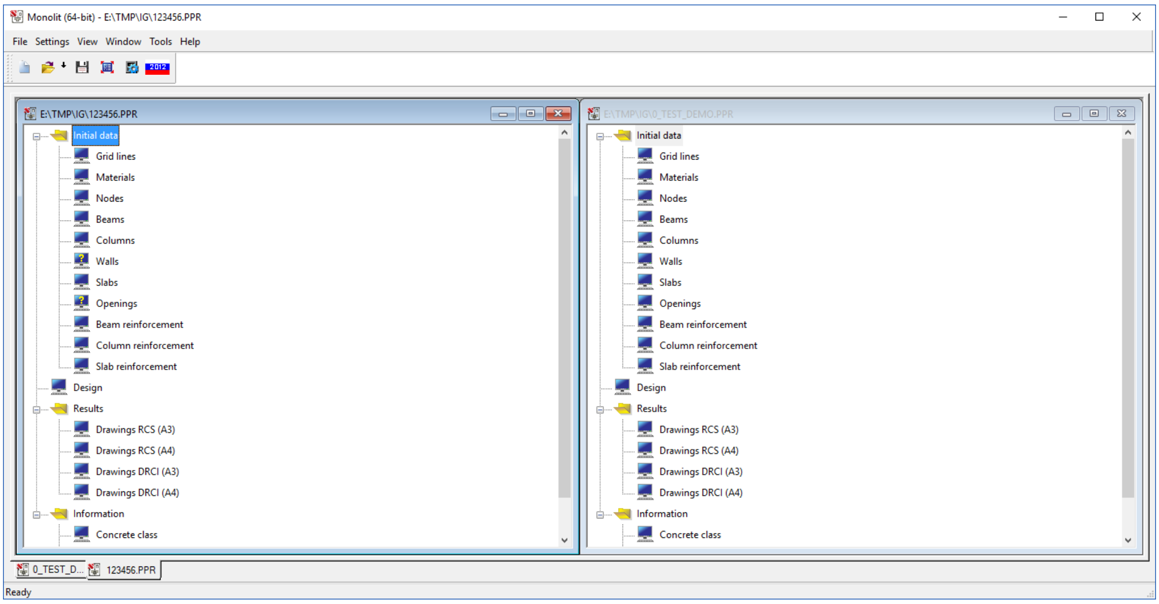


Figure 11.3.2-1. Project tree (shown on the left is the tree at the creation of a new project)

The project tree is displayed in the project window (Fig. 11.3.2-1) and has four levels: **Initial data**, **Design**, **Results**, and **Information**. The items of the first level correspond to the types of the initial data. Some of these data are mandatory, e.g. the grid lines and nodes, without specification of which, it is impossible to specify information on the structural members. Other data may be missing, for example, walls (if the frame comprises only columns), or columns (if the floors are supported only by walls).

The following legend is used in the project tree:

 — stop sign "Do not enter":

- in the **Initial data** item means that at the given moment this type of the initial data cannot be specified because the information required for its entry is missing (for example, nodes – for creating the structural members, or floors – for specifying the parameters of the openings);
- in the **Design** item means that the available data are not sufficient to perform the analysis;
- in the **Results** item means that the design (analysis) has not been performed;

 — the “question mark” in the **Initial data** item means that this type of information has not been specified.

 — this symbol in all items of the project tree means that the information is prepared or that the action can be performed.

To switch to any item of the tree, just place the pointer over its name and left-click.

11.3.3 Open an Existing Project

To open a previously created project, use the **Open** item in the **File** menu or in the toolbar. The computer performs actions accepted in the Windows environment when opening a document, namely: the **Open** dialog box appears where a folder is selected used for storage of the project and the project file name.

Once the dialog box is closed, the window with the project tree appears in the work area.

11.3.4 Data Transfer from the FORUM Preprocessor to Monolit

This operation enables to prepare the initial data for **Monolit** based on the information on the structural (enlarged) elements created in the **FORUM** preprocessor (Fig. 11.3.4-1). In the general case, if the parameters of the structural elements and their position in the model do not contradict the rules used in **Monolit**, a file with the .opr extension will be generated after performing this operation, which can be imported into this program (Fig. 11.3.4-2).

The initial data can include structural elements of slabs, columns, beams, and walls. All these elements must belong to one storey (module).

If the model created in **FORUM** consists of one module, and the elements included in this module form a ribbed floor slab, then the file of this project with the .opr extension can be loaded into **Monolit** without using the considered operation. All elements with the characteristics contradicting the **Monolit** parameters will not be imported into the resulting project.

If the original project in **FORUM** includes floors lying in different levels, consists of several modules, or only a part of the floor is transferred to **Monolit**, the following procedure is recommended to prepare the model for export:

Variant A.

Fragment the original model in such a way so that only the exported part remains in the window (if the entire module is exported, all other modules have to be deactivated);

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Visible fragment** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported model in the standard **Save As …** dialog box.

Variant B.

Select all elements you want to export in the design model;

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Selected elements** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported elements in the standard **Save As…** dialog box.

Variant C.

Select the floors you want to export;

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Selected floors and adjacent elements** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported elements in the standard **Save As …** dialog box.

The **Import** item of the **File** menu enables to load the project with the .opr extension to **Monolit**.

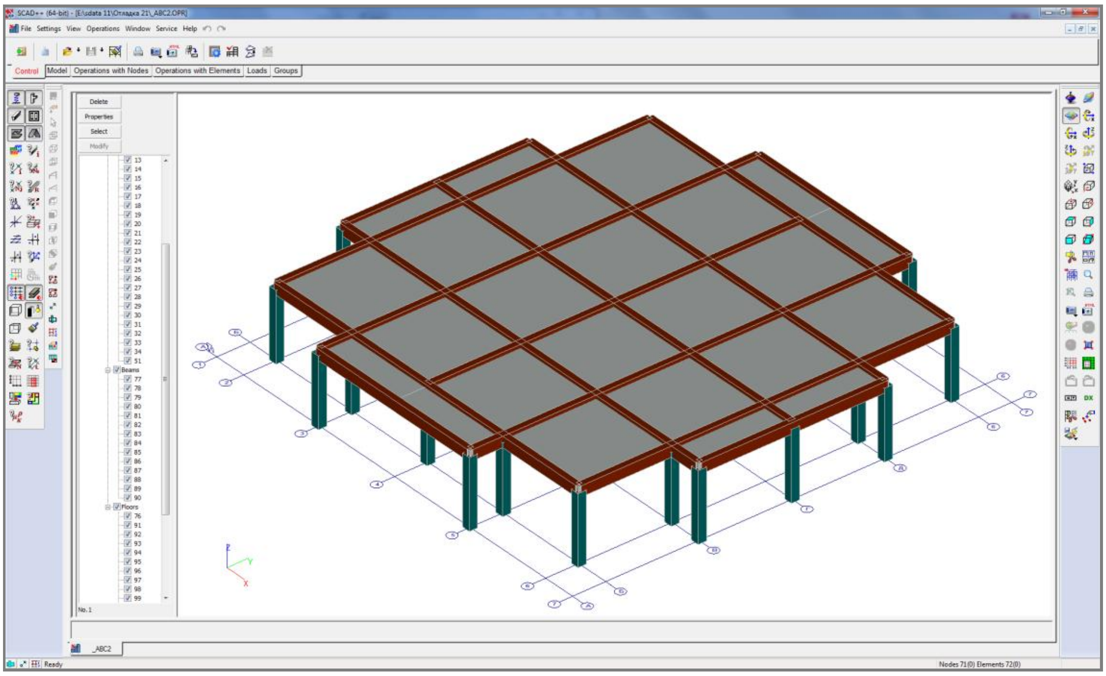


Figure 11.3.4-1. Model in the **FORUM** preprocessor

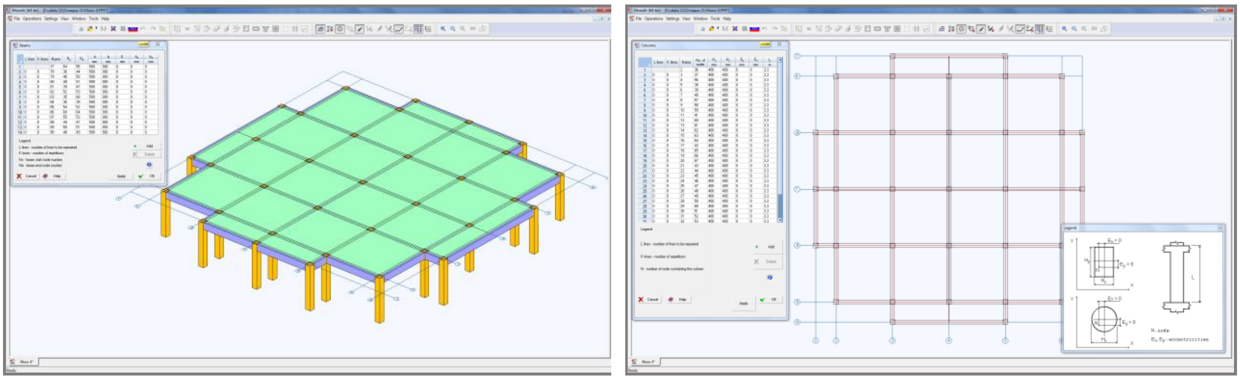
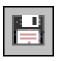


Figure 11.3.4-2. Model imported into **Monolit**

11.3.5 Save the Project



The **Save** and **Save as…** items are used to save a project. In the first case the project will be saved in a file under its current name. In the second case you have to specify the file name and/or its storage address. The **Save as…** operation is performed in the respective standard dialog box.

11.3.6 Arrangement of Windows in a Multi-Window Environment

Since the multiple document interface (MDI) enables to open several projects simultaneously, let's consider the arrangement of windows in a multi-window environment.

The size of the window for each of the projects is set by the user. The maximum size corresponds to the current size of the work area of the main window. If the windows of several projects are open on the screen at the same time, only one of those windows can be active. In order to make a particular window active, you have to point the cursor on its tab (in the bottom left corner of the main window) or on any visible part of this window. As a result, not only this window will become active but also the project this window belongs to.

Hereinafter, the notion "point the cursor on" shall mean the sequence of actions when the pointer is moved over to the object of interest and then the left mouse button is clicked.

Standard Windows operations of the arrangement of several windows on a screen can be used in the program. These operations can be invoked from the **Window** menu. They enable to arrange the windows according to the following rules:

Cascade — windows are arranged one under another in such a way that one active window is fully visible on the screen, whereas only the titles of the windows of other projects can be seen (Fig. 11.3.6-1);

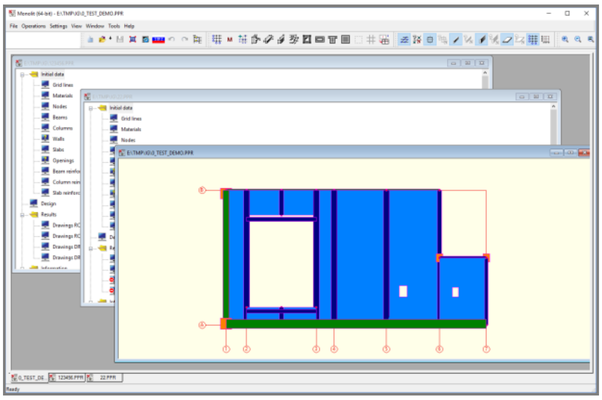


Figure 11.3.6-1. **Cascade** arrangement

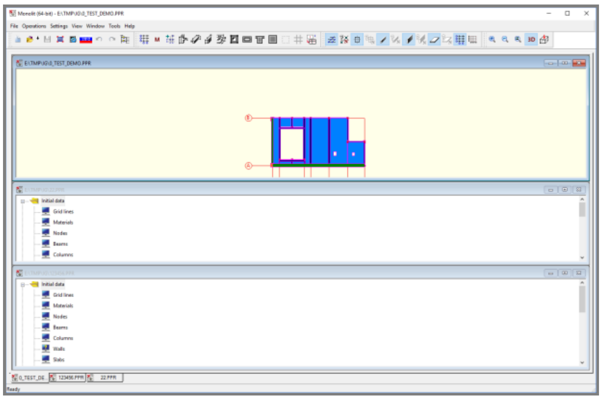


Figure 11.3.6-2. **Tile** arrangement

Tile — all windows open on the screen are fully visible, and their size depends on the number of windows and the size of the main window (Fig. 11.3.6-2).

The same menu can be used to invoke the **Arrange Icons** operation, which is typical for MDI and is used in those cases when one or several windows are minimized. When this operation is invoked the icons of the minimized windows are relocated to the bottom left corner of the work area of the main window.

11.3.7 Preparation of the Initial Data

In order to prepare the initial data for a new project, you have to create the project (see the **Creating a New Project** section) and activate one of the items of the **Initial data** level in the project tree. The area of the main window toolbar will include a set of bars used for the preparation of data.

If this is done by pointing on the specific type of the initial data in the tree (for a new problem those are, as a rule, the **Grid lines** or **Nodes** items), then two windows will be opened – the dialog box, where the initial data of the selected item are specified, and the window in the work area which displays the current state of the structural layout. For example, if the **Grid lines** item has been activated in the project tree, the **Grid Lines** dialog box and the window displaying the structural layout of the current problem will be opened (Fig. 11.3.7-1).

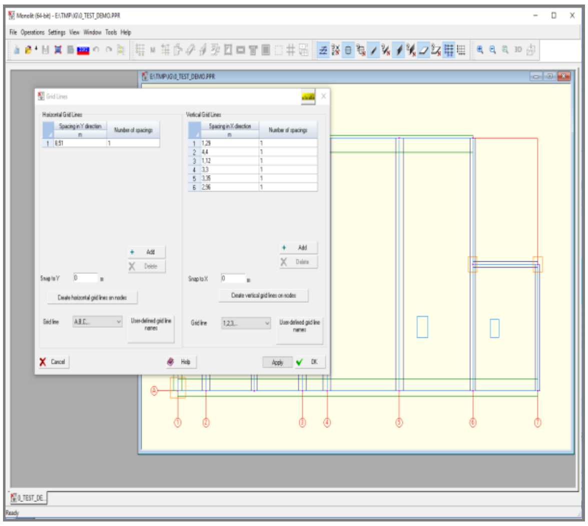


Figure 11.3.7-1. Preparation of the grid lines data

Data preparation can be activated by pointing directly at the level title (**Initial data**) in the project tree. In this case only the window displaying the structural layout will be open, while the dialog box, where the data preparation is performed, will appear on the screen after selection of the type of data in the toolbar or in the **Operations** menu.

11.3.8 Working with Tables

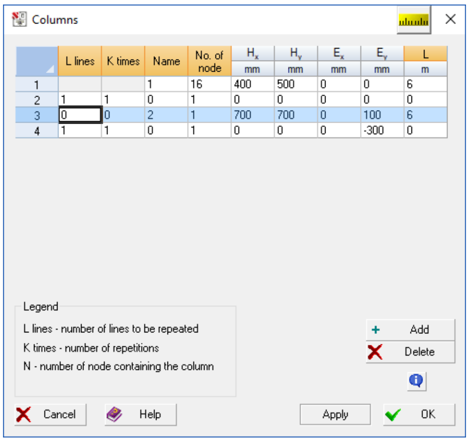


Figure 11.3.8-1. Table for entering the column characteristics

Universal tables (Fig. 11.3.8-1) are used for entering the information in the tabular form. When entering numerical data, the following rules must be observed:

- the fractional and the integer parts of a number must be separated by a period or another separator assigned by the user during the customization of the operating system (see **Settings | Regional Settings | Number**);
- to finish entering a number and switch to the next cell of the table press the **Tab** key on your keyboard;
- the information entered into the table can be stored in the **clipboard**; to do this, you have to highlight the lines with the information to be stored and press the **Ctrl+Ins** keys;
- information can be transferred to the table from the **clipboard**; to do this you have to place the pointer over the cell, starting from which the information is going to be entered, and press the **Shift+Ins** keys;
- lines in the table can be highlighted by pointing the cursor on the number of the line in the first column of the table (hereinafter "pointing the cursor on" means placing the pointer over a control and left-clicking);
- to delete one or several lines (a block) you have to highlight them and click the **Delete** button in the dialog box;
- to add new lines in the end of the table click the **Add** button.

11.3.9 Operations with Highlighted Table Lines

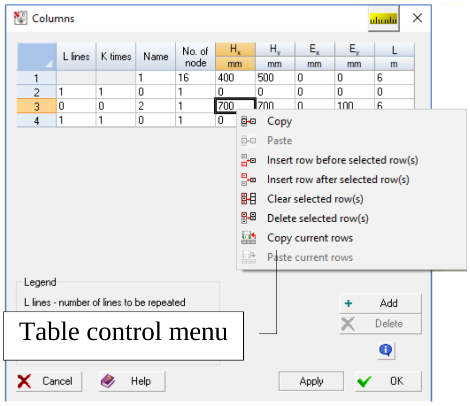


Figure 11.3.9-1. Table control menu

To work with the highlighted lines or cells of a table, use the drop-down menu (Fig. 11.3.9-1), which appears on the screen after right clicking (provided the pointer is placed within the table). The program provides a set of operations with the table objects described in Section 11.3.8 **Working with Tables**.

11.3.10 Repeaters

A specific feature of working with tables in this program is a capacity to repeat any information with a certain step (this applies to the data specified in the following dialog boxes: **Nodes, Columns, Beams, Walls, Slabs, Openings**). First two columns of the table serve for this purpose. The first column (**L lines**) shows the number of lines preceding the current one which have to be repeated, and the second column (**K times**) indicates the number of times they have to be repeated. Other columns of this line are used to specify their steps, which may be different for every item both in value and in sign.

It should be taken into account that the table contents are stored in an expanded form. Therefore, the number of repeated lines (L lines) must be equal to their actual number (with the account of the earlier specified repeaters), and not to the number indicated in the table. The table below shows that, according to line 2, the first line will be repeated five times, and the columns will be snapped to nodes 1 through 6. As a result, the expanded form of the entry will include six lines. Execution of the repeat operation indicated in the third line will result in one repetition of the first six lines of the expanded entry with the incremental step of six regarding the number of the snap node (all in all, there will be twelve columns snapped to the nodes 1 through 6 and 7 through 12).

	L lines	**K times**	**Name**	**No. of node**	**H_x**	**H_y**	**E_x**	**E_y**	**L**
1			1	1	1200	350	600	150	3.3
2	1	5	0	1	0	0	0	0	0
3	6	1	0	6	0	0	0	0	0

The **Apply** button is typical for the dialog boxes used for entering the tabular data. Clicking this button activates the data entry operation, the data control and, at the same time, display of the graphical information in the project window.

11.3.11 Grid Lines

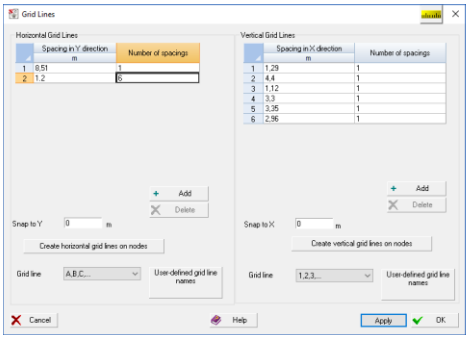


Figure 11.3.11-1. *The* **Grid Lines** *dialog box*

The grid lines can be entered before or after specification of the nodes. In the first case, the grid lines will serve as the basis for creating the grid of nodes which can be **automatically** plotted at the grid line intersections. In the second case, the grid lines will be plotted on the basis of all specified nodes, and each node will be considered as the intersection point of the grid lines.

Setting of grid lines is performed in the respective dialog box (Fig. 11.3.11-1). This dialog box contains two tables: **Horizontal Grid Lines** and **Vertical Grid Lines**. The first table is used to specify the spacing between the grid lines along the Y axis and the number of the repeated spacings. If the grid lines are snapped to the point with the coordinates along the Y axis other than zero, the value of displacement is specified in the **Snap to Y** text field. The vertical grid lines along the X axis are described in a similar way.

If the names of the grid lines differ from the default ones (letter names in the alphabetical order for the horizontal grid lines, and digital notations in the ascending order for the vertical grid lines), you can use the **User-defined grid line names** button.

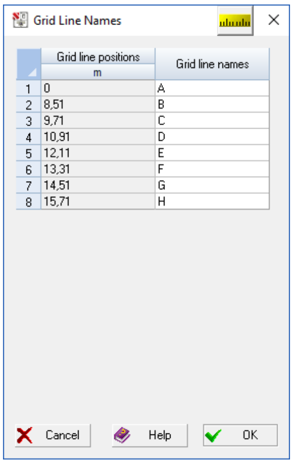
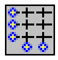
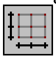


Figure 11.3.11-2. *The* **Grid Line Names** *dialog box*

In the **Grid Line Names** dialog box (Fig. 11.3.11-2) change the program-specified names to different ones (up to three characters).

The grid lines are displayed in the project window after clicking the **Apply** button.

If the grid lines are entered using the earlier specified nodes, they will be automatically generated after clicking the buttons **Create horizontal grid lines on nodes** and **Create vertical grid lines on nodes**.

The grid lines and dimension lines are displayed in the project window after clicking the filter button  — **Display the grid lines** and button  — **Display the dimension lines** respectively.

11.3.12 Characteristics of Materials

Characteristics of the used materials are specified in the **Materials** dialog box (Fig. 11.3.12-1). The types of materials (concrete and reinforcement) which will be used when designing the floor are specified here.

The class of concrete, which is the same for the whole floor, is selected from the respective list. For concrete, you also have to select the type of concrete, the concrete mix, the workability grade, the frost resistance grade and the impermeability grade, and (for light-weight concrete) the grade by average density.

For each class of reinforcement used for the floor, a set of permissible diameters can be specified. To do it you have to select the class name in the **Reinforcement class** list. The list of all diameters of the reinforcement of the specified class will be then given in the **Assortment** list. This list will be also given in the **Selected** list. If no data were removed from the **Selected** list, this means that when designing the floor you can use the entire assortment of the given class. Otherwise, only the diameters remaining in the list will be used.

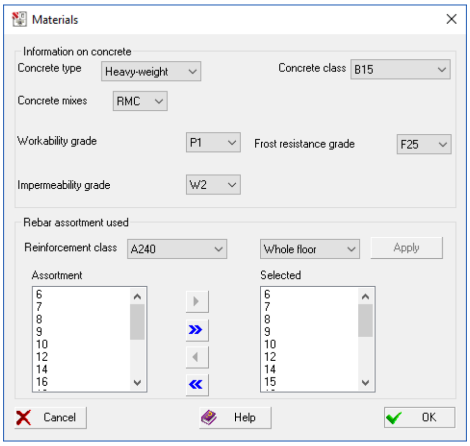


Figure 11.3.12-1. *The* **Materials** *dialog box*

Two buttons are used to transfer the data from the left list to the right one. The > button enables to transfer only the items highlighted in the left list to the right one. The >> button transfers all the contents of the left list to the right one.

Correction of the right list items is performed in a similar way. The < button is used to remove only the highlighted items from the list, whereas the << button is used to clear the right list completely. After making changes in the selected reinforcement class, it is necessary to apply them by clicking the **Apply** button.

If it is necessary to introduce certain limitations of the assortment for different structural members (beams, slabs, columns), you have to select the name of the structural member from the right drop-down list of the **Rebar assortment used** group, specify the reinforcement class and list of permissible bar diameters, and then click the **Apply** button. If the specified data apply to the entire floor, you have to select the **Whole floor** from the right list (selected by default).

11.3.13 Nodes

The floor geometry is specified on the orthogonal grid of nodes. The nodes of the grid are created at the joints of the structural members: beams, walls, columns, and slabs. The position of the nodes is described in the table (Fig. 11.3.13-1) with two coordinates **X**, **Y** of the floor plane. Each table line has five columns.

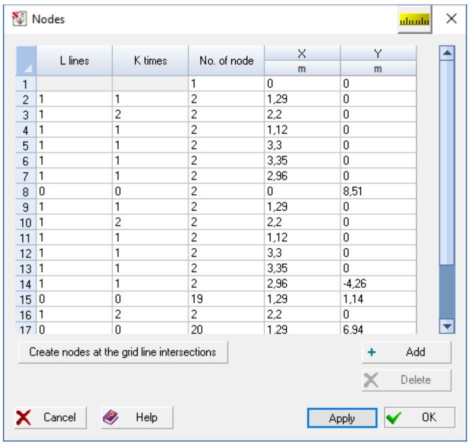


Figure 11.3.13-1. *The* **Nodes** *dialog box*

The *first two columns* are used when working with the repeaters.

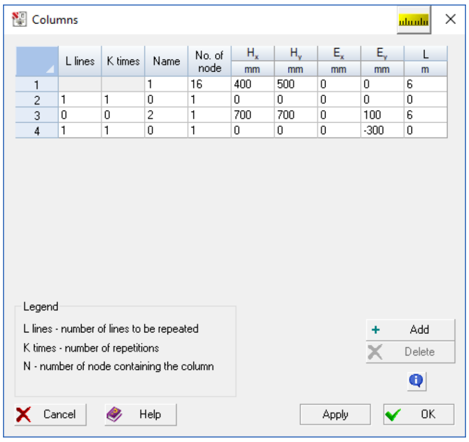
The *third column* is used for specifying the numbers of the layout nodes (**No. of node**). Nodes must not have identical numbers. You should be careful about it, especially when using the repeaters.

The *fourth and fifth columns* are used to specify the node coordinates in meters (**Xm** and **Ym**, respectively).

The **Apply** button can be used at any step of the data preparation process, which makes it possible to perform the graphical control of the specified information in the project window.

If the nodes are created on the earlier specified grid lines, they will be automatically generated after clicking the **Create nodes at the grid line intersections** button.

11.3.14 Columns



	L lines	K times	Name	No. of node	H_x mm	H_y mm	E_x mm	E_y mm	L m
1			1	16	400	500	0	0	6
2	1	1	0	1	0	0	0	0	0
3	0	0	2	1	700	700	0	100	6
4	1	1	0	1	0	0	0	-300	0

Figure 11.3.14-1. *The* **Columns** *dialog box*

The **Columns** dialog box (Fig. 11.3.14-1) is used to describe the cross-sections and arrangement of columns supporting beams and slabs. The data are entered into nine columns of the table.

The *first two columns* are used to describe the repeaters.

The *third column* (**Name**) is used to specify the column name – a number which will be added to the letters «Cm» in the drawing (e.g., if the number 7 is entered, the column name will be Cm7). Any numbers can be used, not necessarily coming in succession and in the ascending order.

The *fourth column* is used to specify the number of the node (**No. of node**) in which the column is located.

The *fifth* (**Hx**) and *sixth* (**Hy**) *columns* are used to specify the dimensions of the column cross-section in mm in the X and Y axes directions, respectively. For the columns of circular section, the diameter is specified instead of **Hx**; in this case, **Hy** must be equal to zero.

The *seventh* and *eighth columns* are used to specify the **Ex** and **Ey** eccentricities of the column location with respect to the snap node (in mm).

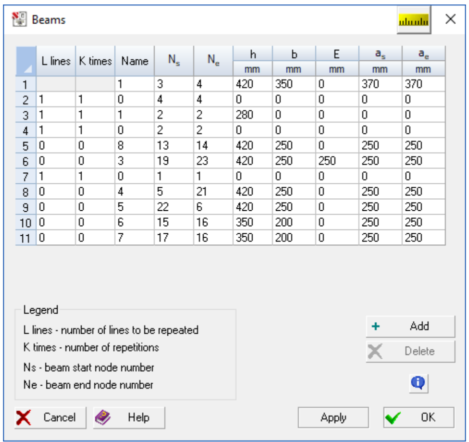
The *ninth column* (**L**) is used to specify the column length in meters. The column length is the distance from the column base to the top of the designed floor.

The rules for specifying the eccentricities and length of a column are given in the information window, which can be invoked by clicking the button.

Clicking the **Apply** button will display the columns and their names in the project window

11.3.15 Beams

The **Beams** dialog box (Fig. 11.3.15-1) is used to describe the beams that form the floor. Beams can be only placed along the coordinate axes. The program implements the concept of a *unified group of beams.* Falling under the same group are the beams that have the following coincident characteristics:



	L lines	K times	Name	N_s	N_e	h mm	b mm	E mm	a_s mm	a_e mm
1			1	3	4	420	350	0	370	370
2	1	1	0	4	4	0	0	0	0	0
3	1	1	1	2	2	280	0	0	0	0
4	1	1	0	2	2	0	0	0	0	0
5	0	0	8	13	14	420	250	0	250	250
6	0	0	3	19	23	420	250	250	250	250
7	1	1	0	1	1	0	0	0	0	0
8	0	0	4	5	21	420	250	0	250	250
9	0	0	5	22	6	420	250	0	250	250
10	0	0	6	15	16	350	200	0	250	250
11	0	0	7	17	16	350	200	0	250	250

Figure 11.3.15-1. *The* **Beams** *dialog box*

- cross-section dimensions;
- length (taking into account bearing on the brick wall);
- number of supporting nodes;
- distance between the respective supporting nodes;
- reinforcement.

All beams of a group must have one (original) name.

The description table consists of ten columns.

The *first two columns* are used to describe the repeaters.

The *third column* (**Name**) is used to specify the beam name – a number which will be added to the letters “Bm” in the drawing (e.g., if the number 12 is entered, the beam name will be Bm12).

The *fourth column* is used to specify the number of the node (**N_s**) in which the beam begins, and the *fifth column* – to specify the number of the node (**N_e**) in which the beam ends. It is advisable to describe beams in the ascending order of the coordinates of the beam start and end nodes.

In the *sixth column* you have to specify the height of the beam section **h** (in mm) (the height of a beam is the distance from the bottom surface of the beam to its top surface that aligns with the top of the floor slab), and in the *seventh column* you have to specify its width **b** (in mm).

The *eighth column* (**E**) is used to specify the eccentricity (offset) of the beam location with respect to the nearest grid line, which passes through the extreme beam nodes. The rule for specifying the offsets is given in the information window, which can be invoked by clicking the button.

If the beginning of the beam bears on a brick wall, it is necessary to specify the embedment length (in mm) in the *ninth column* ($\mathbf{a_s}$).

If the end of the beam bears on a brick wall, it is necessary to specify the embedment length (in mm) in the *tenth column* ($\mathbf{a_e}$).

If two ends of the beam bear on brick walls and the embedment length is the same at both ends, you may specify just $\mathbf{a_s}$. If the embedment length is not specified, the values $\mathbf{a_s}$ and $\mathbf{a_e}$ will be taken according to [3], and the respective warning will appear. If the beam does not bear on a brick wall, these columns remain blank.

Clicking the **Apply** button will display the arrangement of beams on the floor layout in the project window.

11.3.16 Walls

The **Walls** dialog box (Fig. 11.3.16-1) is used to describe the walls that support the floors.

The walls are described according to the following rules:

The *first two columns* are used to describe the repeaters.

The *third column* (**Name**) is used to specify the wall name – a number which will be added to the letter «W» (if the wall is a brick one) or to the letters «Wm» (in case of a monolithic wall) in the drawing (e.g., if the number 5 is entered, the wall name will be W5 for a brick wall, and Wm5 for a monolithic wall).

The *fourth column* is used to specify the number of the node $\mathbf{N_s}$ in which the wall begins, and the *fifth column* – to specify the number of the node $\mathbf{N_e}$ in which the wall ends. It is advisable to describe walls in the ascending order of the coordinates of the nodes where the wall begins and ends.

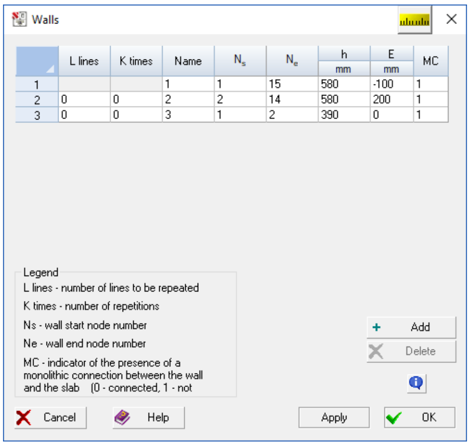


Figure 11.3.16-1. *The* **Walls** *dialog box*

In the *sixth column* you have to specify the wall thickness **h** (in mm).

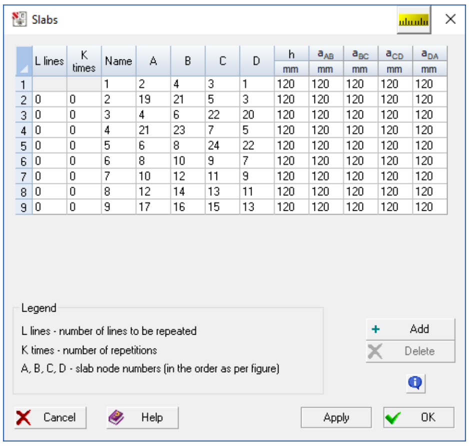
The *seventh column* is used to specify the wall eccentricity (offset) **E** with respect to the nearest grid line, which passes through the extreme wall nodes (in mm). The rule for specifying the offsets is given in the information window, which can be invoked by clicking the button.

In the *eighth column* it is necessary to specify the indicator of the presence of a monolithic connection between the wall and the slab – **MC.** In case of a reinforced concrete wall, MC = 0, which means that there is monolithic connection between the wall and the slab. In case of a brick wall, MC = 1, i.e. there is no monolithic connection between the wall and the slab.

Clicking the **Apply** button will display the arrangement of walls on the floor layout in the project window.

11.3.17 Slabs

The **Slabs** dialog box (Fig. 11.3.17-1) is used to specify the information on the floor slabs. In the case of a ribbed floor, the segments enclosed from all sides by beams or walls (including the case when the slab edges are not fixed – cantilever overhangs) are considered here.



	L lines	K times	Name	A	B	C	D	h mm	a_{AB} mm	a_{BC} mm	a_{CD} mm	a_{DA} mm
1			1	2	4	3	1	120	120	120	120	120
2	0	0	2	19	21	5	3	120	120	120	120	120
3	0	0	3	4	6	22	20	120	120	120	120	120
4	0	0	4	21	23	7	5	120	120	120	120	120
5	0	0	5	6	8	24	22	120	120	120	120	120
6	0	0	6	8	10	9	7	120	120	120	120	120
7	0	0	7	10	12	11	9	120	120	120	120	120
8	0	0	8	12	14	13	11	120	120	120	120	120
9	0	0	9	17	16	15	13	120	120	120	120	120

Figure 11.3.17-1. *The* **Slabs** *dialog box*

Slabs can only have rectangular shape. The program implements the concept of a **unified group of slabs**. Falling under the same group are the slabs that have the following coincident characteristics:

- dimensions (taking into account bearing on the brick wall);
- pattern of the support along four sides;
- reinforcement.

All slabs of the given group must have one (original) name.

The *first two columns* are used to describe the repeaters.

The *third column* (**Name**) is used to specify the slab segment name – a number which will be added to the letters «Sm» in the drawing (e.g., if the number 8 is entered, the slab name will be Sm8).

The *columns four through seven* are used to specify the numbers of the nodes describing the slab in the nodes **A**, **B**, **C**, **D** sequence order, as shown in Fig. 11.3.17-2. Moreover, the rule for specifying slabs is given in the information window, which can be invoked by clicking the button.

The *eighth column* is used to specify the slab thickness **h** (in mm).

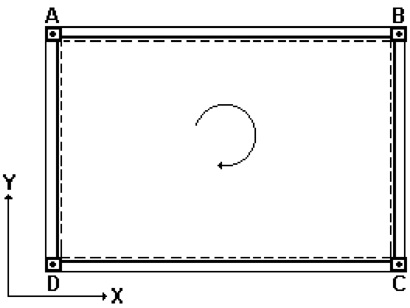


Figure 11.3.17-2. Slab description rules

The *columns nine through twelve* are used to specify the embedment length of the respective slab side into the brick wall **aAB, aBC, aCD, aDA** (in mm). These columns are filled only if such embedment takes place.

If two and more slab sides bear on brick walls and the embedment length is the same, you can specify the information only for the first supported side. If the embedment length is not specified, its value will be taken according to [3]. In the cases when a number of slabs have the same parameters but bear on the walls with different sides, you can specify the embedment length for all sides of each slab. This will enable to include all slabs in one unification group.

Clicking the **Apply** button will display the specified slabs in the project window.

11.3.18 Openings

This dialog box (Fig. 11.3.18-1) is used to specify the information on the openings (holes) in the floor slabs. Openings can only be of rectangular or round shape. If you specify the name of the unified group of slabs, all the slabs included in this group will have openings.

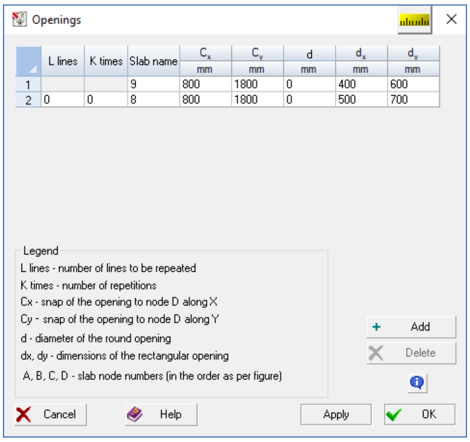


Figure 11.3.18-1. *The* **Openings** *dialog box*

The *first two columns* are used to describe the repeaters.

The *third column* is used to specify the name of the slab with the opening. The **Slab name** is a number which is added to the letters «Sm» (see the **Slabs** dialog box).

The *fourth column* is used to specify the snap of the opening center along the X axis (**C_x**) to the left bottom slab node (node **D**) in mm.

The *fifth column* is used to specify the snap of the opening center along the Y axis (**C_y**) to the left bottom slab node (node **D**) in mm.

The *sixth column* is used to specify the opening diameter (**d**) in mm if the opening is round (otherwise it is zero).

The *seventh and eighth columns* are used to specify the dimensions of a rectangular opening (**dx** and **dy**) in mm (zero – for round openings).

The rule for specifying openings is given in the information window, which can be invoked by clicking the button.

The snap of an opening to the nearest node of the slab "in the clear" is given in the output documents. The drawings of openings are given separately (two openings per one sheet in the order of their description) with the encasing bars and their dimensions.

Notes:

If the floor layout contains openings encased with reinforced ribs, they do not have to be specially entered in the **Openings** document. To describe them it suffices to specify the encasing ribs as beams (**Beams** dialog box), considering that there is no floor segment between these beams (ribs).

It is recommended to specify only one opening for one slab.

11.3.19 Slab Reinforcement (Ribbed Floors)

The **Slab Reinforcement** dialog box (Fig. 11.3.19-1) is used to specify the information on slab reinforcement, which is required for floor designing and consists of two parts. The first part contains the information related to the entire floor: class and standard of the primary and secondary reinforcement, the values of the concrete covers, and maximum width of the welded meshes. The second part contains the data on the area of the upper and lower longitudinal reinforcement along the X and Y axes and has to be specified separately. These data are entered in the table with eight columns.

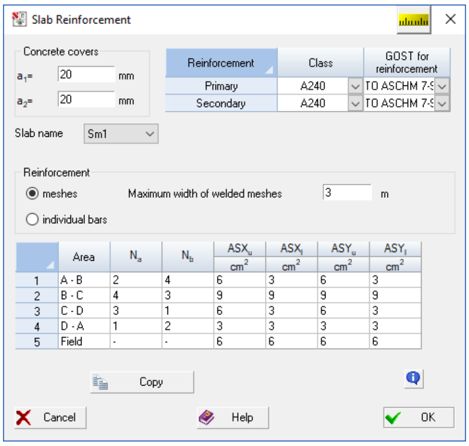


Figure 11.3.19-1. *The* **Slab Reinforcement** *dialog box*

The *first four columns* are filled automatically on the basis of the information specified in the **Slabs** dialog box.

The *first column* contains the name of the slab for which the data are specified.

The *second column* (**Area**) contains the designations of the sides (edges) of a slab, which are supported on a beam or a wall, or are simply supported (for example, **A-B)**, as well as **Field** – the central area of the slab (Fig. 11.3.19-2). The width of the slab boundary areas is taken as 0.25 of the slab size in the respective direction. The **Field** is the remaining central part of the slab space.

The *third and fourth columns* contain the numbers of the slab nodes adjacent to the edge.

The next *four columns* are filled with the values of running area of the reinforcement (cm^2/m) — **ASX$_u$** and **ASX$_l$,** the upper and the lower one, respectively, along the X axis; **ASY$_U$** and **ASY$_L$ –** the upper and the lower one, respectively, along the Y axis (Fig. 11.3.19-3). If the information on the reinforcement area was obtained as a result of the analysis performed by **SCAD**, the relation $ASX_U \Rightarrow AS2$, $ASX_L \Rightarrow AS1$, $ASY_U \Rightarrow AS4$ and $ASY_L \Rightarrow AS3$ holds true. We can determine the maximum values of the areas in the directions along the considered side and perpendicular to this side (at the top and bottom of the slab) for the finite elements located in the strip adjacent to the considered slab side.

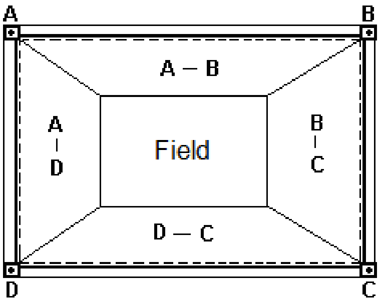


Figure 11.3.19-2. Areas of the slab reinforcement description

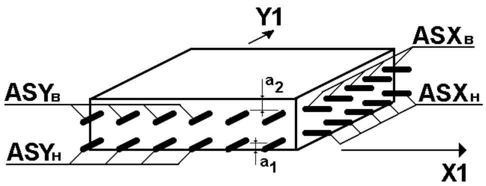


Figure 11.3.19-3. Reinforcement arrangement in a slab

It is recommended to stick to the following order of data preparation in this dialog box:

- specify the class of the primary and secondary reinforcement, as well as the standard for the selected reinforcement assortment;
- specify the values of the concrete covers;
- specify the maximum width of welded meshes in case of the mesh reinforcement, or the maximum bar length if the slab is reinforced by individual bars;
- from the list of slabs select the slab for which the data will be specified (the selected slab will be highlighted on the layout in the project window);
- fill the table.

Once the table is filled select the next slab in the list and fill only the table for it, since the rest of the specified information applies to all floor slabs.

In those cases when the characteristics of slab reinforcement can be taken either entirely or partially from the characteristics specified earlier for another slab, you can use the **Copy** button. The number (name) of the slab the reinforcement characteristics of which you want to copy for the current slab is selected in the **Copy** dialog box.

The rule for specifying the slab reinforcement is given in the information window, which can be invoked by clicking the button.

In the **Reinforcement** group of the dialog box you can specify whether the slab is reinforced by meshes or by individual bars (in the latter case, you have to specify the maximum bar length).

It should be noted that the **Slab name** list will include only original names, i.e. there will be no repetitions of identical names. If there are unified groups in the project, the list will contain only the name of the first slab of the group with a list of its corresponding nodes.

The drawings with the results of the design, as well as other project documents for each group of slabs (**Meshes**, **Steel consumption**, **Per-slab specification**, **Overall specification**) will include the information on all slabs of the groups.

11.3.20 Beam Reinforcement

This dialog box (Fig. 11.3.20-1) is used to specify the information on beam reinforcement, which is required for designing. This information includes the data on the areas of the upper and lower longitudinal, as well as transverse reinforcement, class and standards of the reinforcement, and the value of the concrete cover.

Beams are divided into two types: **Main** and **Secondary**. All nodes of a *secondary beam* can be **supporting.** A beam can be connected to a column, a wall or another beam by these nodes. If a beam supports other beams, it is considered to be the *main* beam, and some of its nodes may be non-supporting ones. For example, those can be the nodes the secondary beams rest on.

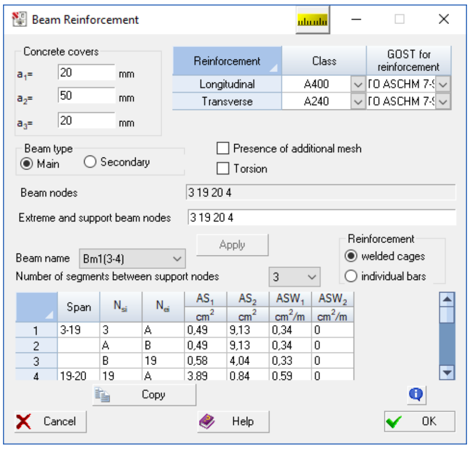


Figure 11.3.20-1. *The* **Beam Reinforcement** *dialog box*

The beam type is specified using the radio buttons. The lists of all nodes the given beam passes through are displayed in the **Beam nodes** and **Extreme and support beam nodes** text fields. It is necessary to correct the list in the second field (*Extreme and support beam nodes*), removing the numbers of non-supporting nodes from it. This means that the reinforcement area in a beam is specified only in the segments between the supporting nodes.

Data on the reinforcement areas are entered in the table that contains information only on one beam. The information is divided into sections according to the number of spans between the supporting nodes of the beam. **Span** is a part of a beam between the supporting nodes or between one supporting node and the extreme unfixed node (in case of a cantilever). The reinforcement area is specified for the given number of segments between the supporting nodes of the span (this value is entered in the respective text field), and in the case of an unfixed node – between this unfixed node and the supporting node.

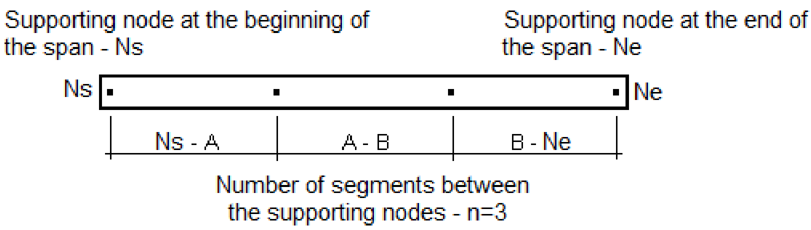


Figure 11.3.20-2. Example of description of a span divided into three segments

Once you have specified the beam type, the number of segments between supporting nodes, and the list of supporting nodes, click the **Apply** button. Then the number and length of segments in the table will be corrected in accordance with the specified number of spans and segments.

The first three columns of the table are filled automatically and include the following data:

the first column – numbers of the nodes at the beginning and in the end of the span (**N_s** and **N_e**, respectively);

the second and third columns – identifiers of the segment in the span (**N_{si}**, **N_{ei}**), for example if the beam span between the nodes 33 and 34 is divided into three segments, the segments will have the following designations: 33-A, A-B, B-34;

the fourth column – the area of the lower longitudinal reinforcement in the segment — **AS_1** (cm^2);

the fifth column – the area of the upper longitudinal reinforcement in the segment — **AS_2** (cm^2);

the sixth column – the area of the vertical transverse reinforcement in the segment — **ASW_1** (cm^2/m);

the seventh column – the area of the horizontal transverse reinforcement in the segment — **ASW_2** (cm^2/m). It is specified if the action of shear force on the beam has to be taken into account. If this column is blank, structural reinforcement connecting the cages is installed.

It is recommended to stick to the following order of data preparation in this dialog box:

- select the beam from the **Beam name** list you want to specify the data for (the selected beam will be highlighted in the layout);
- assign the class of longitudinal and transverse reinforcement, as well as the standard for the selected reinforcement assortment;
- specify the values of the concrete covers;
- select the beam type and correct the list of supporting nodes for the main beam;
- specify the number of segments between the supporting nodes;
- click the **Apply** button to correct the description of sections in the table;
- fill the table.

Once the table is filled select the next beam in the list and repeat the above steps.

In those cases when the characteristics of beam reinforcement can be taken either entirely or partially from the characteristics specified earlier for another beam, you can use the **Copy** button. The number (name) of the beam the reinforcement characteristics of which you want to copy for the current beam is selected in the **Copy** dialog box.

The rules for specifying the beam reinforcement are given in the information window, which can be invoked by clicking the button.

In the **Reinforcement** group of the dialog box you can specify whether the beam is reinforced by welded cages or by individual bars (without bends).

It should be noted that the **Beam name** list will include only original names, i.e. there will be no repetitions of identical names. If there are unified groups of beams in the project, the list will contain only the name of the first beam of the group with a list of its corresponding nodes.

The drawings with the results of the design are generated only for the first beam of each group. All other project materials (**Cages**, **Steel consumption**, **Overall specification**) will include the information on all beams of the respective groups.

If the beam reinforcement was selected with the account of torsion, you have to check the **Torsion** checkbox. In this case, the respective recommendations both for the tied reinforcement and welded cages will be taken into account in the designing process.

If there is a possibility of fire action on the floor, beams are provided with additional reinforcement in the form of a reinforcing mesh for the installation of which it is necessary to check the **Presence of additional mesh** checkbox.

Example

The floor structural layout is given in Fig. 11.3.20-3, where the Bm1 beam is assigned the *Main* type. The beam is connected to the Cm1 columns in nodes 1, 3, 4 and 6, which are considered to be the supporting nodes. The secondary beams Bm4 and Bm5 bear on the Bm1 beam in nodes 2 and 5.

When preparing the data in the **Beam Reinforcement** dialog box, after selecting the Bm1 beam and specifying its type (*Main*), it is necessary to correct the list of supporting nodes in the **Beam name** list by removing the nodes 2 and 5 from it. This done, you have to click the **Apply** button, after which the reinforcement table will display three segments: 1-3, 3-4 and 4-6, where the reinforcement area values are specified. Similar operations have to be performed to describe the Bm3 main beam, in which case the node 14 has to be removed from the list of supporting nodes of this beam.

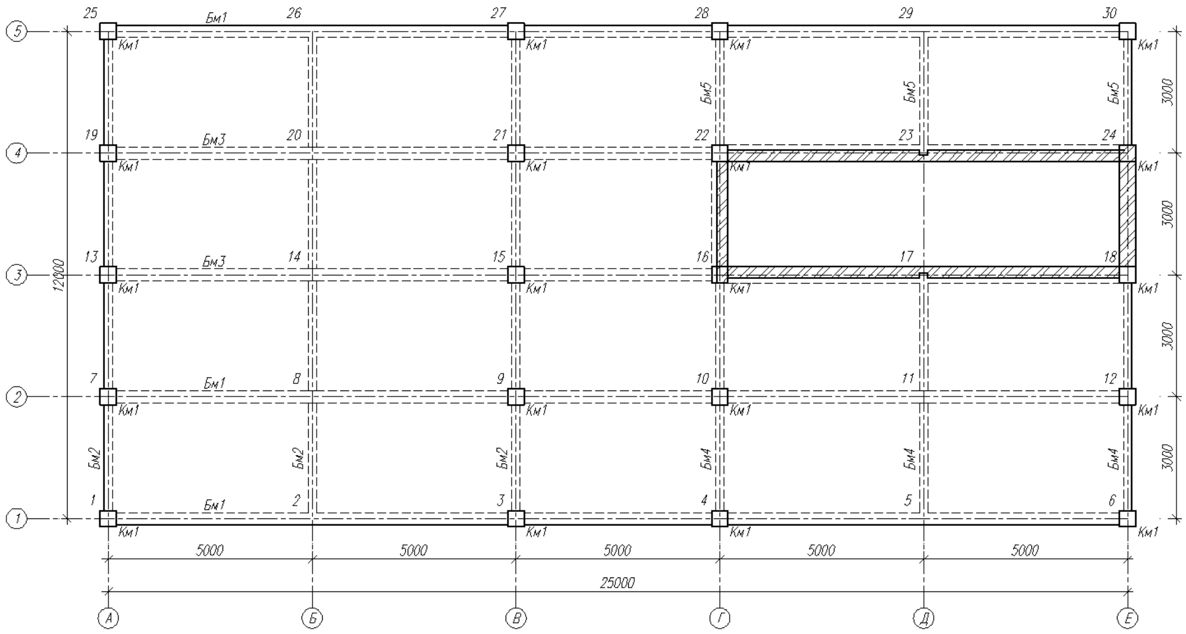


Figure 11.3.20-3. Structural layout of the floor

11.3.21 Column Reinforcement

This dialog box (Fig. 11.3.21-1) is used to specify the information on column reinforcement, which is required for designing. This information includes the data on the areas of the longitudinal and transverse reinforcement, class and standards of the reinforcement, and the value of the concrete cover.

In the **Reinforcement** group of the dialog box you can specify whether the column is reinforced by cages or by individual bars. This information applies to all columns.

Since the program implements the unification at the level of column names, for specification of the reinforcement information it suffices to describe the reinforcement of one column from each unified group. The **Column name** list contains original names of columns of each unified group. The node number next to the name refers to the first column in the group list.

The data on the reinforcement area are specified in the table. The information is divided into sections according to the number of column segments. **Segment** is a part of a column, the transverse reinforcement of which is different from other segments. The number of segments can be from two to five. The transverse reinforcement is more dense in the lower part of a column (the starter bars from the foundation or the ground floor), than in its middle part. The transverse reinforcement can also be denser in the upper part of a column. The longitudinal reinforcement is constant along the whole length of the column; therefore it may be specified only for the first segment.

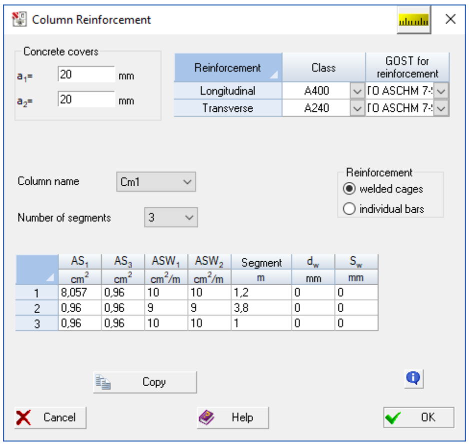


Figure 11.3.21-1. *The* **Column Reinforcement** *dialog box*

The length of a segment is specified in meters. If the number of segments has been changed, the number of lines in the table will be corrected according to the specified number of segments.

When specifying the reinforcement (Fig. 11.3.21-2) for the rectangular and circular cross-sections one table is used, however in case of a circular cross-section not all columns are filled. The data on the transverse reinforcement can be specified in two ways. In the first case the areas are specified (the third and fourth columns), and in the second case the diameter and spacing are specified (the sixth and seventh columns).

The table columns contain the following data:

the first column – the area of the longitudinal reinforcement running along the X axis of the floor (for the rectangular cross-section), and for the circular cross-section, the area of longitudinal reinforcement of the whole column section – AS_1 (cm^2);

the second column – the area of the longitudinal reinforcement running along the Y axis of the floor (for the rectangular cross-section), and for the circular cross-section, this column is not filled – AS_3 (cm^2);

the third column – the area of the transverse reinforcement running along the Y axis of the floor (for the rectangular cross-section), and the area of transverse reinforcement for the circular cross-section – ASW_1 (cm^2/m);

the fourth column – the area of the transverse reinforcement running along the X axis of the floor (for the rectangular cross-section), and for the circular cross-section, this column is not filled – ASW_2 (cm^2/m);

the fifth column – the length of a segment (in meters);

the sixth column – the diameter of transverse reinforcement;

the seventh column – the spacing between transverse reinforcement.

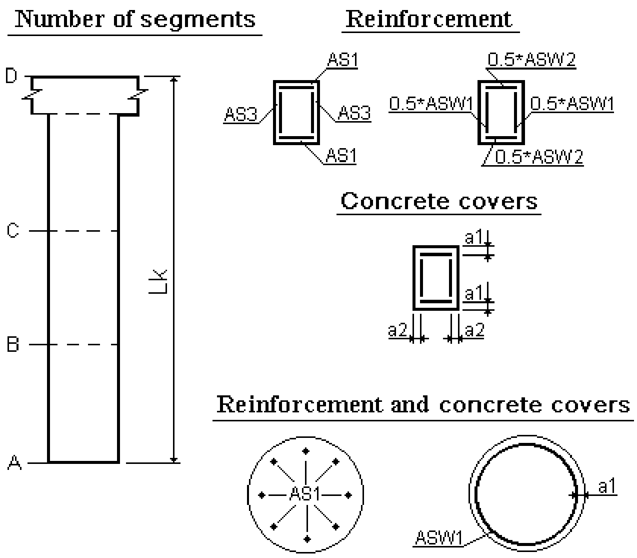


Figure 11.3.21-2. Column reinforcement arrangements

It is recommended to stick to the following order of data preparation in this dialog box:

- select the column from the list you want to specify the data for (the selected column will be highlighted in the layout);

- assign the class of longitudinal and transverse reinforcement, as well as the standard for the selected reinforcement assortment;
- specify the values of the concrete covers;
- specify the number of segments;
- fill the table.

Once the table is filled select the next column in the list and repeat the above steps.

In those cases when the characteristics of column reinforcement do not differ from the earlier specified data, you can use the **Copy** button. The number (name) of the column the reinforcement characteristics of which you want to copy for the current column is selected in the **Copy** dialog box.

The rules for specifying the column reinforcement are given in the information window, which can be invoked by clicking the  button.

In the **Reinforcement** group of the dialog box you can specify whether the column is reinforced by welded cages or by individual bars.

It should be noted that the **Column name** list will include only original names, i.e. there will be no repetitions of identical names. If there are unified groups of columns in the project, the list will contain only the name of the first column of the group with a corresponding node.

The drawings with the results of the design are generated only for the first column of each group. All other project materials (**Cages**, **Steel consumption**, **Overall specification**) will include the information on all columns of the respective groups.

11.3.22 Dividing Slabs by Beams

Due to this operation it is enough to describe several “large” slabs in the initial data dividing them further along the support lines into beams. In the result of the division of the initial slab, new slabs will be formed with their edges resting on the extreme and supporting nodes of the beams. Slabs generated in the result of the division will receive numbers arranged in the ascending order behind the maximum number in the name of the slab before the division. If necessary the numbers can be corrected in the table with a description of the geometry of the slab. The division should be performed before specifying the reinforcement of the slabs.

11.3.23 Initial Data Control

Clicking the **Apply** or **OK** buttons in the dialog boxes for the initial data preparation activates the control system. If errors have been detected in the result of the control, the screen will display the **Control results** dialog box (Fig. 11.3.23-1, *a*), which contains a table with the list of warnings (errors). Unfolding the list opens the description of the errors detected by the control system (Fig. 11.3.23-1, *b*).

The difference between clicking the **Apply** and **OK** button is that in the first case the control is performed on the background of the open input dialog box, while in the second case this dialog box is closed.

After clicking the **Apply** button (input dialog box remains open), the program response to clicking the **Go to edit** and **Continue** buttons in the **Control results** dialog box will be the same, i.e. the control will return to the input dialog box.

If control is performed after clicking the **OK** button (in which case the input dialog box is closed), then after clicking the **Go to edit** button the dialog box will automatically open again for making appropriate corrections in the data. If the **Continue** button is clicked, the program status will be the same as it had been before the **Control results** dialog box was displayed. In this case, the messages about "missed" errors will appear if you try to perform the design (Fig. 11.3.23-1, *c*).

a *b* *c*

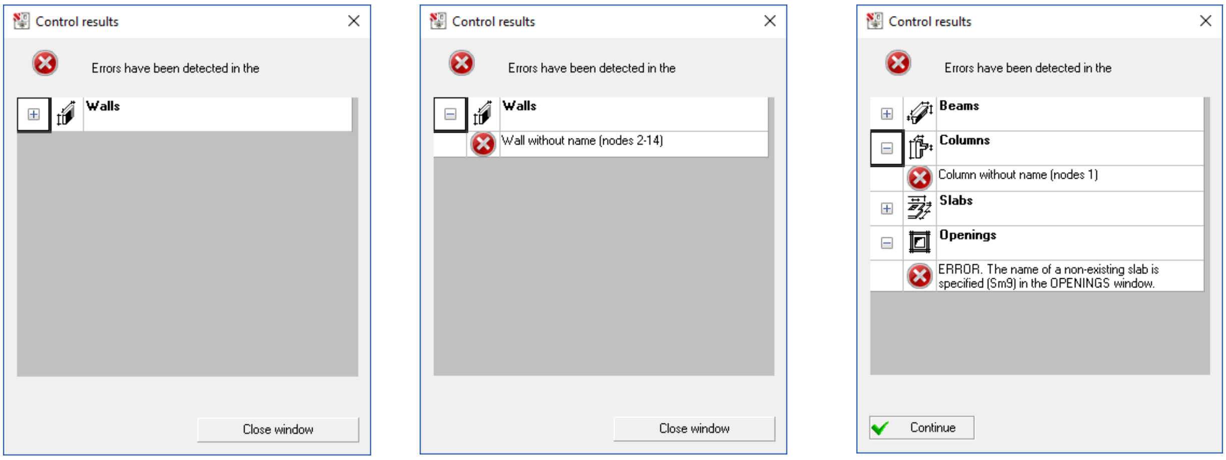
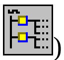


Figure 11.3.23-1. *The* **Control results** *dialog box*

11.4 Design

Once all the initial data are entered, you have to proceed to the project tree (the **Exit to project control screen** button ) and select the **Design** mode. This mode enables to perform the control of the initial data and to generate the output documents.

If some errors have been detected in the initial data or if certain situations arise when designing cannot be performed, a message window will appear containing warnings and/or description of the errors (see the *Initial Data Control* Section). Otherwise, designing of the floor is performed and the project documentation is generated.

To view the results of designing you have to go to the **Drawings RCS** and **Drawings DRCI** items in the project tree. Drawings can be generated in A3 or A4 formats.

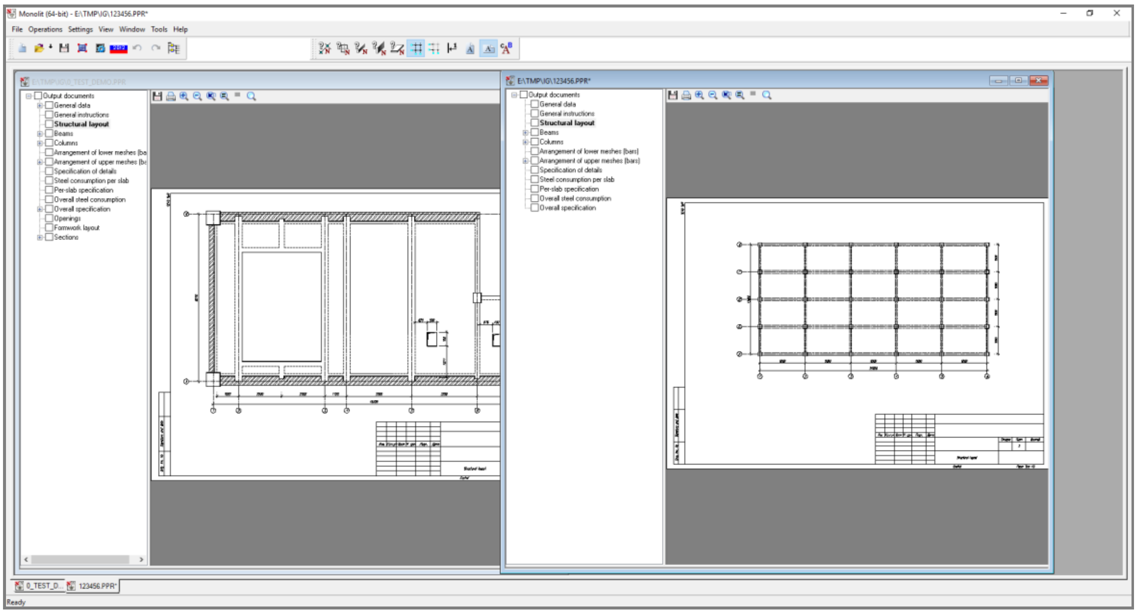


Figure 11.4-1. Windows with the designing results

Selecting the type of drawings and paper size in the project tree displays the window with the results of designing. The list of the generated project documents is given in the form of a tree (see Fig. 11.4-1), which works in the same way as the project tree, i.e. clicking on a project item displays the respective project materials in the window.

The tree has several levels. The first level contains the list of individual documents, e.g. *General instructions*, *Structural layout*, *Formwork layout*, etc., as well as groups of the output documents, each of which includes references to the list of documents of one type, e.g. *Sections*, *Beams*, *Columns*. The second level, e.g. the *Beams* group, lists all beams of the floor, and the third level contains drawings of the spans of each beam. Next to the group name there is the button that serves to open the contents of the group (marked with a cross), as well as the checkbox-button — ☐ (it is located near each document). Checking this checkbox — ☑ means that, in the documenting mode the selected document or group must be printed out or opened in the DXF format.

In the design results analysis mode, two toolbar strips are available – **General operations** and **Display filters**. The **General operations** strip is identical to the respective toolbar strip of the initial data entry mode and is described above. The set of operations (buttons) of the filters strip includes the display of the numbers of nodes, columns, beams, walls and slabs, two buttons for displaying the grid lines, one button for the specification of sections, two buttons for the selection of the orientation of drawings, and one button for the specification of the font when exporting the drawings into the DXF formats of the AutoCAD software file (Fig. 11.4-2).

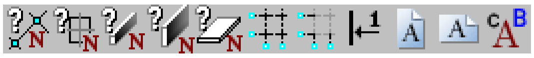


Figure 11.4-2. *The* **Display filters** *strip*

11.4.1 Output Documents

In the result the program generates a set of output documents for the designed floor.

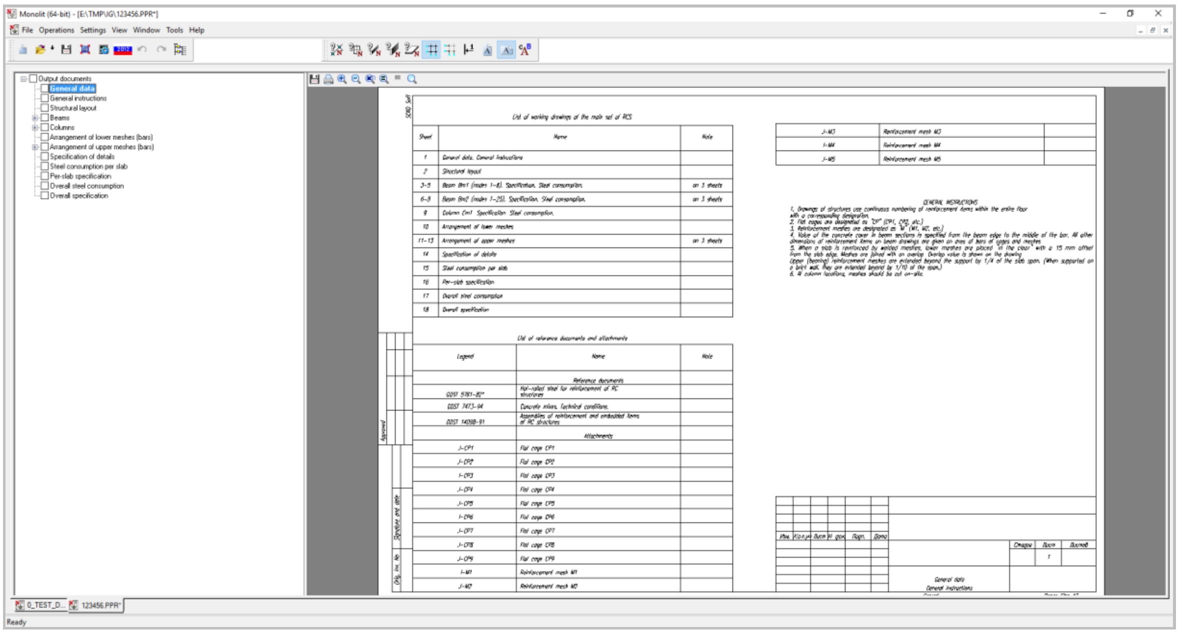


Figure 11.4.1-1. The *General data* document

General data — includes the list of working drawings of the main set, and the list of reference documents and attachments. If the list takes more than one sheet, the results tree will display the group with the list of sheets. The list of drawings includes the stamp (Form 3) and can be given either in the "Portrait" orientation (A4 paper size) or "Landscape" orientation (A3 paper size). When the A3 paper size is used, the *General instructions* can also fit in the same sheet (Fig. 11.4.1-1). In this case the *General data* and *General instructions* documents may coincide. If the A4 paper size is used, the *General instructions* are given on a separate sheet (Fig. 11.4.1-2).

Structural layout (Fig. 11.4.1-3) — the drawing shows the floor layout. Depending on the selected display filters, the layout may indicate the numbers of nodes, the names of columns, walls, beams, and slabs.

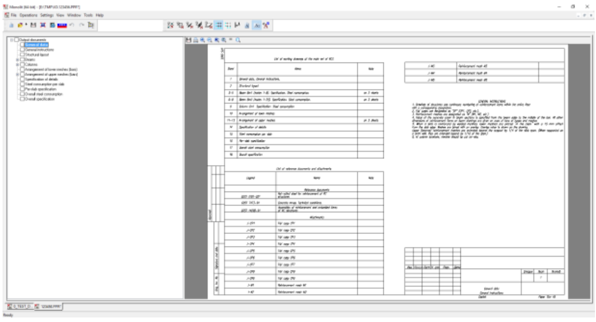


Figure 11.4.1-2. The *General instructions* document

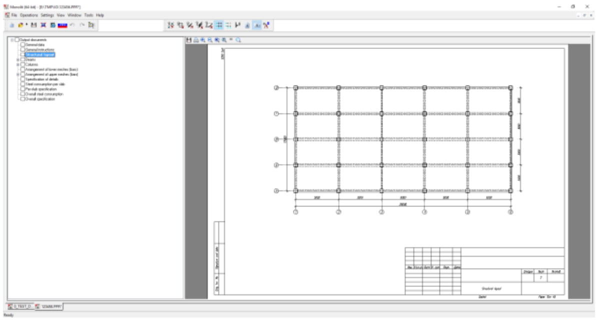


Figure 11.4.1-3. The *Structural layout* document

Beams — each beam drawing includes one or two spans, depending on the paper size and number of spans. Shown for each span are three sections (near the supports and in the middle) with the arranged cages, meshes, bars, their names and snaps, as well as the necessary dimensions. Each beam is provided with the specification and information on the steel consumption.

In case of the A4 paper size (Fig. 11.4.1-4, *b*), one sheet contains only one beam span, and this sheet can be generated in any orientation.

A sheet containing text information (specification, information on the steel consumption) is generated only in the portrait orientation. In case of a multispan beam there can be two spans on one A3 sheet (Fig. 11.4.1-4, *a*). A3 sheets are generated only in the landscape orientation.

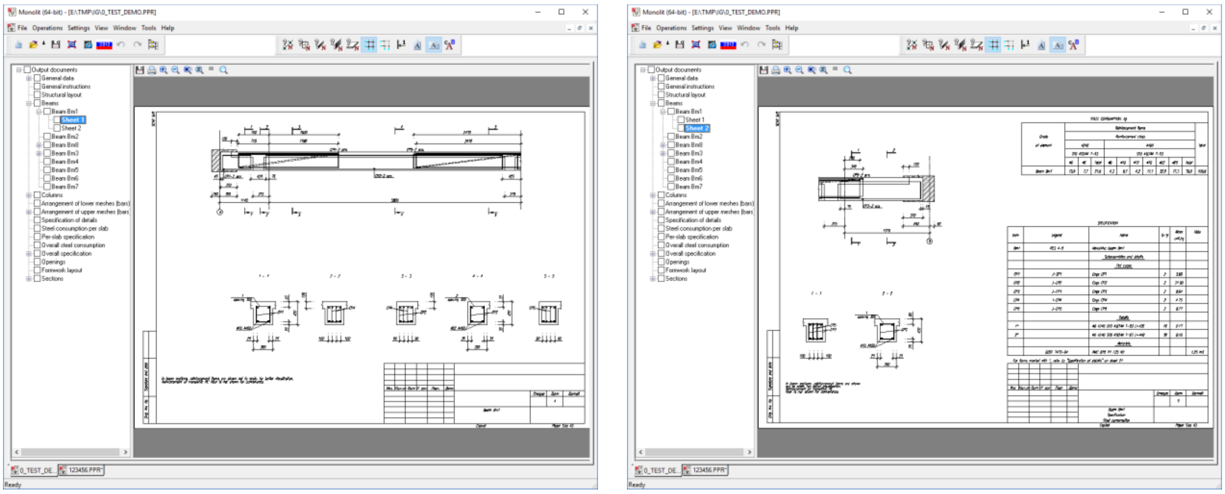


Figure 11.4.1-4, *a*. Beam drawings in the A3 formats

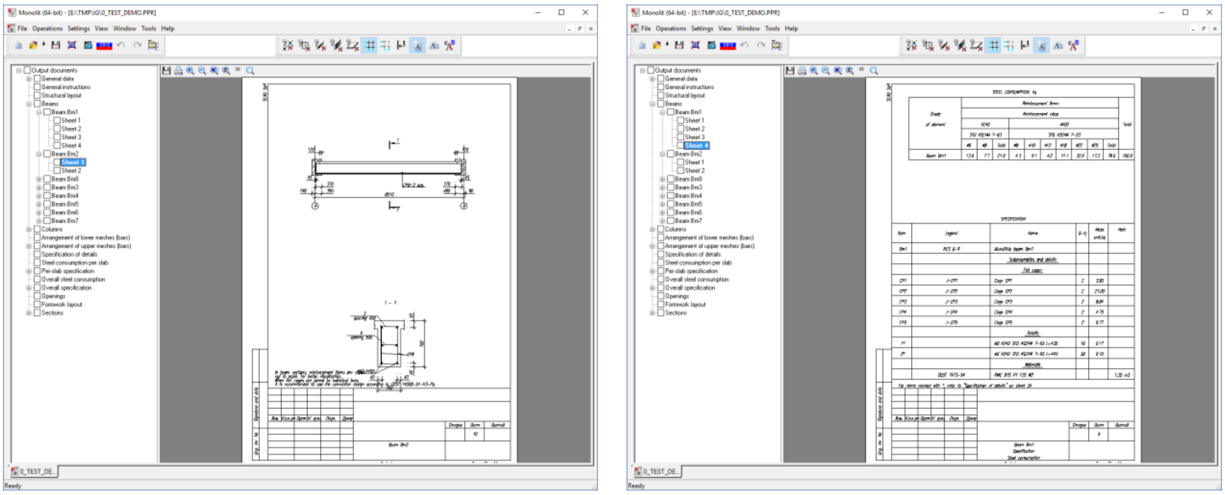


Figure 11.4.1-4, *b*. Beam drawings in the A4 formats

Columns — each column drawing includes two or three sections (depending on the number of the specified segments) with the arranged cages, bars, their names and snaps, as well as the necessary dimensions. Each column is provided with the specification and information on the steel consumption. In case of the A4 paper size (Fig. 11.4.1-5, *a*), two sheets are always generated for one column in the portrait orientation: one sheet contains the drawing, and the other one contains the specification and information on the steel consumption. In case of the A3 paper size (Fig. 11.4.1-5, *b*), one sheet is generated for each column in the landscape orientation.

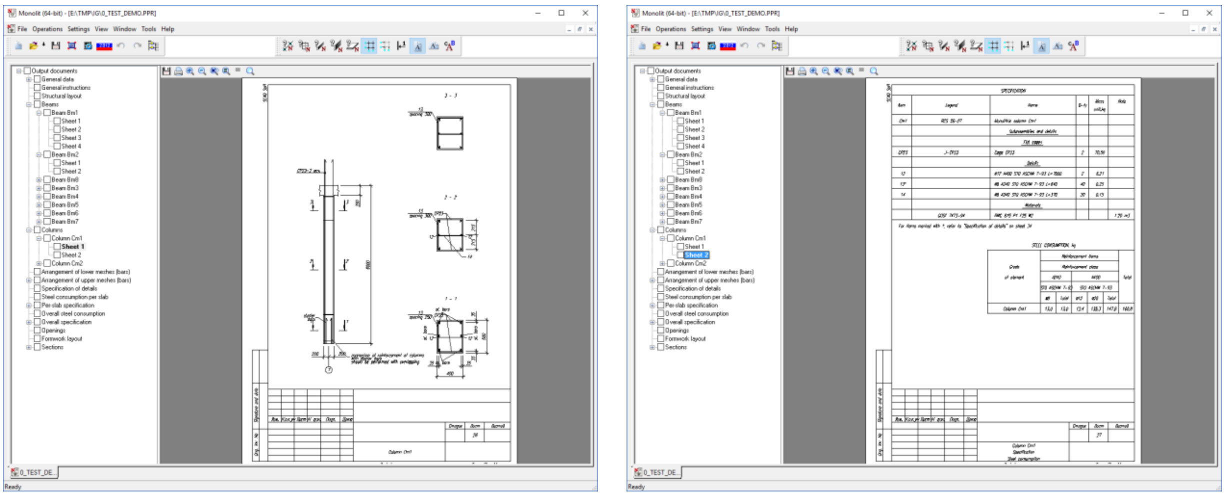


Figure 11.4.1-5, *a*. Column drawings in the A4 formats

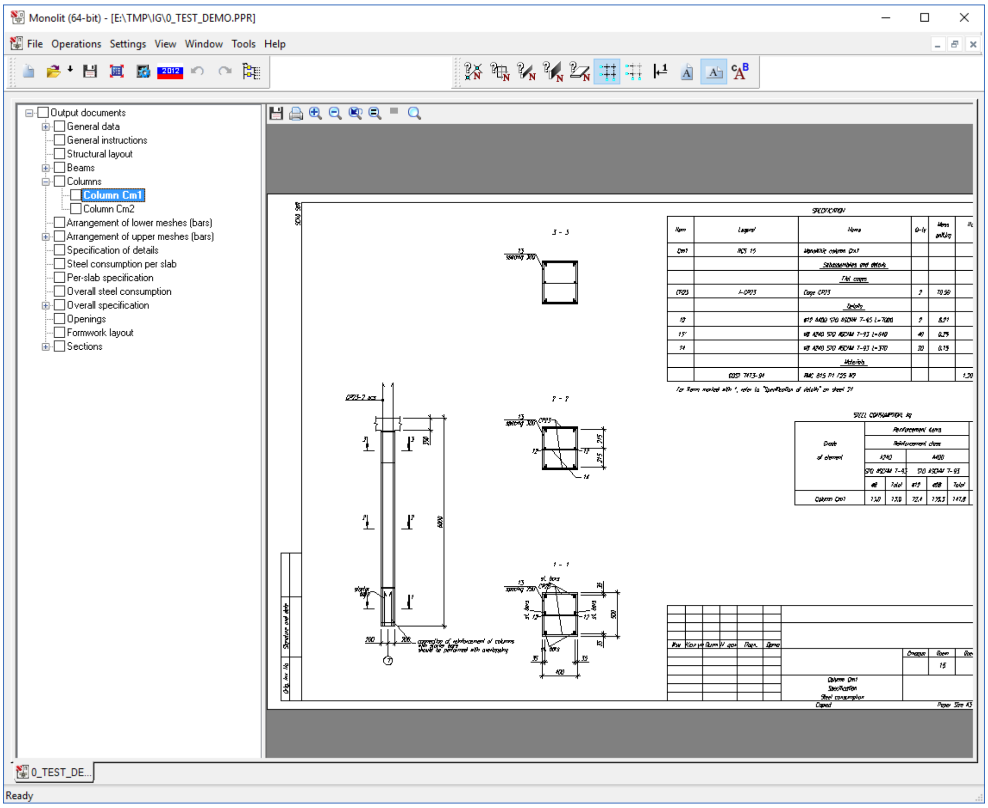


Figure 11.4.1-5, *b*. Column drawings in the A3 formats

Arrangement of lower meshes (bars) (Fig. 11.4.1-6) — the drawing depicts the arrangement layout of the lower meshes with the indication of their names and quantity or the arrangement layout of individual bars with the indication of their spacing. Drawings can be generated in any orientation.

a *b*

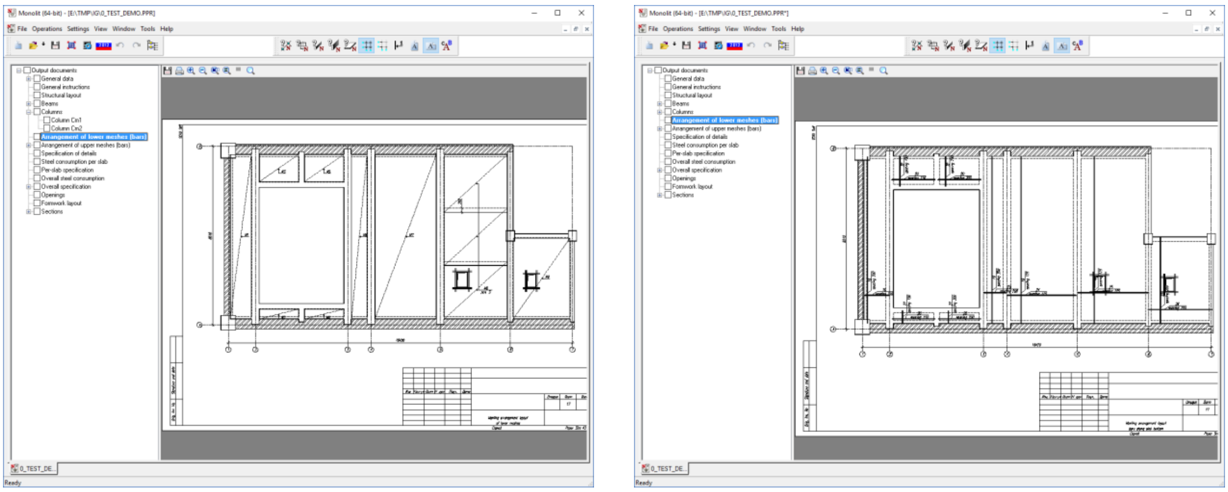
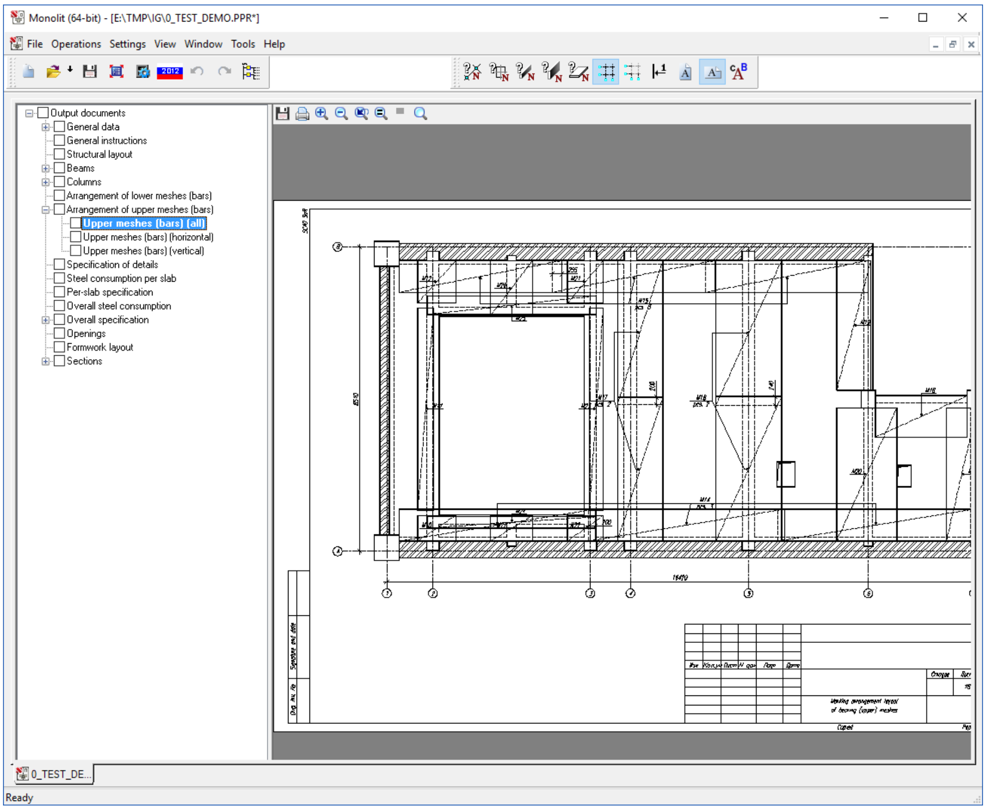


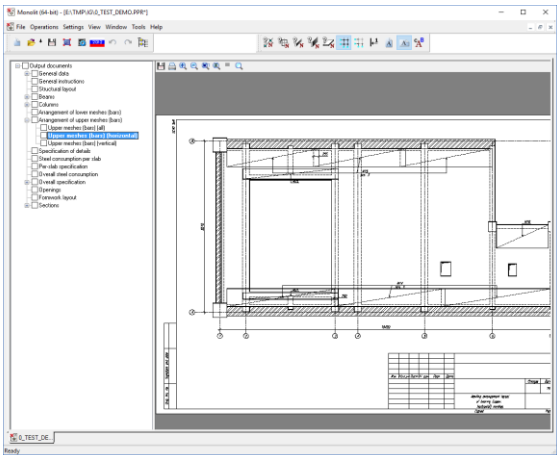
Figure 11.4.1-6. Drawings of the arrangement of the lower reinforcement meshes (*a*) and bars (*b*)

Arrangement of upper meshes (bars) (Fig. 11.4.1-7) — the drawing depicts the arrangement layout of the upper meshes with the indication of their names and quantity or the arrangement layout of individual bars with the indication of their spacing. You can obtain drawings of three types: Upper meshes (bars) (all); Upper meshes (bars) (horizontal), running along the X axis; Upper meshes (bars) (vertical), running along the Y axis. It is recommended to generate the drawings of the second and third type for those projects in which, when plotting all meshes (bars) on one sheet, their identification is difficult due to high density of the image.

a



b



c

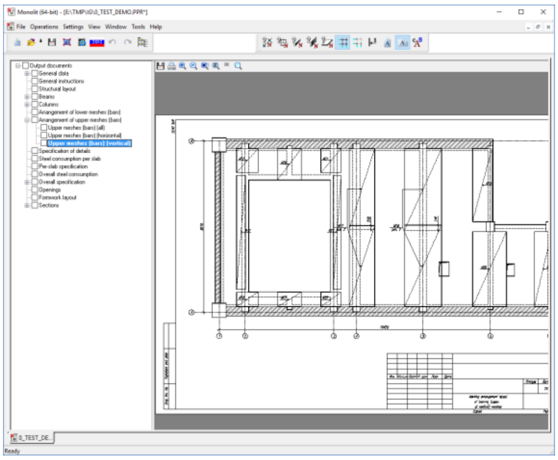


Figure 11.4.1-7. Drawings of the arrangement of the upper reinforcement meshes:
a) all; *b*) only horizontal meshes (running along the X axis); *c*) only vertical meshes (running along the Y axis);

Specification of details (Fig. 11.4.1-8) — if the bent bars were used in the designing of the floor, they will be included in the respective specification. This document can be of either portrait (A4 paper size) or landscape (A3 paper size) orientation.

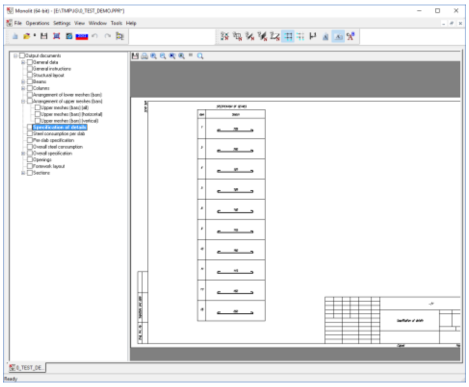


Figure 11.4.1-8. Specification of details

Steel consumption per slab (Fig. 11.4.1-9) — this document contains information on the steel consumption for all slabs (segments) of the floor. The document is generated in the landscape orientation (A3 paper size). In case of the A4 paper size the document can be generated in any orientation.

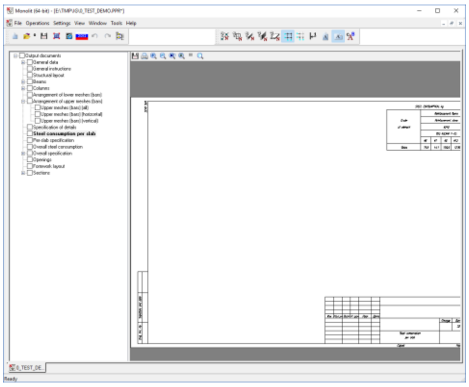


Figure 11.4.1-9. Steel consumption per slab

Per-slab specification (Fig. 11.4.1-10) — this document contains the list and parameters of the meshes or details used in all floor slabs, as well as the total amount of concrete required for the designing of slabs. This document can be of either portrait (A4 paper size) or landscape (A3 paper size) orientation.

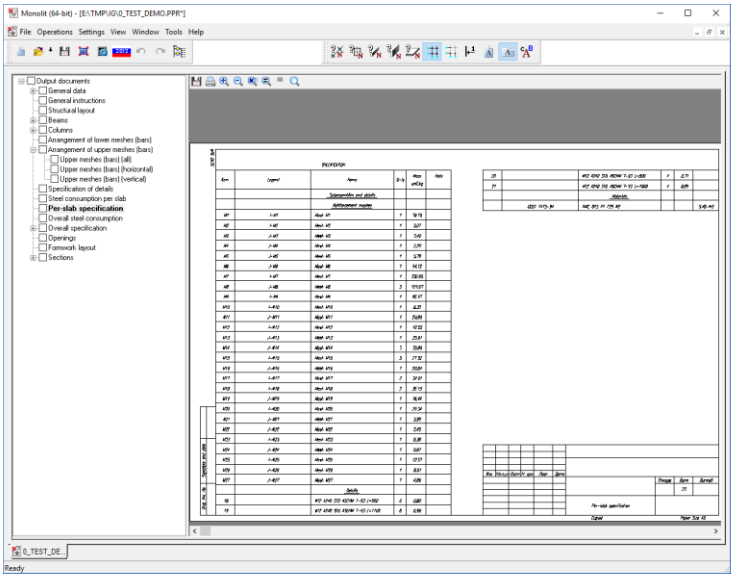


Figure 11.4.1-10. Per-slab specification

Overall steel consumption (Fig. 11.4.1-11) — this document provides information on the steel consumption for the whole floor. This document is generated in the landscape orientation.

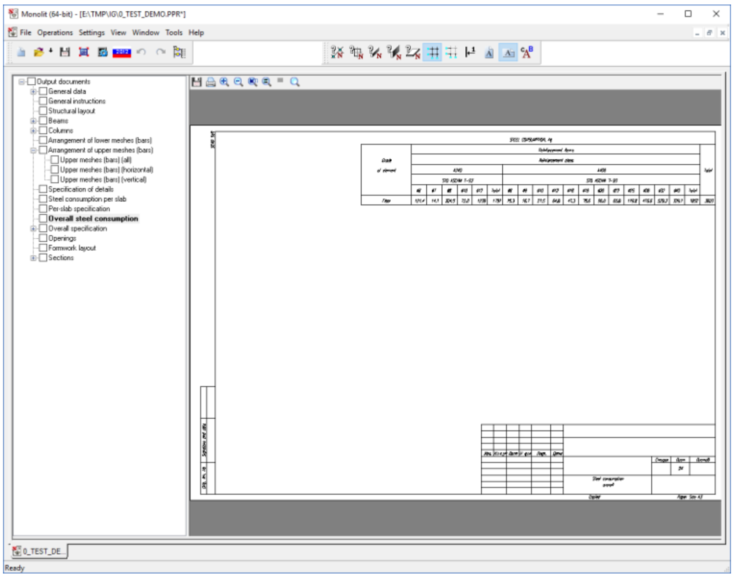


Figure 11.4.1-11. Overall steel consumption

Overall specification (Fig. 11.4.1-12) — this document contains the list and parameters of the cages, meshes and details used in all floor beams and slabs, as well as the total amount of concrete required for the designing of the floor. If the specification takes more than one sheet, the group will include references to those sheets. The document is generated in the portrait orientation for the A4 paper size and in the landscape orientation for the A3 paper size.

a *b*

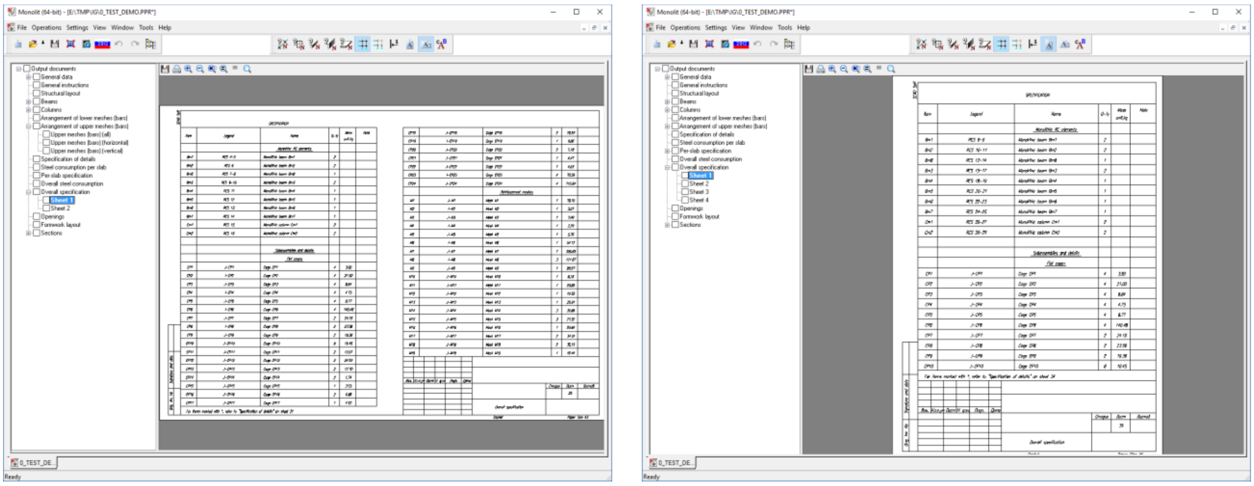


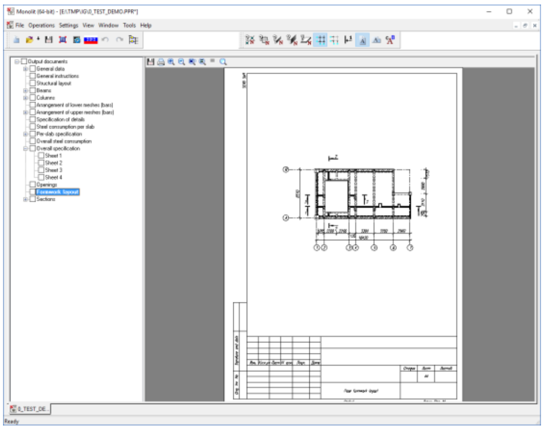
Figure 11.4.1-12. Overall specification:
a) A3 paper size; *b*) A4 paper size

The *Formwork layout* and *Sections* items in the results tree (Fig. 11.4.1-13) appear after the specification of sections (the **Setting of sections** button in the **Display filters** toolbar of the project documentation generation mode). Sections are specified in the **Sections** dialog box (Fig. 11.4.1-14) according to the following rules:

- horizontal sections (along the X axis) and vertical sections (along the Y axis) are specified in the respective tables;

- before specifying the characteristics of each subsequent section it is necessary to add a new line in the respective table (click the **Add** button);
- each section must have a unique number (the **Section** column);
- in the current version of the program, the openings in slabs through which the section line passes are not shown in this section.

a



b

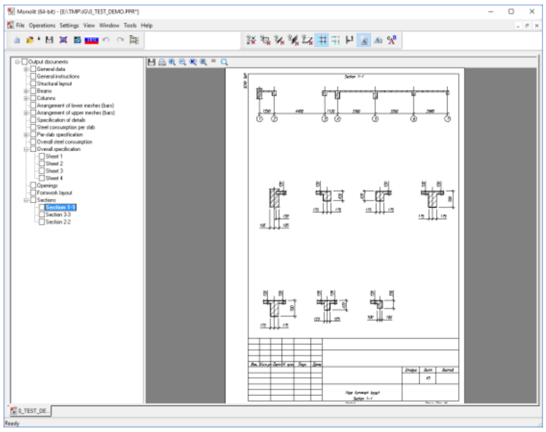


Figure 11.4.1-13. Formwork layout (*a*) and sections (*b*)

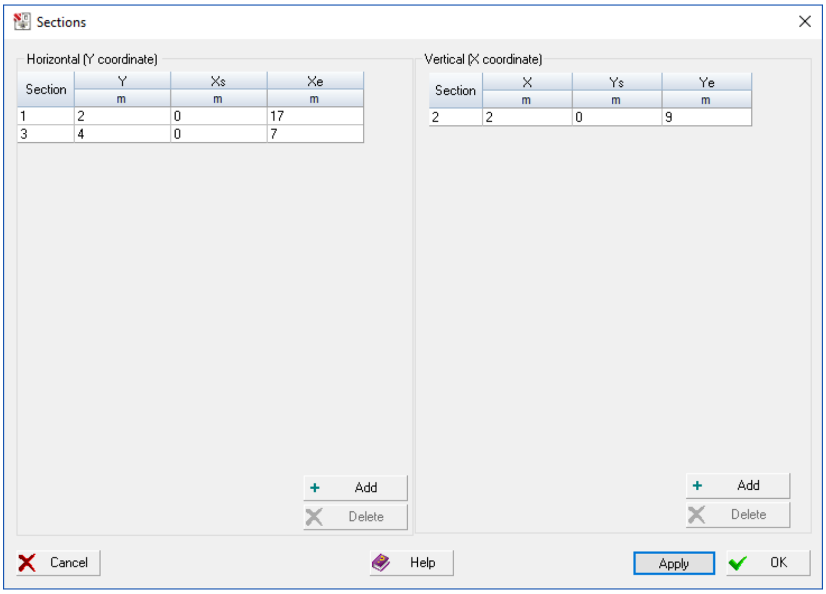


Figure 11.4.1-14. *The* **Sections** *dialog box*

Cages and meshes — the set of DRCI drawings includes cages and meshes (Fig. 11.4.1-15) and is accessible in the project tree in those cases when reinforcement of beams and slabs is performed with the corresponding reinforcement items.

a *b*

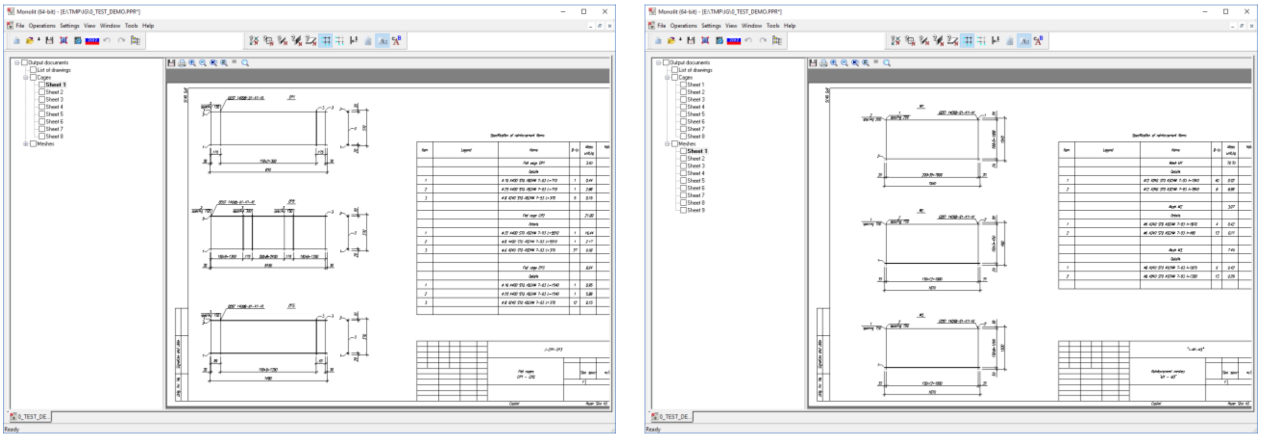


Figure 11.4.1-15. Drawings of cages (*a*) and meshes (*b*)

11.5 Information Modes

The **Concrete class**, **Concrete grade** and **Reinforcement** information modes are accessible from the **Information** level of the project tree.

11.6 References

[1] Guide on designing of concrete and reinforced concrete structures made of heavy-weight concrete (no prestressing) (to SP 52-101-2003). Moscow, 214 p.

[2] Design of reinforced concrete structures: Reference manual / Golyshev A.B., Bachinsky V.Ya., Polischuk V.P., Kharchenko A.V., Rudenko I.V.; Ed. A.B.Golysheva — 2-nd ed., rev. and expanded — K.: Budivelnyk, 1990.— 544 p.

[3] Manual on designing of concrete and reinforced concrete structures 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 Reinforced Concrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1978, 175 p.

12. Decor

DECOR enables you to perform structural analysis and various checks of members and joints of timber structures for compliance with requirements of SNiP II-25-80 "Timber structure" [69], KMK 2.03.08.98 (or SP 64.13330.2011, SP 64.13330.2017). The application also provides reference data which is often used when designing timber structures.

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

12.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 12.1-1) with a set of buttons for selecting a working mode.

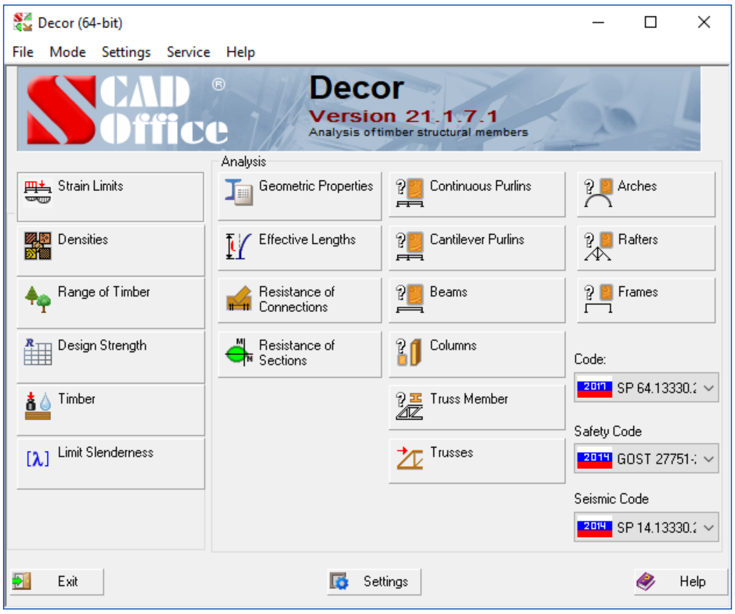


Figure 12.1-1. *The main window*

Design codes can be selected from the respective 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 according to SNiP II-25-80, KMK 2.03.08.98 and SP 64.13330.2011, СП 64.13330.2017 (updated edition of SNiP II-25-80).

A mode can be invoked by clicking the respective button. They are classified into reference modes (the **Information** group) or the design modes (the **Analysis** group).

The reference modes include:

Deflection and Strain Limits — provides the limit values of deflections of timber structural members and limit values of deformations of joints from Tables 15, 16 of SNiP II-25-80, KMK 2.03.08.98 (Tables 18, 19 of SP 64.13330.2011, Table 17 of SP 64.13330.2017);

Densities — provides information on the density of wood according to Annex 3 of SNiP II-25-80, KMK 2.03.08.98 (Annex E of SP 64.13330.2011, Annex G of SP 64.13330.2017);

Range of Timber — provides information on the sizes of softwood sawn timber according to GOST 24454-80;
Design Strength — enables to calculate the design strength for the type of stress-strain state selected by the user;
Timber — provides data on the maximum moisture content of timber, types and grades of glue;
Limit Slenderness — provides information on the limit slenderness of structural members.

The design modes include:

Geometric Properties — enables to calculate the geometric properties of a given cross-section;
Effective Lengths — implements recommendations of Sec. 4.21 of SNiP II-25-80, KMK 2.03.08.98 (Sec. 6.23 of SP 64.13330.2011, Sec. 7.23 of SP 64.13330.2017) for determination of the effective lengths;
Resistance of Connections — enables to determine the utilization factors of restrictions for notched connections and connections with cylindrical dowels;
Resistance of Sections — enables to determine the utilization factors of restrictions for any cross-section type available in the application database, under the action of arbitrary forces; and to plot the interaction curves for any admissible combination of the pairs of forces;
Continuous Purlins — enables to check continuous purlins for the ultimate and serviceability limit states;
Cantilever Purlins — enables to check cantilever purlins under uniformly distributed loads for the ultimate and serviceability limit states;
Beams — enables to check single-span beams with various boundary conditions for the ultimate and serviceability limit states;
Columns — this mode enables to check columns and posts;
Truss Member — this mode enables to check a separate truss member;
Trusses — this mode enables to perform all the necessary strength and stability checks of the truss members for the most common structural designs. The work begins with determining the design values of internal forces caused by the given vertical external loads;
Arches — this mode enables to perform strength and stability checks of arches from glued laminated timber of a rectangular cross-section;
Rafters — this mode enables to check rafters;
Frames — this mode enables to check frame structures.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

12.2 Information Modes

The reference modes provide data given in SNiP and SP. All values in the tables are given in the same units of measurement as in SNiP and SP, and do not depend on the settings of the application.

12.2.1 Deflection and Strain Limits

This mode (Fig. 12.2.1-1) provides the limit values of deflections of timber structural members and limit values of deformations of joints from Tables 15, 16 of SNiP II-25-80, KMK 2.03.08.98 (Tables 18, 19 of SP 64.13330.2011, Table 17 of SP 64.13330.2017).

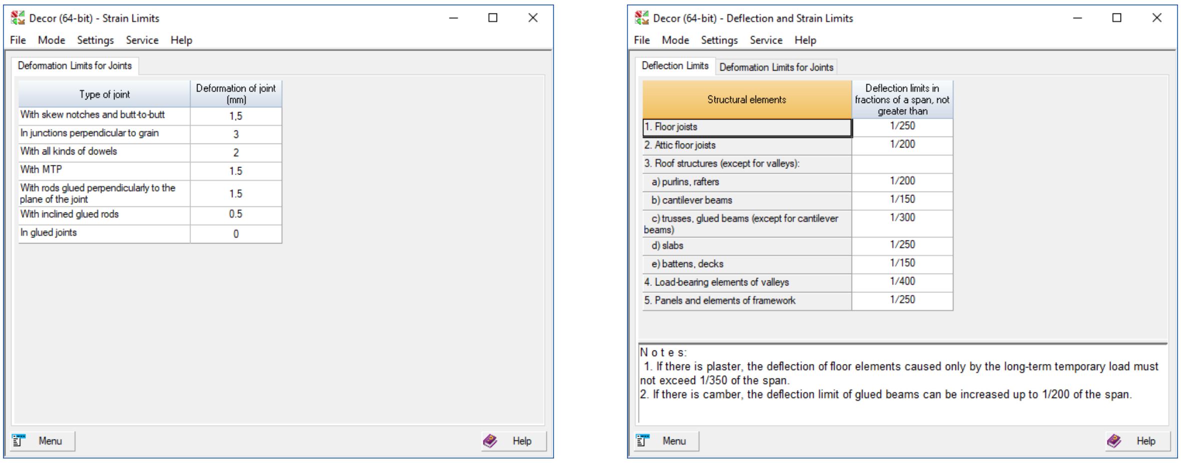


Figure 12.2.1-1. *Dialog boxes of the* **Deflection and Strain Limits** *mode*

12.2.2 Densities

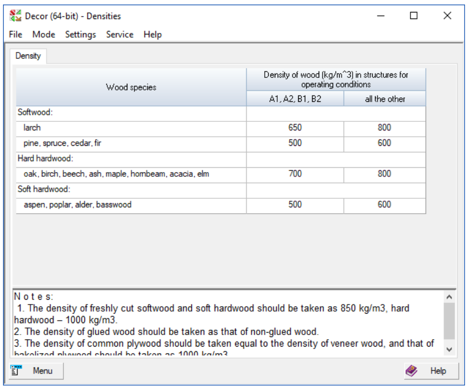


Figure 12.2.2-1. *The* **Densities** *dialog box*

Information provided in this mode (Fig. 12.2.2-1) includes the data on the density of wood according to Annex 3 of SNiP II-25-80, KMK 2.03.08.98 (Annex E of SP 64.13330.2011, Annex F of SP 64.13330.2017).

12.2.3 Range of Timber

This mode provides information on the sizes of edged and unedged softwood sawn timber according to GOST 24454-80 (Fig. 12.2.3-1). Moreover, the **Sawn Timber for GLTS** tab (Fig. 12.2.3-2) enables to specify the dimensions of sawn timber before planing and to obtain the dimensions after planing calculated in compliance with GOST 7307-75* (ГОСТ 7307-2016).

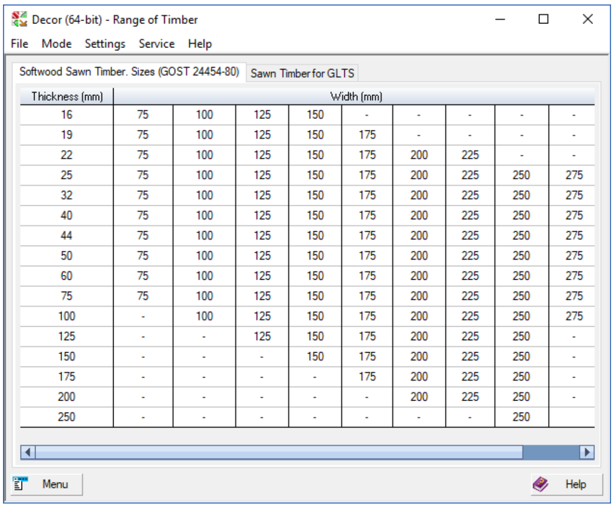


Figure 12.2.3-1. *The* **Softwood Sawn Timber** *tab of the* **Range of Timber** *dialog box*

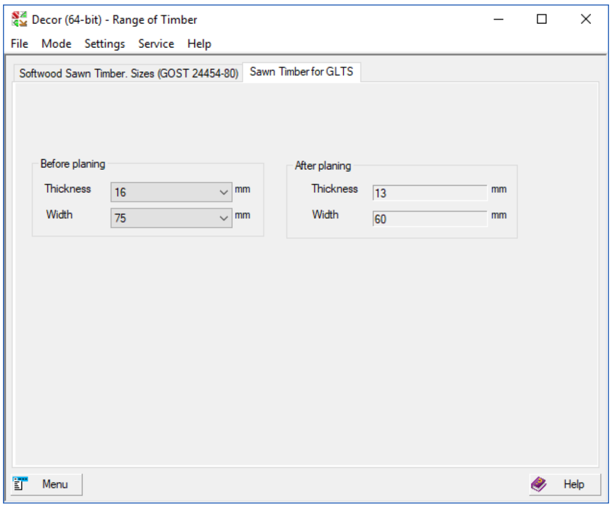


Figure 12.2.3-2. *The* **Sawn Timber for GLTS** *tab of the* **Range of Timber** *dialog box*

12.2.4 Design Strength

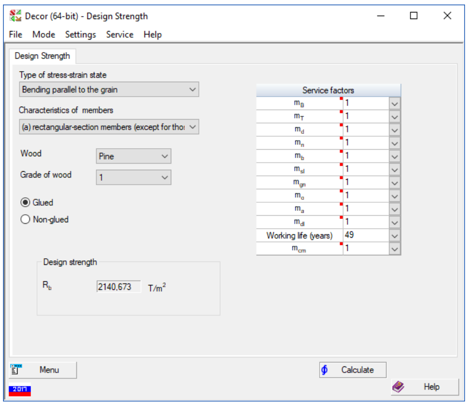


Figure 12.2.4-1. *The* **Design Strength** *dialog box*

In order to determine the design strength of a member in compliance with Sec. 3.1, 3.2 of SNiP II-25-80, KMK 2.03.08.98 (or Sec. 5.1,5.2 of SP 64.13330.2011, Sec. 6.1-6.9 of SP 64.13330.2017), you have to select the following data from the respective lists of the **Design Strength** dialog box (Fig. 12.2.4-1):

- type of stress-strain state;
- characteristics of members;
- wood species;
- grade of wood.

Use the radio buttons to select the type of section (glued or non-glued) and specify the service factors (m_a, m_T, m_d, ...) by selecting their values from the lists, or by entering the values directly in the list fields if the factors are different from those recommended by the design codes (the lists contain all values given in the respective tables of SNiP or SP).

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

12.2.5 Timber

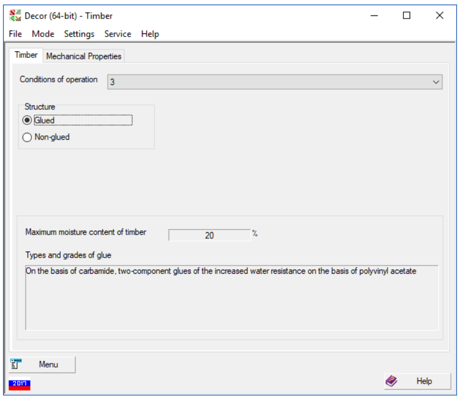


Figure 12.2.5-1. *The* **Timber** *tab of the* **Timber** *dialog box*

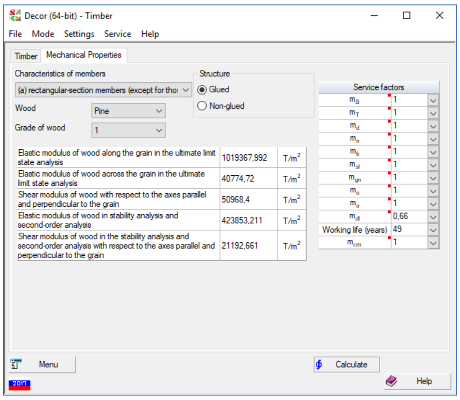


Figure 12.2.5-2. *The* **Mechanical Properties** *tab of the* **Timber** *dialog box*

This mode (Fig. 12.2.5-1) provides data on the maximum moisture content of timber according to Table 1 of SNiP II-25-80, KMK 2.03.08.98 or SP 64.13330. The moisture content depends on the given conditions of operation of the structure. Moreover, the types and grades of glue are given for glued laminated timber (according to Table 2 of SNiP II-25-80 or SP 64.13330).

The mechanical properties of wood (elastic moduli, shear moduli and Poisson's ratios) can be determined on a separate tab (Fig. 12.2.5-2) according to Sec. 3.5 of SNiP II-25-80, KMK 2.03.08.98 (Sec. 5.3-5.5 of SP 64.13330.2011).

12.2.6 Limit Slenderness

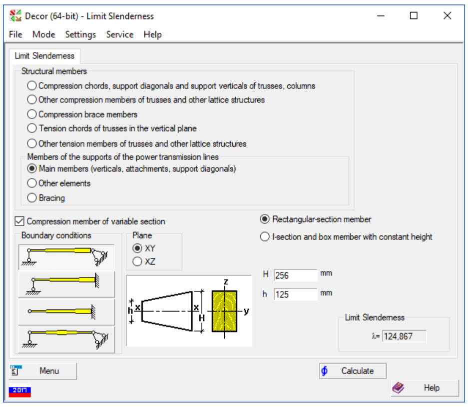


Figure 12.2.6-1. *The* **Limit Slenderness** *dialog box*

The limit slenderness is calculated for various kinds of structural members on the basis of the data from Table 14 of SNiP II-25-80, KMK 2.03.08.98 (Table 17 of SP 64.13330.2011, Table 16 of SP 64.13330.2017). The dialog box (Fig. 3.6-1) contains two groups of radio buttons which define the type of members (members of the supports of the power transmission lines make up a separate group); a group of radio buttons for selecting the section plane; radio buttons for selecting the section shape (rectangular, I-section, or box).

If the *Compression member of variable section* checkbox is checked, you have to use the respective buttons to specify the boundary conditions. The limit slenderness value appears in the respective field once you click the **Calculate** button (according to Table 14 of SNiP and Table 1 of Annex 4 of SNiP II-25-80, KMK 2.03.08.98 or Table 17 of SP 64.13330.2011 and Table F.1 of Annex F of SP 64.13330.2011 or Table 16 of SP 64.13330.2017 Table F.1 of Annex F of SP 64.13330.2017).

12.3 Analysis

12.3.1 General Operations

Most modes include a number of common controls. These include groups of controls for creating cross-sections, characteristics of timber, service factors etc. Such groups of controls are described below.

Creating Cross-Sections

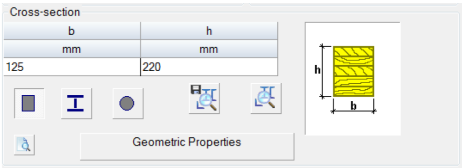


Figure 12.3.1-1. *Section types*

DECOR works with three types of cross-sections of timber members: rectangular, I-section, and round. The **Cross-section** group (Fig. 12.3.1-1) is used to select the shape of a section and to specify its sizes.

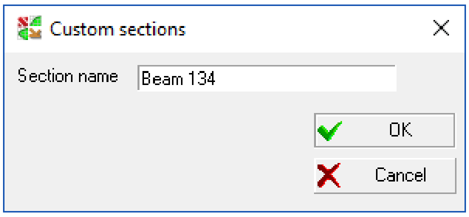


Figure 12.3.1-2. *The* **Custom sections** *dialog box*

The button  provides access to the custom sections database. This database is created gradually as you work with the application. To save a new section in the database, use the button , which invokes the **Custom sections** dialog box where you can specify a name for the section (Fig. 12.3.1-2).

Since the application does not verify the uniqueness of the names used, it is the user who has to take care of it.

The geometry of the specified section can be checked by clicking the **Preview** button , which will open the **Preview** dialog box with this section (Fig. 12.3.1-3).

When you access the custom sections database, a dialog box appears (Fig. 12.3.1-4), that contains a list of all sections saved in the database. The sections can be deleted from the database or renamed (the **Delete** and **Rename** buttons), viewed (**Preview** ) or opened in the application for subsequent use. In the latter case you have to select a line with the name of the section you want to use and click the **Apply** button.

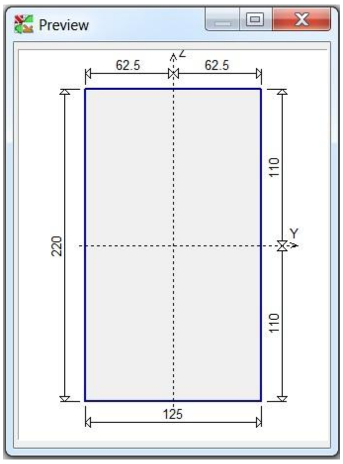


Figure 12.3.1-3. *The* **Preview** *dialog box*

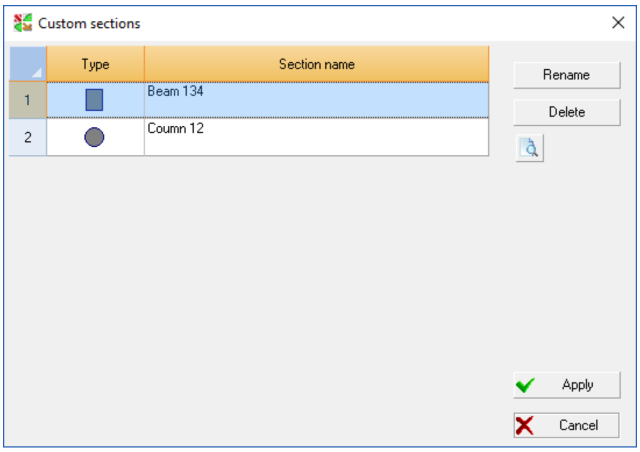


Figure 12.3.1-4. *A dialog box with the list of the custom sections*

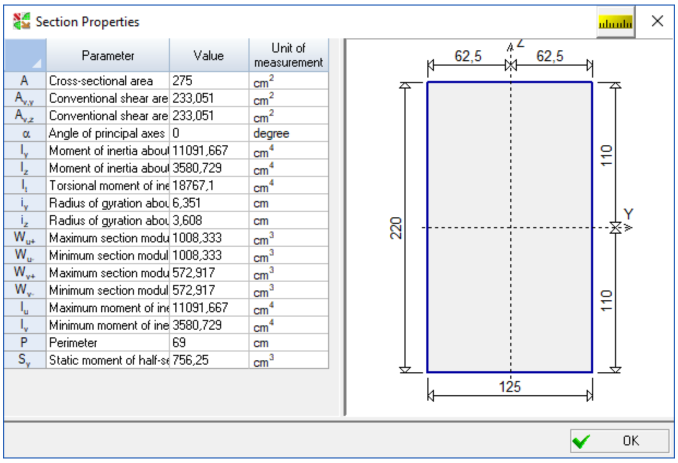


Figure 12.3.1-5. *The* **Section properties** *dialog box*

The **Geometric Properties** button of the **Cross-section** group invokes the **Section properties** dialog box (Fig. 12.3.1-5), which contains the geometric properties of the section (area, moments of inertia etc.) and the section with its sizes and axes.

Characteristics of Timber

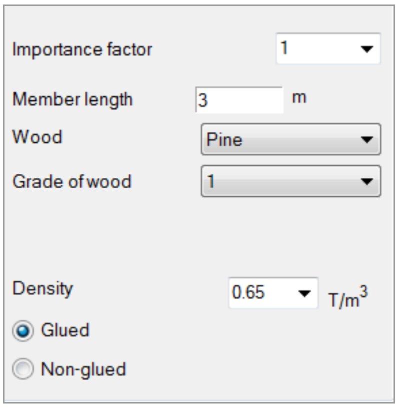


Figure 12.3.1-6. A group of controls used to specify the characteristics of timber

The characteristics of timber are required in all the design modes. A standard group of controls (Fig. 12.3.1-6) is usually used to specify this information. Such characteristics as the wood species (pine, fir, oak, ...), the grade of wood, and the dead weight are selected from the respective lists. The latter parameter is required only in those modes where the load from the self-weight can be specified (e.g., beams or purlins). Moreover, the respective radio buttons are used to indicate whether a section is a glued one.

This group of controls usually contains a list for selecting or entering the importance factor according to GOST 27751-88.

Service Factors

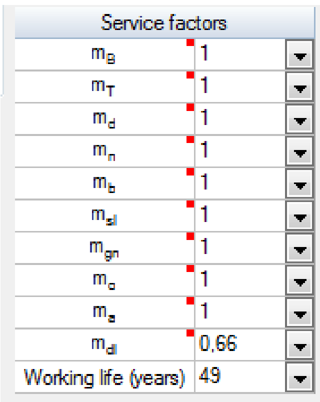


Figure 12.3.1-7. Service factors

Information on the service factors (m_a, m_T, m_d, ...) is required in order to analyze a structural member or joint. The service factors can be specified by selecting their values from the lists (the lists contain all values defined in the respective sections and tables of SNiP or SP) or by entering them directly in the list fields (Fig. 12.3.1-7).

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

Some coefficients are determined automatically by the program. For example, the service life factor is calculated on the basis of the service life data (see Figure 12.3.1-8).

12.3.2 Geometric Properties

This mode is used to calculate the geometric properties of cross-sections according to the rules described above (see Section 12.3.2). The initial data are entered in the **Section Properties** tab (Fig. 12.3.2-1). The results are output in the **Geometric Properties** tab in the form shown in Fig. 12.3.2-2.

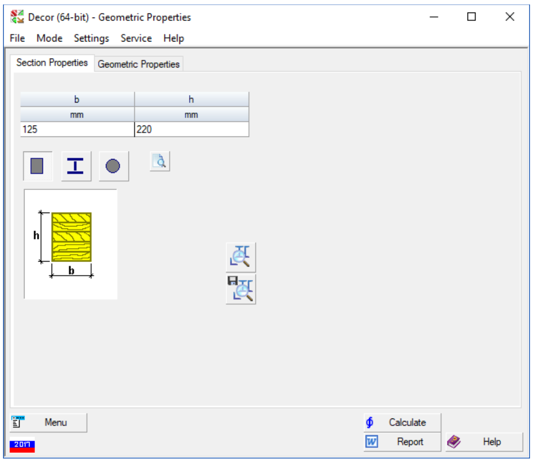


Figure 12.3.2-1. *The* **Section Properties** *tab of the* **Geometric Properties** *dialog box*

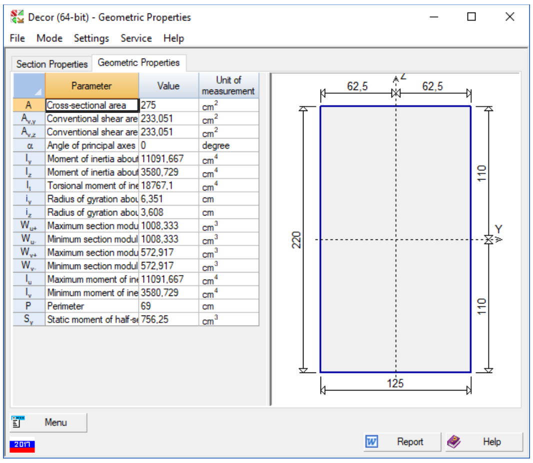


Figure 12.3.2-2. *The* **Geometric Properties** *tab of the* **Geometric Properties** *dialog box*

12.3.3 Effective Lengths

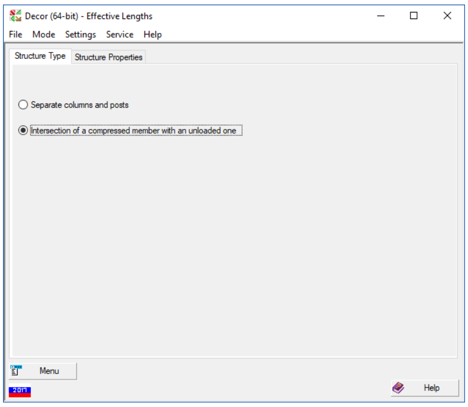


Figure 12.3.3-1. *The* **Structure Type** *tab of the* **Effective Lengths** *dialog box*

This mode implements recommendations of Sec. 4.21 of SNiP II-25-80, KMK 2.03.08.98 (Sec. 6.23 of SP 64.13330.2011, Sec. 7.23 of SP 64.13330.2017) for the determination of the effective lengths. Once this mode is invoked the **Structure Type** tab appears (Fig. 12.3.3-1), where you have to select the type of structure you want the effective length to be calculated for (two structure types are implemented: *Separate columns and posts* and *Intersection of a compressed member with an unloaded one*). The second tab of this mode depends on the selected radio button (Fig. 12.3.3-2, 12.3.3-3).

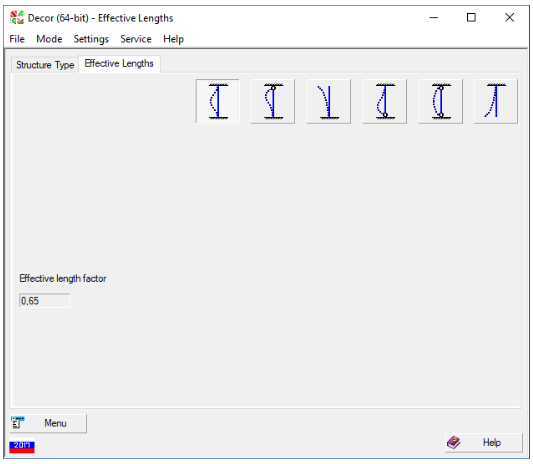


Figure 12.3.3-2. *The* **Effective Lengths** *tab of the* **Effective Lengths** *dialog box*

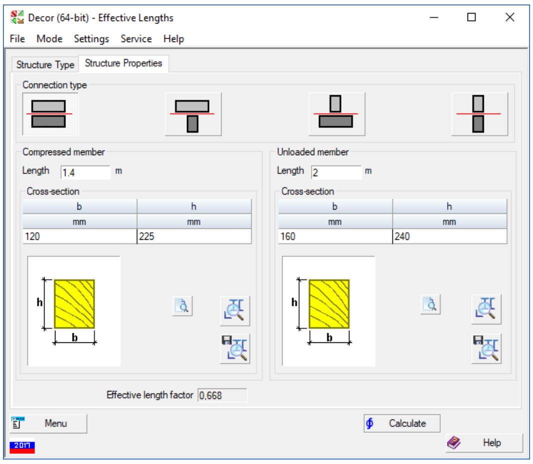


Figure 12.3.3-3. *The* **Structure Properties** *tab of the* **Effective Lengths** *dialog box*

For *separate columns and posts*, the **Effective Lengths** tab has a group of buttons corresponding to particular boundary conditions. The result of the analysis defined by Sec. 4.21 of SNiP II-25-80, KMK 2.03.08.98, Sec. 6.23 of SP 64.13330.2011 or Sec. 7.23 of SP 64.13330.2017 (the ratio of the effective length to the geometric length of the member) will be displayed in the **Effective length factor** field once you click the respective button.

For *intersection of a compressed member with an unloaded one*, you have to specify sizes of the cross-sections of the members, their lengths, and the way they are connected in the **Structure Properties** tab. The effective length factor will be calculated once you click the **Calculate** button.

12.3.4 Resistance of Connections

This mode enables to determine the utilization factors of restrictions for notched connections and connections with cylindrical dowels. The dialog box of this mode contains two tabs. The **General** tab (Fig. 12.3.4-1) is used to select the type of connection — either a notched connection or a connection with cylindrical dowels. This tab also contains standard groups of controls for entering information on timber and service factors. If the notched connection is selected, you also have to specify the information about the section.

a *b*

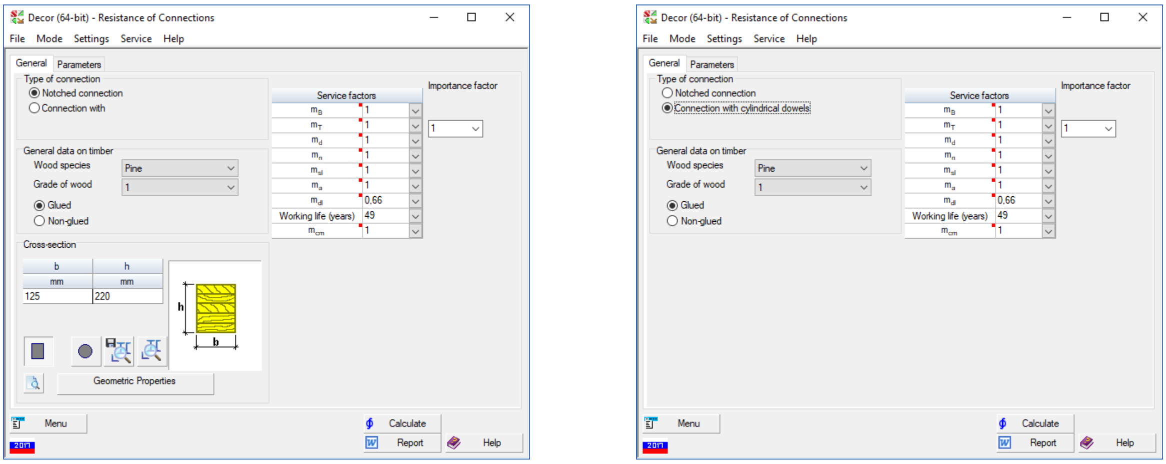


Figure 12.3.4-1. *The* **General** *tab of the* **Resistance of Connections** *dialog box*

Depending on the considered type of connection, the following data has to be specified in the **Parameters** tab:

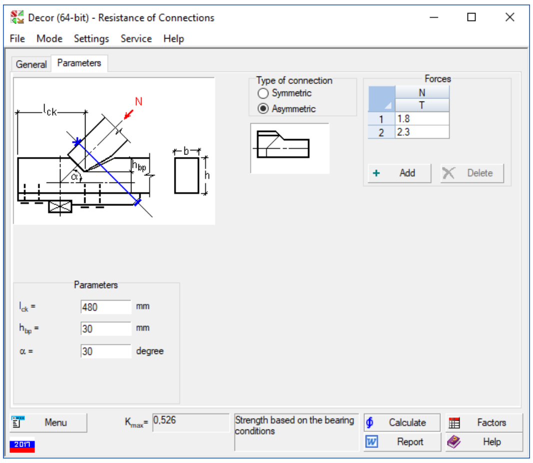
a) for a notched connection (Fig. 12.3.4-2, *a*):

- parameters of the notch;
- type of the connection (either symmetric or asymmetric);
- forces;

b) for a connection with cylindrical dowels (Fig. 12.3.4-2, *b*):

- type of the dowel (nail, steel, aluminum, fiberglass, oak);
- arrangement of the dowels (in-line, staggered, oblique);
- number of design seams for one dowel;
- diameter of the dowel;
- number of dowels and their geometric arrangement;
- forces.

a



b

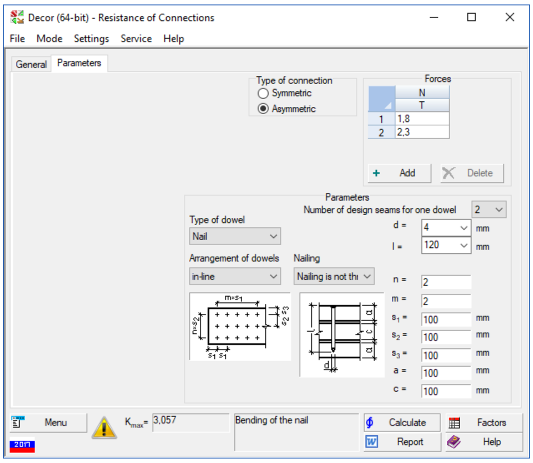


Figure 12.3.4-2. *The* **Parameters** *tab of the* **Resistance of Connections** *dialog box*

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

The notched connections are checked for:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017
Strength based on the bearing conditions	Sec. 5.2	Sec. 7.2	Sec. 8.2
Strength based on the shearing conditions	Sec. 5.2	Sec. 7.2	Sec. 8.2

The connections with cylindrical dowels are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017
Load-bearing capacity for bearing of the side member	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Load-bearing capacity for bearing of the middle member	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Bending of the steel dowel	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017
Bending of the nail	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Bending of the fiberglass dowel	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Bending of the aluminum dowel	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Load-bearing capacity for bearing of the thicker member	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Load-bearing capacity for bearing of the thinner member	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16
Bending of the oak dowel	Sec. 5.13-5.15	Sec. 7.13-7.15	Sec. 8.13-8.16

12.3.5 Resistance of Sections

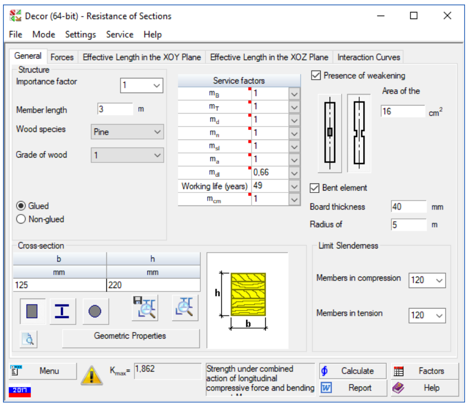


Figure 12.3.5-1. *The* **General** *tab of the* **Resistance of Sections** *dialog box*

This mode enables to determine the load-bearing capacity of any cross-section available in the application database. In the general case, the calculations are performed for the action of a longitudinal force, bending moments, and lateral forces in the principal planes of inertia. The whole set of checks for strength, stability and slenderness is implemented in compliance with the selected design code.

The section is checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Slenderness of the member in the XOY plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Slenderness of the member in the XOZ plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Strength of the member under a longitudinal tensile force	Sec. 4.1	Sec. 6.1	Sec. 7.1		
Strength of the member under a longitudinal compressive force	Sec. 4.2	Sec. 6.2	Sec. 7.2		
Stability in the XOZ plane under **a** longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability in the XOY plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength of the member under the bending moment M_y	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength of the member under the bending moment M_z	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under combined action of M_y and M_z	Sec. 4.12	Sec. 6.12	Sec. 7.12	Sec. 9.4.3	Sec. 6.1.6
Strength under combined action of longitudinal force and bending moment M_z	Sec. 4.16, 4.17	Sec. 6.16, 6.17	Sec. 7.16, 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4
Strength under combined action of longitudinal force and bending moment M_y	Sec. 4.16, 4.17	Sec. 6.16, 6.17	Sec. 7.16, 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4.
Strength under the lateral force Q_z	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Strength under the lateral force Q_y	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Stability of in-plane deformation	Sec. 4.14, 4.15, 4.18	Sec. 6.14, 6.15, 6.20	Sec. 7.14, 7.15, 7.20	Sec. 9.4.4, 9.6.2, 9.6.3	Sec. 6.2.4, 6.3.3
Strength of the member under a torsion moment M_x				Sec. 9.7.1	Sec. 6.1.8

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

The **General** tab (Fig. 12.3.5-1) is used to specify information on the cross-section (see Section 7.3.1), timber (see Section 12.3.1), service factors (see Section 12.3.1), and values of the limit slenderness. Moreover, the behavior of weakened sections can be analyzed as well (the **Presence of weakening** checkbox). If the latter checkbox is checked, a type of weakening and its area have to be specified.

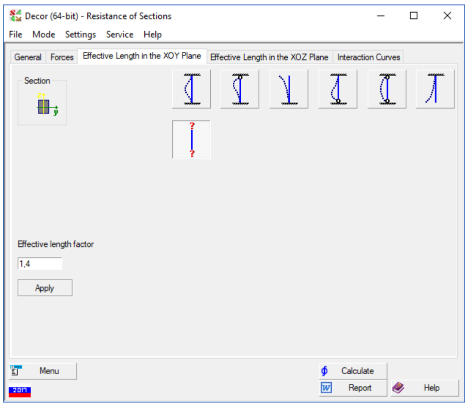
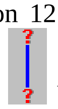


Figure 12.3.5-2. *The* **Effective Length** *tab of the* **Resistance of Sections** *dialog box*

The **Effective Length in the XoY (XoZ) Plane** tabs are the exact replicas of the **Effective Lengths** tab for the case of **Separate columns and posts** from the **Effective Lengths** mode. They suggest six possible conditions of end support for a compressed bar member, which differ from one another in combinations of the boundary conditions (free end, hinge, clamped). This dialog box is described in Section 12.3.3. Unlike the **Effective Lengths** mode, this dialog has the  button (Fig. 12.3.5-2), clicking which enables you to enter any desired values for the effective length factor and confirm your choice by clicking the **Apply** button. In all other cases this field is inaccessible.

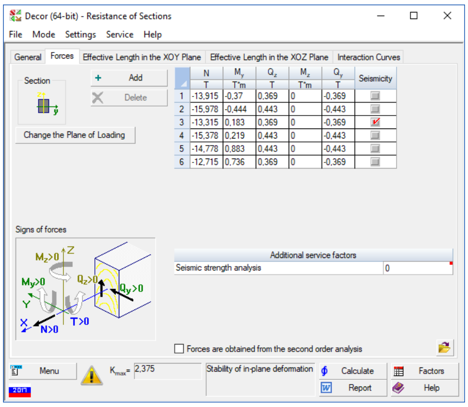


Figure 12.3.5-3. *The* **Forces** *tab of the* **Resistance of Sections** *dialog box*

The **Forces** tab (Fig. 12.3.5-3) is used to specify forces acting in the cross-section of the member. It displays a cross-section with the principal axes of inertia and the positive directions of forces. The tab contains a table for specifying the forces acting in the section from one or more load cases. The number of rows in the table corresponds to the number of the load cases. The table can be also filled by importing the data from **SCAD** which describe the design combinations of forces (DCF). A file with the **.rsu2** extension is created in the **Element Information** mode of the **SCAD** software and then can be imported into **Decor** by clicking the button .

In this mode, the forces acting in the section of the member are specified on the basis of a static analysis performed externally, therefore you have to indicate the type of the design model used to calculate the forces — 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). In the case of a nonlinear analysis the **Forces are obtained from the second order analysis** checkbox should be checked.

The **Seismic** checkbox can be checked for some loadings. In this case, requirements of the respective code (selected in the main window) on the use of the additional service factor at the construction in seismic regions will be automatically taken into account. Moreover, a special table will appear in this dialog box where you can specify the coefficient allowing for seismic action at the strength analysis. If a zero value is specified, the value is taken in accordance with the respective seismic codes by default.

The **Resistance of Sections** mode enables to change the load plane (the **Change the Load Plane** button). Clicking this button will transfer the values of M_y and Q_y to the respective columns of the table for M_z and Q_z, and vice versa.

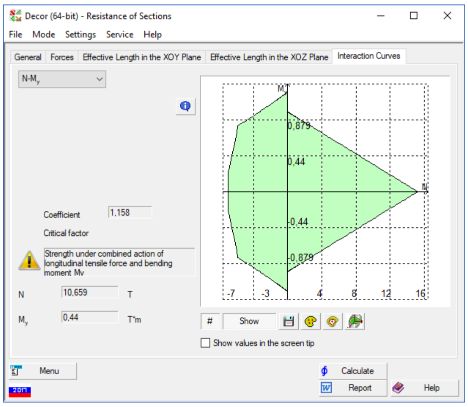


Figure 12.3.5-4. *The* **Interaction Curves** *tab of the* **Resistance of Sections** *dialog box*

The curves enclosing an area of the section load-bearing capacity under various pairs of forces which can arise in the considered section are plotted in the **Interaction Curves** tab (Fig. 12.3.5-4).

The curves surround the coordinate origin by a closed line inside which there are points with conditionally acceptable pairs of the considered forces. A pair of forces is deemed acceptable when $K_{max} \leq 1$. All other forces are taken as zero.

Using your mouse pointer, you can explore the area of the forces variation shown in the graph. Every position of the pointer corresponds to a pair of numerical values of the acting forces; their values are displayed in the respective fields.

Since the slenderness factors do not depend on the forces, they are not calculated when plotting the interaction curves.

The dialog also displays the maximum value of the utilization factor of restrictions that corresponds to these forces and the type of check in which it takes place. When the pointer is placed over a point where $K_{max} > 1$, a warning sign is displayed .

Clicking the right mouse button will display the list of performed checks and values of the factors for the set of forces corresponding to the position of the pointer on the interaction curve.

Limitations

1. The design modulus of the weakened section is taken as the gross section modulus (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. In the analysis of stability of in-plane deformation the k_F factor is taken as 1.13.
3. The power *n* in formula (33) of SNiP, KMK (formula (38) of SP 64.13330.2011, formula (44) of SP 64.13330.2017) is taken as 1.
4. The analysis of stability of in-plane deformation for members under combined bending and compression is performed for a rectangular section only, because formula (33) of SNiP, KMK (formula (38) of SP 64.13330.2011, formula (44) of SP 64.13330.2017) includes the φ_M factor calculated by formula (23) of SNiP, KMK (formula (25) of SP 64.13330.2011, formula (31) of SP 64.13330.2017). The latter formula is applicable only for rectangles.
5. In the analysis of stability of in-plane deformation for beams of a variable rectangular cross-section, the k_{GM} factor is taken as 1 because, when the number of out-of-plane restraints is $n \geq 4$, SNiP, KMK (Sec. 4.14, Sec. 6.14 of SP 64.13330.2011, Sec. 7.14 of SP 64.13330.2017) requires that k_{GM}=1, and for other values of *n* it refers to Table 2 of Annex 4 of SNiP (Table F.2 of Annex F of SP) where nothing is said about what should be done in the case of an arbitrary moment diagram.
6. When analyzing the strength of members under eccentric tension or combined bending and tension by formula (27) of SNiP, KMK (formula (29) of SP 64.13330.2011, formula (35) of SP 64.13330.2017), the value of M_d is calculated by formulas (29), (30) of SNiP (formulas (31), (32) of SP 64.13330.2011, formulas (37), (38) of SP 64.13330.2017).

12.3.6 Continuous Purlins

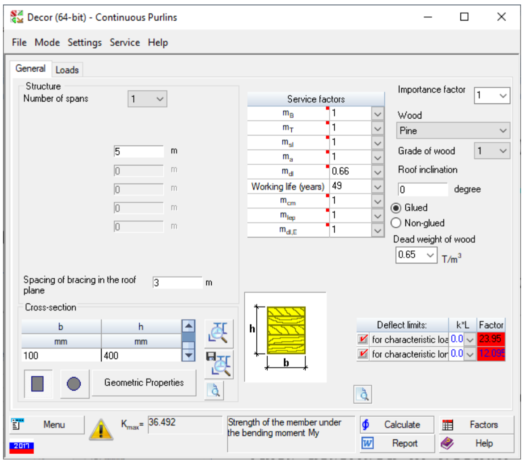


Figure 12.3.6-1. *The* **General** *tab of the* **Continuous Purlins** *dialog box*

This mode enables to perform checks of continuous girders of a round or rectangular section. The dialog box of this mode (Fig. 12.3.6-1) contains two tabs: **General** and **Loads**.

The **General** tab is used to specify the number of spans and their lengths. You also have to specify (according to the rules described in Section 12.3.1) information on the cross-section, timber, and service factors. The roof inclination can also be specified here (pitch force component will be taken into account). If it is necessary to perform the analysis for the serviceability limit state, check the **Deflection limitation** checkboxs (for characteristic and/or characteristic long-term loads) and select (or enter) the maximum ratio of the maximal deflection to the length of the span in the respective drop-down list.

The **Loads** tab (Fig. 12.3.6-2) is used to specify the loads acting on the purlin. The application is capable of analyzing several patterns of loading and each load case can in its turn consist of multiple loads.

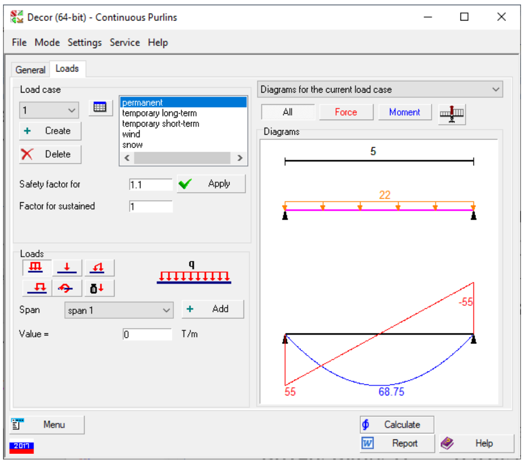


Figure 12.3.6-2. *The* **Loads** *tab of the* **Continuous Purlins** *dialog box*

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Load case** group;
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors according to SNiP to be used with the loads of this load case in a combination of loads;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;

click the **Add** button.

A few load components can be specified for each load case. It is assumed that the *design* values of loads are entered.

Depending on the load type, its parameters may include:

- for distributed loads — the load intensity;
- for a distributed load on a part of the span — the intensity of the load, its position and width of application;
- for a concentrated force — the value of the force and its position in the span;
- for a concentrated moment — the value of the moment and its position in the span;
- for a trapezoid load — value of the load at the beginning of the application area, distance from the beginning of the bar to the beginning of the application area, width of the load application area, value of the load at the end of the application area;
- no additional data is required to specify the dead weight.

For each load (except for the dead weight) you have to specify a span to which this load is applied (it is selected in the **Span** drop-down list).

The **Delete** button is used to delete a load case (not a separate load included in it).

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Load case** list.

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** field with the superimposed diagrams of the bending moments and shear forces underneath (Fig. 12.3.6-2).

Once you have entered all the loadings, you can view the envelopes of moments and their respective shear forces, or the envelopes of shear forces and their respective bending moments. The criterion (maximal/minimal moment, maximal/minimal shear force) can be selected from the drop-down list. The envelope diagrams are plotted according to SNiP 2.01.07-85* “Loads and actions”.

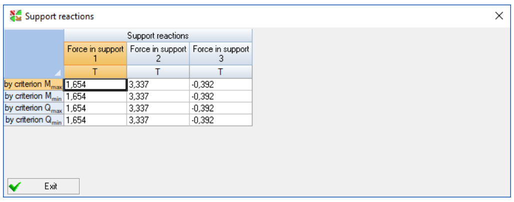
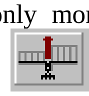


Figure 12.3.6-3. *The* **Support reactions** *dialog box*

The **All**, **Force**, and **Moment** buttons enable you to select a mode for displaying the diagrams: diagrams of moments and shear forces together, only shear forces, and only moments respectively. Furthermore, clicking the button  will display the design values of support reactions (Fig. 12.3.6-3).

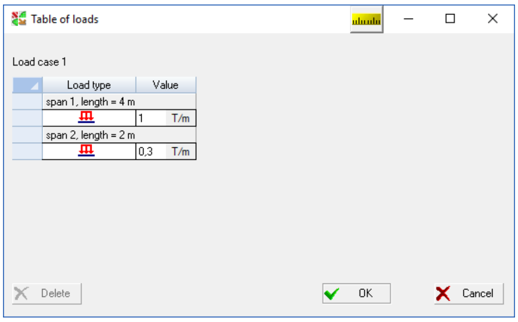


Figure 12.3.6-4. *The* **Table of loads** *dialog box*

If you place the mouse pointer in the diagram field, values of the moment and shear force in a particular cross-section corresponding to the position of the pointer will be displayed.

If it is necessary to perform the analysis for the serviceability limit state, you should check whether the safety factor for load is specified correctly for each load case, because the calculation of deflections is based on the characteristic load values.

If you want to change the load value or delete the load from a load case, use the table of loads (the button in the **Load case** group). The **Table of loads** dialog box (Fig. 12.3.6-4), which opens by clicking this button, displays the type, the value, and the position of the load. Changes made to the load parameters will be saved after you exit the table by clicking the **OK** button.

Calculation

To perform the calculation of the load-bearing capacity utilization factors, click the **Calculate** button.

The purlins are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength of the member under the bending moment M_y	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength of the member under the bending moment M_z	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under the lateral force Q_z	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Strength under the lateral force Q_y	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Strength under combined action of M_y and M_z	Sec. 4.12	Sec. 6.12	Sec. 7.12	Sec. 9.4.3	Sec. 6.1.6
Stability of in-plane deformation	Sec. 4.14	Sec. 6.14	Sec. 7.14	Sec. 9.4.4	Sec. 6.2.4
Deflection	Sec. 4.33	Sec. 6.35	Sec. 7.35		

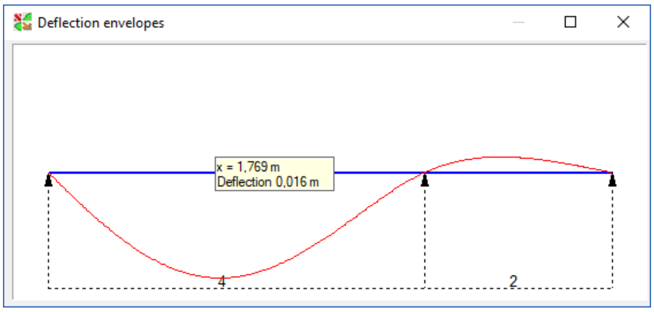


Figure 12.3.6-5. *The* **Deflection envelopes** *dialog box*

If the **Deflection limitation** checkbox is checked in the **General** tab, the deflection envelopes will be calculated. The maximum value of the deflection will be displayed in the **Maximum deflection** group in the **General** tab. The button in the same group invokes a dialog box with deflection diagrams (Fig. 12.3.6-5).

It should be noted that if the roof inclination is not zero, the purlins are bent in two planes. Therefore, the deformed axis of a purlin is a spatial curve. **Bending moment diagrams and deflection envelopes are plotted for the zero roof inclination (!)**. However, the values of the factors for strength of the element under the action of the bending moments My and Mz, the value of the factor for strength at the biaxial bending, and the extreme value of the deflection and the respective factor are calculated precisely taking into account the roof inclination specified by the user.

Limitations

Formula (50) of SNiP, KMK (formula (55) of SP 64.13330.2011, formula (61) of SP 64.13330.2017) is not used when calculating the maximum deflection because purlins are assumed to have a constant height, and Table 3 of Annex 4 of SNiP, KMK (Table F.3 of Annex F of SP 64.13330.2011, Table F.4 of Annex F of SP 64.13330.2017) requires that k=1, c=0 (the first line in the table is the most frequently used case).

12.3.7 Cantilever Purlins

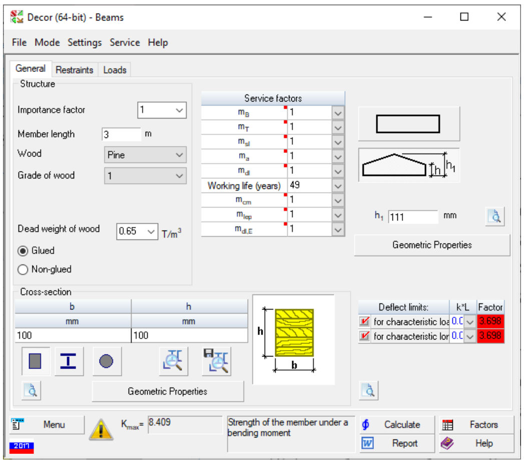


Figure 12.3.7-1. *The* **General** *tab of the* **Cantilever Purlins** *dialog box*

This mode is similar to the **Continuous Purlins** mode**.** The differences are that in the **General** tab (Fig. 12.3.7-1) you only have to specify the length of a standard span (all standard spans are assumed to have the same length), and you can specify the arrangement of hinges to provide equal moments or equal deflections. If the **other** option is selected, you have to specify the distance from the support to the hinge.

It should be noted that this mode can be used to analyze simply supported purlins if you specify the zero distance between the support and the hinge.

Only two types of loads can be specified for cantilever purlins: a uniformly distributed load on all spans, and the dead weight.

The purlins are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength of the member under the bending moment M_y	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength of the member under the bending moment M_z	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under the lateral force Q_z	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Strength under the lateral force Q_y	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength under combined action of M_y and M_z	Sec. 4.12	Sec. 6.12	Sec. 7.12	Sec. 9.4.3	Sec. 6.1.6
Stability of in-plane deformation	Sec. 4.14	Sec. 6.14	Sec. 7.14	Sec. 9.4.4	Sec. 6.2.4
Deflection	Sec. 4.33	Sec. 6.35	Sec. 7.35		

Limitations

Formula (50) of SNiP, KMK (formula (55) of SP 64.13330.2011, formula (61) of SP 64.13330.2017) is not used when calculating the maximum deflection because purlins are assumed to have a constant height, and Table 3 of Annex 4 of SNiP, KMK (Table F.3 of Annex F of SP 64.13330.2011, Table F.4 of Annex F of SP 64.13330.2017) requires that k=1, c=0 (the first line in the table is the most frequently used case).

The analysis is based on the assumption that the end span has a recommended (see [1]) length (~ 0.85 of the standard span length). The maximum values of bending moments, shear forces, and deflections cannot appear in the end span, therefore it is excluded from the analysis.

12.3.8 Beams

This multitab dialog box (Fig. 12.3.8-1) enables to perform checks of regular or double tapered beams. The dialog box contains three tabs: **General**, **Restraints**, **Loads**.

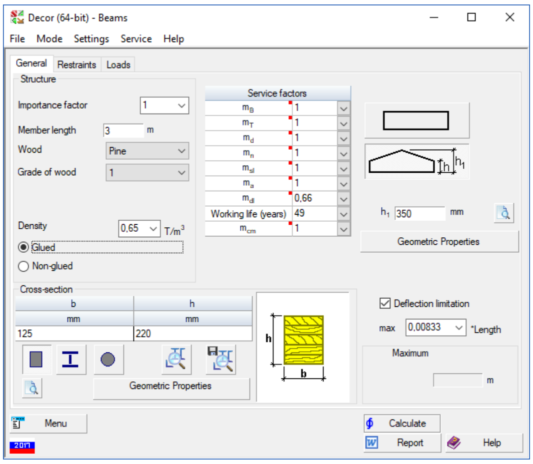


Figure 12.3.8-1. *The* **General** *tab of the* **Beams** *dialog box*

This mode is similar to the **Continuous Purlins** mode. The differences are that only one length of a beam is specified and if the section is rectangular, a double tapered beam can be analyzed (the height of the section in the middle of the span is specified additionally in this case).

The **Restraints** tab (Fig. 12.3.8-2) contains two groups of buttons for specifying a system of beam restraints in and out of the bending plane. The selection within each group is performed independently by clicking a respective button. If the last model of restraints out of the bending plane is selected, a field for specifying the number of segments of the beam span will appear.

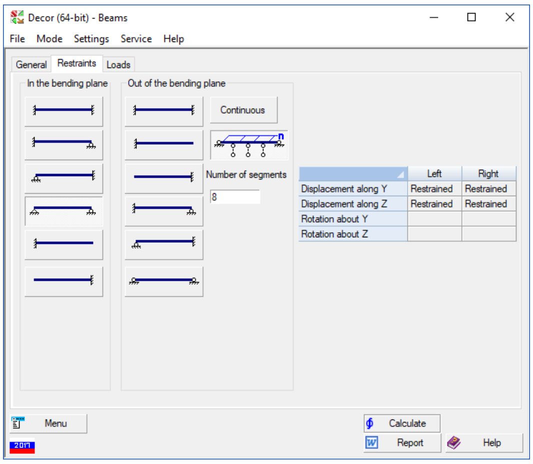


Figure 12.3.8-2. *The* **Restraints** *tab of the* **Beams** *dialog box*

The check of the entered initial data is performed with the help of a table displaying the selected system of restraints.

The loads are specified in the same way as in the **Continuous Purlins** mode (you do not have to specify the number of a span to which the load is applied).

The beams are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength of the member under a bending moment	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength of the member under a lateral force	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Stability of in-plane deformation	Sec. 4.14, 4.15	Sec. 6.14, 6.15	Sec. 7.14, 7.15	Sec. 9.4.4	Sec. 6.2.4
Deflection	Sec. 4.33	Sec. 6.35	Sec. 7.35		

Limitations

When calculating the maximum deflection for beams of a rectangular section, factors k and c in formula (50) of SNiP, KMK (formula (55) of SP 64.13330.2011, formula (61) of SP 64.13330.2017) are calculated according to line 4 of Table 3 of Annex 4 of SNiP, KMK (Table F.3 of Annex F of SP 64.13330.2011, Table F.4 of Annex F of SP 64.13330.2017), i.e.:

k=0.15+0.85β;

c=15.4+3.8β.

Line 5 of Table 3 of Annex 4 of SNiP, KMK (Table F.3 of Annex F of SP 64.13330.2011, Table F.4 of Annex F of SP 64.13330.2017) is used for I-beams, i.e.:

k=0.4+0.6β;

c=(45.3+6.9β)γ.

12.3.9 Columns

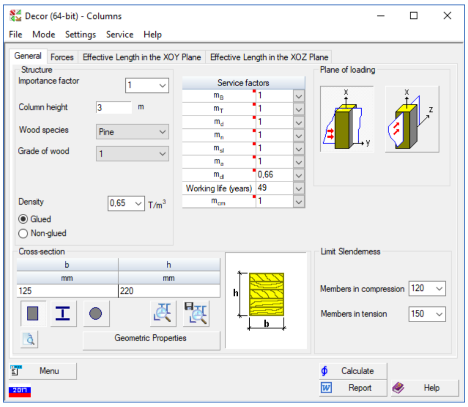


Figure 12.3.9-1. *The* **General** *tab of the* **Columns** *dialog box*

This mode enables to perform checks of posts and columns. The whole set of checks for strength, stability, and slenderness is implemented in compliance with the selected design code. A planar loading is assumed.

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

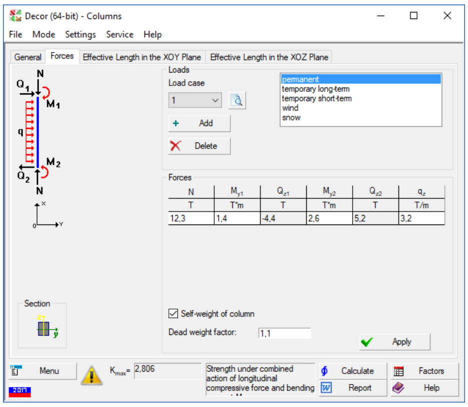


Figure 12.3.9-2. *The* **Forces** *tab of the* **Columns** *dialog box*

The **General** tab (Fig. 12.3.9-1) contains a text field for entering the column height and two buttons for selecting the load plane (an orientation of the deformation plane). You also have to specify (according to the rules described in Section 7.3) information on the cross-section, timber, and service factors. Moreover, it is necessary to specify (by entering or selecting from the drop-down list) the limit slenderness values (for members in compression and for members in tension).

The **Forces** tab (Fig. 12.3.9-2) is used to specify all the loads for each load case simultaneously.

General equilibrium conditions must be satisfied for these forces and moments. In particular, lateral forces Q_1 and Q_2, as well as nodal moments M_1 and M_2 have to be taken from the results of the calculation of the system as a whole. The equilibrium conditions are as follows:

$$Q_1 - Q_2 + qL = 0,$$

$$M_2 - M_1 - Q_1L - qL^2/2 = 0.$$

Note that in this mode the positive longitudinal force corresponds to compression.

It should be noted that all the loads act in the XOY plane or in the XOZ plane (the X axis is oriented along the bar). The plane is selected in the **General** tab.

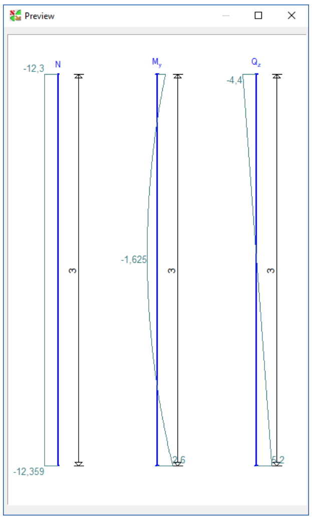


Figure 12.3.9-3. *The* **Preview** dialog box

Clicking the button  will open the **Preview** dialog box with diagrams of N, M_y and Q_z (Fig. 12.3.9-3).

The **Effective Length in the XoY Plane** and **Effective Length in the XoZ Plane** tabs are equivalent to those described in Section 12.3.5.

The columns are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Slenderness of the member in the XOY plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Slenderness of the member in the XOZ plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Strength of the member under a longitudinal tensile force	Sec. 4.1	Sec. 6.1	Sec. 7.1	Sec. 9.2.1	Sec. 6.1.2
Strength of the member under a longitudinal compressive force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.1	Sec. 6.1.4
Stability in the XOZ plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability in the XOY plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Strength of the member under the bending moment M_y	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength of the member under the bending moment M_z	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under combined action of M_y and M_z	Sec. 4.12	Sec. 6.12	Sec. 7.12	Sec. 9.4.3	Sec. 6.1.6
Strength under combined action of longitudinal force and bending moment M_z	Sec. 4.16, 4.17	Sec. 6.16, 6.17	Sec. 7.16, 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4
Strength under combined action of longitudinal force and bending moment M_y	Sec. 4.16, 4.17	Sec. 6.16, 6.17	Sec. 7.16, 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4
Strength under the lateral force Q_z	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Strength under the lateral force Q_y	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7
Stability of in-plane deformation	Sec. 4.14, 4.15, 4.18	Sec. 6.14, 6.15, 6.20	Sec. 7.14, 7.15, 7.20	Sec. 9.4.4, 9.6.2, 9.6.3	Sec. 6.2.4, 6.3.3

Limitations

1. In the analysis of stability of in-plane deformation the k_F factor is taken as 1.13.
2. The power *n* in formula (33) of SNiP, KMK (formula (38) of SP 64.13330.2011, formula (44) of SP 64.13330.2017) is taken as 1.
3. The analysis of stability of in-plane deformation for members under combined bending and compression is performed for a rectangular section only, because formula (33) of SNiP, KMK (formula (38) of SP 64.13330.2011, formula (44) of SP 64.13330.2017) includes the φ_M factor calculated by formula (23) of SNiP

(formula (25) of SP 64.13330.2011, formula (31) of SP 64.13330.2017). The latter formula is applicable only for rectangles.

4. In the analysis of stability of in-plane deformation for columns of a variable rectangular cross-section, the k_{GM} factor is taken as 1 because, when the number of out-of-plane restraints is $n \geq 4$, SNiP, KMK (Sec. 4.14, Sec. 6.14 of SP 64.13330.2011, Sec. 7.14 of SP 64.13330.2017) requires that k_{GM}=1, and for other values of *n* it refers to Table 2 of Annex 4 of SNiP (Table F.2 of Annex F of SP) where nothing is said about what should be done in the case of an arbitrary moment diagram.

5. When analyzing the strength of members under eccentric tension or combined bending and tension by formula (27) of SNiP, KMK (formula (29) of SP 64.13330.2011, formula (35) of SP 64.13330.2017), the value of M_d is calculated by formulas (29), (30) of SNiP, KMK (or formulas (31), (32) of SP 64.13330.2011, formulas (37), (38) of SP 64.13330.2017).

12.3.10 Trusses

This mode enables to perform all necessary checks of truss members for strength and stability, and it also checks their slenderness. The work begins with calculating the design values of the forces caused by the given external loads in structural designs most frequently used in practice.

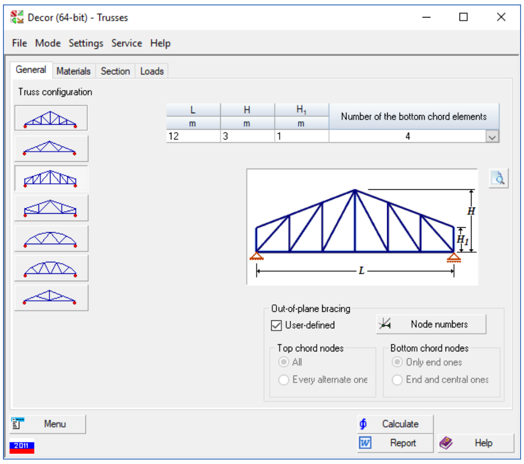


Figure 12.3.10-1. *The* **General** *tab of the* **Trusses** *dialog box*

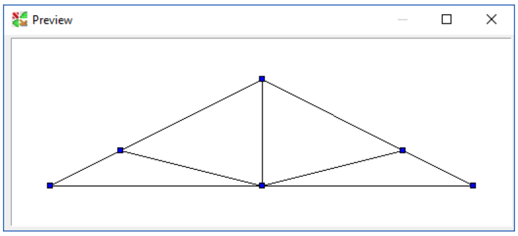
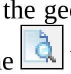


Figure 12.3.10-2. *The* **Preview** *dialog box*

This mode is used to perform the analysis of timber trusses. If you have to analyze composite steel-and-timber trusses, you can use the following procedure. First, you have to specify the truss configuration and the loads. All the considered trusses are statically determinate, therefore the forces in their members do not depend on the material. Every timber member can be checked for compliance with SNiP II-25-80, KMK 2.03.08.98 or SP 64.13330 using **DECOR**, and steel members can be checked using **KRISTALL**.

The initial data are specified in the four tabs of this mode: **General**, **Materials**, **Section**, and **Loads**.

The **General** tab (Fig. 12.3.10-1) contains a group of buttons for selecting the truss configuration. All trusses are statically determinate and are assumed to be fixed in the end nodes of their bottom chord in a statically determinate way according to a beam scheme.

Specify the span of the truss, its height on the support and in the middle, and the number of elements of the bottom chord panels for the selected configuration. Having entered all the geometric sizes, you can view the given truss in the **Preview** dialog box (Fig. 12.3.10-2), which is invoked by the  button.

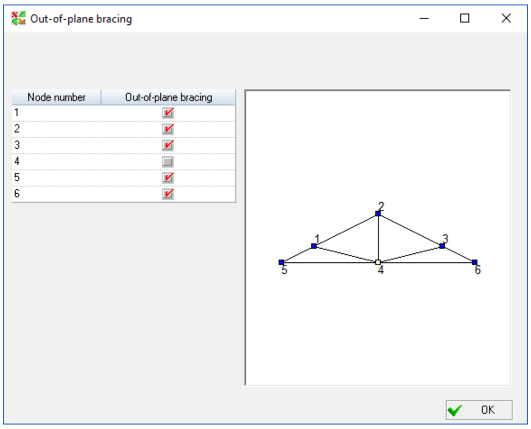


Figure 12.3.10-3. *The* **Out-of-plane bracing** *dialog box*

The **Out-of-plane bracing** group has checkboxes and radio buttons which are used to specify the method of out-of-plane bracing of the top and bottom chord nodes (the bracing in the truss plane is assumed to be statically determinate: a hinge support for the left support node and a roller support for the right one).

You can also specify a nonstandard arrangement of the out-of-plane bracing. To do it, check the **User-defined** checkbox and click the  button to open a dialog box with a table (Fig. 12.3.10-3). The braced nodes are highlighted in blue in the model.

The **Materials** tab (Fig. 12.3.10-4) is used to specify information on timber and service factors.

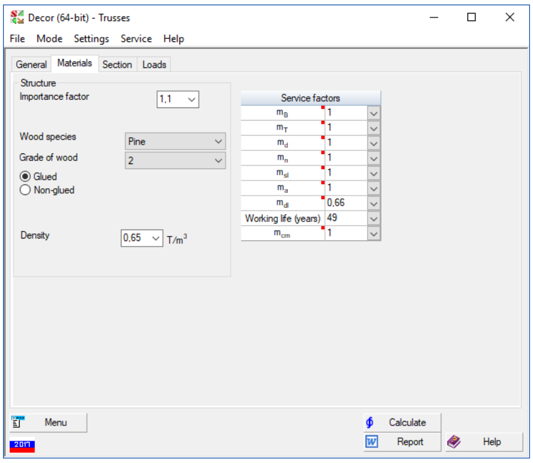


Figure 12.3.10-4. *The* **Materials** *tab of the* **Trusses** *dialog box*

The **Section** tab (Fig. 12.3.10-5) is used to specify the sizes of cross-sections for every group of the truss members (the top chord, the bottom chord, diagonals, verticals, and support diagonals.

The **Loads** tab (Fig. 12.3.10-6) is much similar to that of the **Continuous Purlins** mode (see Section 12.3.6). However, there are some differences. Firstly, only a uniformly distributed load or a concentrated load is allowed. Secondly, the location of the concentrated force is defined by specifying the node number. The distributed load can be specified simultaneously for a group of members selected by the user: the top/bottom chord or the left/right side of the chord.

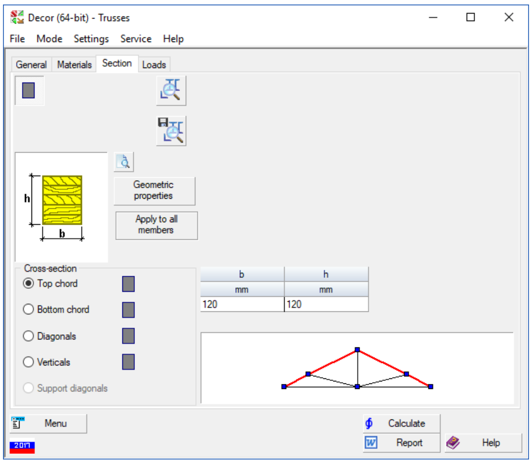


Figure 12.3.10-5. *The* **Section** *tab of the* **Trusses** *dialog box*

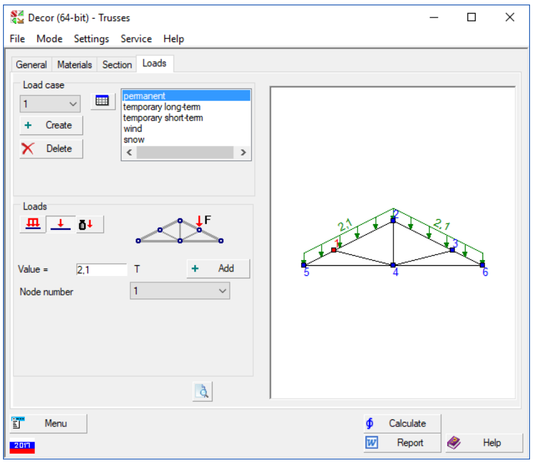


Figure 12.3.10-6. *The* **Loads** *tab of the* **Trusses** *dialog box*

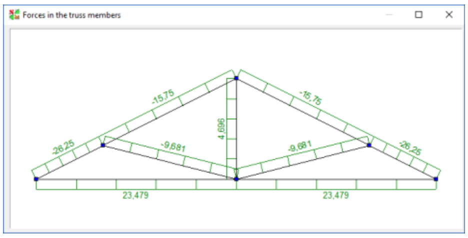


Figure 12.3.10-7. *The* **Forces in the truss members** *dialog box*

Forces in the truss members for the current load case are displayed as diagrams in the **Forces in the truss members** dialog box, which can be invoked by clicking the  button (Fig. 12.3.10-7).

The trusses are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Slenderness of the member out of the truss plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Slenderness of the member in the truss plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Strength of the member under a longitudinal tensile force	Sec. 4.1	Sec. 6.1	Sec. 7.1	Sec. 9.2.1	Sec. 6.1.2
Strength of the member under a longitudinal compressive force	Sec. 4.2	Sec. 6.1	Sec. 7.1	Sec. 9.3.1	Sec. 6.1.4
Stability in the truss plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability out of the truss plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Rigidity of truss	Sec. 4.33	Sec. 6.35	Sec. 7.35		

Decor (unlike SCAD) assumes that a truss is subjected to a nodal load. Thus the specified distributed load is not transferred onto the truss members; instead, it is assumed to be applied to some enclosing roof structure which performs the function of transferring the load onto the truss nodes.

As a rule, the top truss chord is subjected to a distributed load, and this fact is taken into account in the truss analysis performed by SCAD. The longitudinal force changes over the length in inclined members. SCAD calculates and displays maximum value of the force in the member. When converting the distributed load into the nodal loads, Decor will display a value corresponding to the force in the middle of a SCAD finite element. Therefore the results generated by the two applications may differ.

12.3.11 Truss Member

The **Trusses** mode described above provides the most common but not complete set of truss structures. To analyze members of an arbitrary truss, the application contains a **Truss Member** mode which enables to analyze the load-bearing capacity of a truss member (it is assumed that the static problem has been solved previously and the forces in the member are known).

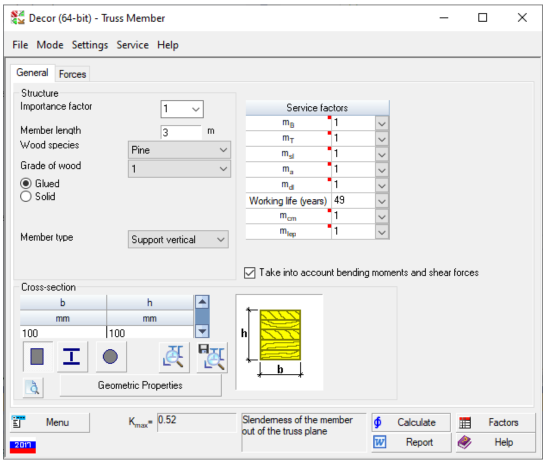


Figure 12.3.11-1. *The* **Truss Member** *dialog box*

The **General** tab (Fig. 12.3.11-1) is used to specify information on the cross-section of the member, its type (a chord member, a lattice member, ...), timber, and service factors. A separate table is used to specify forces (longitudinal forces) for any number of load cases.

Truss members are calculated for longitudinal forces by default. A special checkbox enables to take into account the bending moments and shear forces as well. In this case the calculation is similar to that performed in the **Resistance of Sections** mode, but the values of effective lengths are selected automatically in accordance with the recommendations of the codes for truss members.

A truss member is checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Slenderness of the member out of the truss plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Slenderness of the member in the truss plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Strength of the member under a longitudinal tensile force	Sec. 4.1	Sec. 6.1	Sec. 7.1	Sec. 9.2.1	Sec. 6.1.2
Strength of the member under a longitudinal compressive force	Sec. 4.2	Sec. 6.1	Sec. 7.1	Sec. 9.3.1	Sec. 6.1.4
Stability in the truss plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability out of the truss plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2

12.3.12 Arches

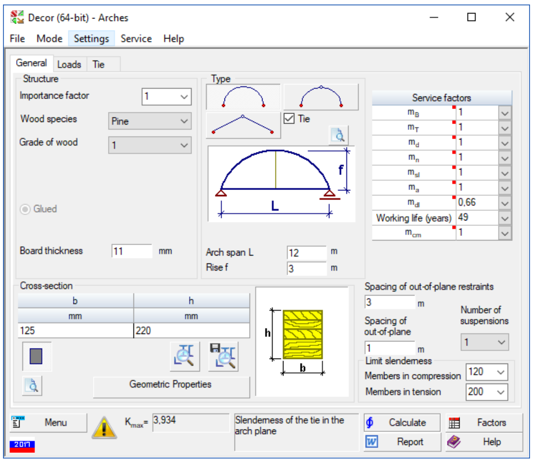


Figure 0-1. *The* **General** *tab of the* **Arches** *dialog box*

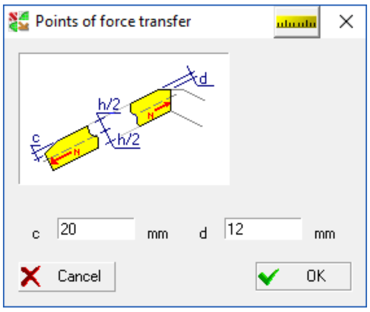


Figure 0-2. *The* **Points of force transfer** *dialog box*

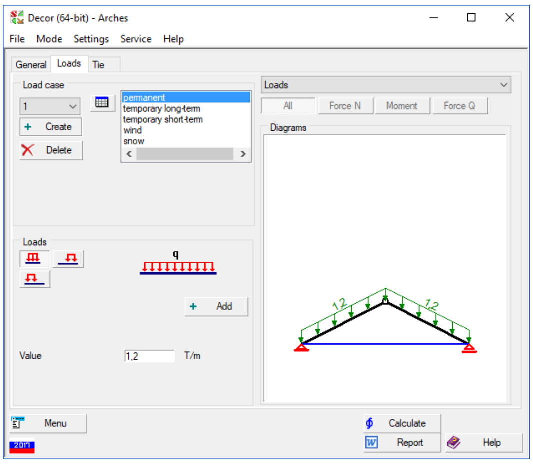


Figure 0-3. *The* **Loads** *tab of the* **Arches** *dialog box*

This mode enables to perform checks of three types of arches from glued laminated timber of a rectangular cross-section — two-hinged and three-hinged round ones, and three-hinged triangular ones. A tie from double angles or circular hollow sections can be specified.

If there is no tie, the **Arches** dialog box will contain two tabs: **General** and **Loads**. If there is a tie, there will also be a **Tie** tab (Fig. 0-1).

The **General** tab is used to specify the following data:

- the type of the arch;
- the presence of a tie;
- the values of the span and the rise;
- the number of suspensions (it defines the spacing of in-plane restraints of the tie);
- the spacing of out-of-plane restraints;
- the spacing of restraints of the tie (if there is a tie);
- limit slenderness for members in compression and for members in tension.

You also have to specify (according to the rules described in Section 3.1) information on the cross-section, timber, importance factors, and service factors. The effective lengths are taken according to Sec. 6.25 of SNiP, KMK (Sec. 8.56 of SP 64.13330.2011, Sec. 9.36 of SP 64.13330.2017).

Artificial eccentricities can be specified for triangular arches in order to reduce the moment in the span. These data are entered in the **Points of force transfer** dialog box (Fig. 0-2), which can be invoked by clicking the **Control of the force transfer** button.

The **Loads** tab (Fig. 0-3) is used to specify the loads acting on the arch. The application is capable of analyzing several patterns of loading and each load case can in its turn consist of multiple loads.

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Load case** group;
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors according to SNiP 2.01.07-85* to be used with the loads of this load case in a combination of loads;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;
- click the **Add** button.

A few load components can be specified for each load case. It is assumed that the design values of loads are entered. You can specify a uniformly distributed load over the whole span, on the right or left half of the arch, or a triangular load on each half of the arch.

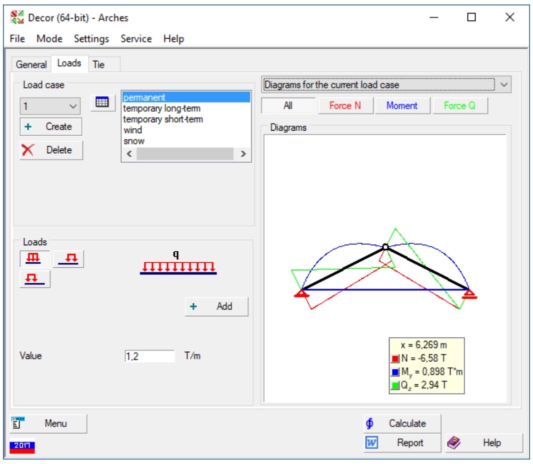


Figure 0-4. *The* **Loads** *tab of the* **Arches** *dialog box* in the diagram plotting mode

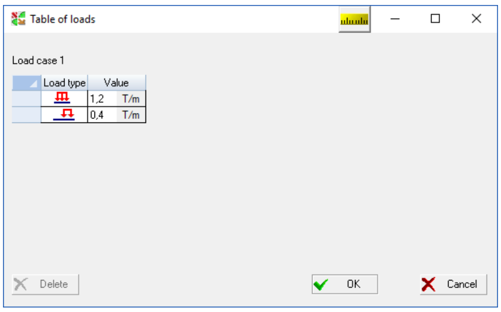


Figure 0-5. *The* **Table of loads** *dialog box*

The **Delete** button is used to delete a load case (not a separate load included in it). Particular loads can be deleted in the **Table of loads** dialog box (Fig. 0-5), which is invoked by clicking the button. To delete a load, follow these steps:

- use the **Load case** drop-down list to select the number of the load case that contains the load you want to delete;
- click the **Table of loads** button;
- select a line (lines) that contains the load(s) you want to delete;
- click the **Delete** button;
- exit the dialog by clicking the **OK** button.

The table of loads can also be used to edit the value of a load. To do it, just change the value in the respective column and exit the dialog box by clicking the **OK** button.

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Load case** list.

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** field. If you select the **Diagrams for the current load case** mode from the list above the design model and click the **All** button, the diagrams of moments, longitudinal and shear forces will be displayed together (Fig. 0-4). The buttons **Force N**, **Moment**, **Force Q** are used to obtain a separate diagram of the respective force, as well as the extreme values of each force.

The check of the arch will be performed once you click the **Calculate** button. The arches are checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Slenderness in the arch plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Slenderness out of the arch plane	Sec. 4.4	Sec. 6.4	Sec. 7.4		
Strength under a longitudinal compressive force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.1	Sec. 6.1.4
Stability in the arch plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability out of the arch plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability of in-plane deformation	Sec. 4.18	Sec. 6.20	Sec. 7.20	Sec. 9.4.1	Sec. 6.1.6
Strength under a bending	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.5.1,	Sec. 6.2.3,

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
moment				9.5.2, 9.6.2	6.2.4
Strength under combined action of longitudinal force and bending moment	Sec. 4.17	Sec. 6.17	Sec. 7.17	Sec. 9.4.4, 9.6.2, 9.6.3	Sec. 6.2.4, 6.3.3
Strength under a lateral force	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7

If there is a tie, the following additional checks will be performed:

Strength of the tie — (Sec. 5.1 of SNiP II-23-81*, Sec. 7.1.1 of SP 16.13330, Sec. 7.1 KMK 2.03.05-97);

Slenderness of the tie in the arch plane — (Sec. 6.16 of SNiP II-23-81*, Sec. 10.4.1 of SP 16.13330, Sec. 8.13 KMK 2.03.05-97);

Slenderness of the tie out of the arch plane — (Sec. 6.16 of SNiP II-23-81*, Sec. 10.4.1 of SP 16.13330, Sec. 8.13 KMK 2.03.05-97).

12.3.13 Rafters

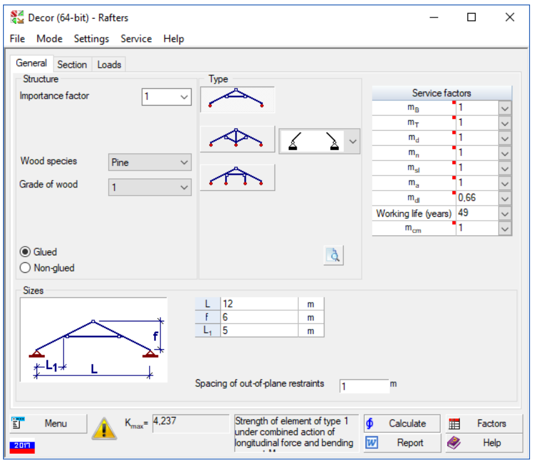


Figure 12.3.13-1. *The* **General** *tab of the* **Rafters** *dialog box*

This mode enables to perform checks of three types of timber rafter structures of a rectangular or round cross-section.

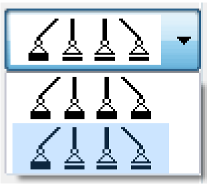
The **Rafters** dialog box contains three tabs: **General, Section**, and **Loads**.

The **General** tab is used to specify the following data:

- the type of the rafter structure;
- geometric sizes;
- the spacing of out-of-plane restraints.

You also have to specify (according to the rules described in Section 12.3.13) information on the timber, importance factors, and service factors. The effective lengths are taken according to Sec. 6.25 of SNiP (Sec. 8.56 of SP).

A separate drop-down list



enables to select one of the variants of the supports.

Principal rafters are given in the **Trusses** section.

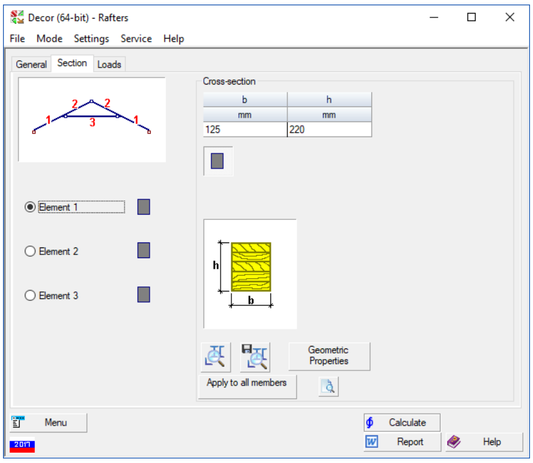


Figure 12.3.13-2. *The* **Section** *tab of the* **Rafters** *dialog box*

The **Section** tab (Fig. 12.3.13-2) is used to specify the type and sizes of cross-sections for every group of the members.

Rafter structures consist of the following members:

1-st type:

- rafters (elements 1, 2);
- tie (element 3);

2-nd type:

- rafters (elements 1, 2);
- struts (element 3);
- vertical (element 4);

3-rd type:

- rafters (elements 1, 2);
- support frame (elements 3, 4).

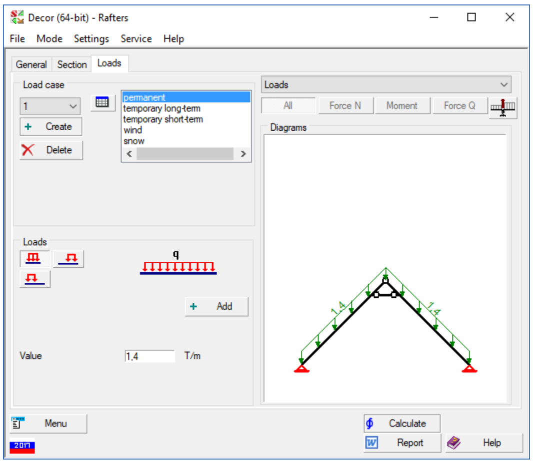


Figure 12.3.13-3. *The* **Loads** *tab of the* **Rafters** *dialog box*

The **Loads** tab (Fig. 12.3.13-3) is used to specify the loads acting on the rafters. The application is capable of analyzing several patterns of loading and each load case can in its turn consist of multiple loads.

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Load case** group;
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors according to SNiP 2.01.07-85* to be used with the loads of this load case in a combination of loads;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;
- click the **Add** button.

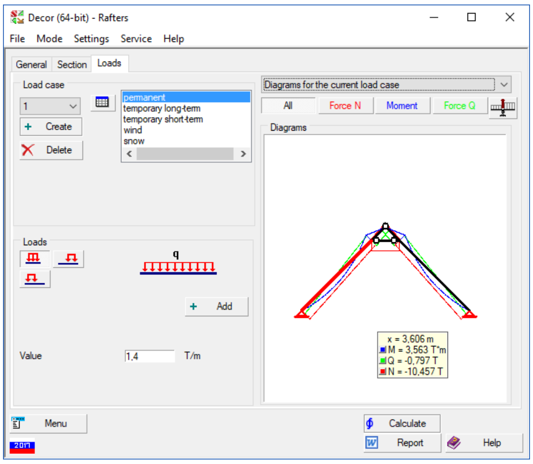


Figure 12.3.13-4. *The* **Loads** *tab of the* **Rafters** *dialog box* in the diagram plotting mode

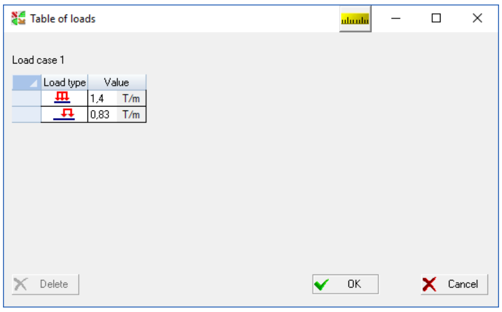
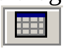


Figure 12.3.13-5. *The* **Table of loads** *dialog box*

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** field. If you select the **Diagrams for the current load case** mode from the list above the design model and click the **All** button, the diagrams of moments, longitudinal and shear forces will be displayed together (Fig. 12.3.13-4). The buttons **Force N**, **Moment**, **Force Q** are used to obtain a separate diagram of the respective force, as well as the extreme values of each force.

A few load components can be specified for each load case. It is assumed that the design values of loads are entered. You can specify a uniformly distributed load over the whole span, on the right or left half of the rafters.

The **Delete** button is used to delete a load case (not a separate load included in it). Particular loads can be deleted in the **Table of loads** dialog box (Fig. 12.3.13-5), which is invoked by clicking the  button. To delete a load, follow these steps:

- use the **Load case** drop-down list to select the number of the load case that contains the load you want to delete;
- click the **Table of loads** button;
- select a line (lines) that contains the load(s) you want to delete;
- click the **Delete** button;
- exit the dialog by clicking the **OK** button.

The table of loads can also be used to edit the value of a load. To do it, just change the value in the respective column and exit the dialog box by clicking the **OK** button.

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Load case** list.

The check of the rafters will be performed once you click the **Calculate** button. Each element of the rafter structure is checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength under a longitudinal compressive force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.1	Sec. 6.1.4
Stability in the rafter plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability out of the rafter plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability of in-plane deformation	Sec. 4.14, 4.18	Sec. 6.14, 6.20	Sec. 7.14, 7.20	Sec. 9.4.4, 9.6.2, 9.6.3	Sec. 6.2.4, 6.3.3

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength under a bending moment	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under combined action of longitudinal force and bending moment	Sec. 4.17	Sec. 6.17	Sec. 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4.
Strength under a lateral force	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7

12.3.14 Frames

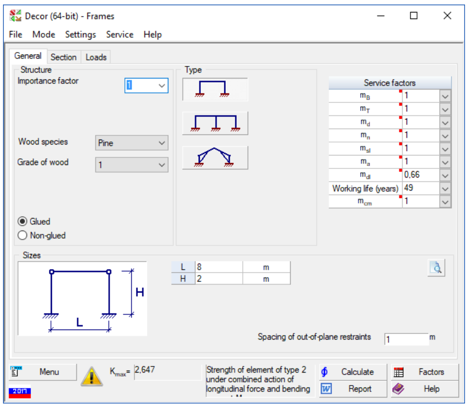


Figure 0-1. *The* **General** *tab of the* **Frames** *dialog box*

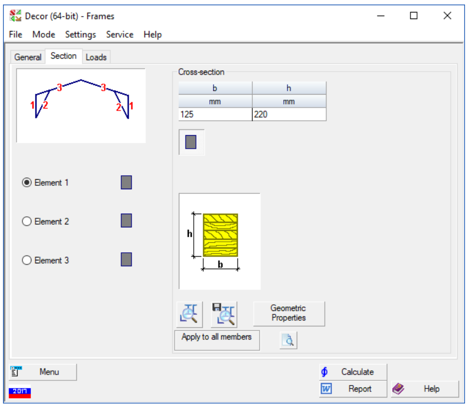


Figure 0-2. *The* **Section** *tab of the* **Frames** *dialog box*

This mode enables to perform checks of three types of single-storey frames from glued and non-glued timber of a rectangular cross-section — two-span and single-span ones with a horizontal girder, single-span ones with a ridge girder and struts.

The **Frames** dialog box (Fig. 0-1) contains three tabs: **General, Section**, and **Loads**.

The **General** tab is used to specify the following data:

- the type of the frame;
- structural characteristics, which include the importance factor, wood species, grade of wood, and the radio-buttons to specify whether the timber is glued or non-glued;
- service factors;
- sizes of the frame;
- the spacing of out-of-plane restraints.

Clicking the button will open the **Preview** dialog box with the model of the specified frame.

The **Section** tab (Fig. 0-2) is used to specify sections for the elements of the frame. You can also use the respective button in this tab to open a section created earlier from the custom sections database, save the sections created for the current problem into the database, browse through the geometric properties of the section, and to apply the section selected from the database to all the elements of the model. Clicking the button will open the **Preview** dialog box with the specified section.

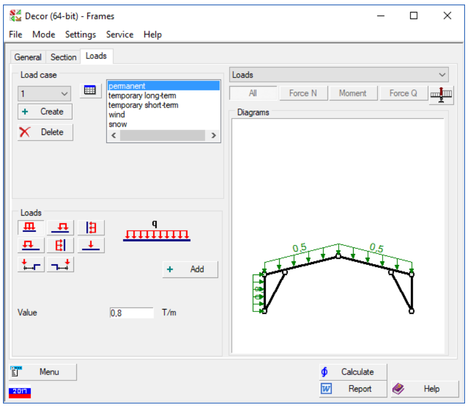


Figure 0-3. *The* **Loads** *tab of the* **Frames** *dialog box*

The **Loads** tab (Fig. 0-3) is used to specify the loads acting on the frame. The application is capable of analyzing several patterns of loading and each load case can in its turn consist of multiple loads.

To enter a new load case (including the first one), follow these steps:

- click the **Create** button in the **Load case** group;
- select a load case type (permanent, temporary long-term, temporary short-term, snow or wind), which determines the combination factors according to SNiP 2.01.07-85* to be used with the loads of this load case in a combination of loads;
- select a load type by clicking the respective button;
- enter values for the parameters of the load;
- click the **Add** button.

A few load components can be specified for each load case. It is assumed that the design values of loads are entered. The application enables to specify the following types of loads:

— uniformly distributed load on the girder;

— concentrated load on the girder (only for frames with struts and arch frames);

— uniformly distributed loads on the right and left spans, on the right and left parts of the girder (except for a single-span frame);

— loads from the adjacent parts of the structure, on the left and on the right;

— wind load on the left and right verticals (except for an arch frame).

The **Delete** button is used to delete a load case (not a separate load included in it). Particular loads can be deleted in the **Table of loads** dialog box (Fig. 0-4), which is invoked by clicking the button. To delete a load, follow these steps:

- use the **Load case** drop-down list to select the number of the load case that contains the load you want to delete;
- click the **Table of loads** button;
- select a line (lines) that contains the load(s) you want to delete;
- click the **Delete** button;
- exit the dialog by clicking the **OK** button.

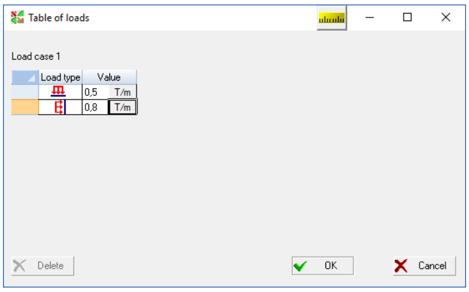


Figure 0-4. *The* **Table of loads** *dialog box*

The table of loads can also be used to edit the value of a load. To do it, just change the value in the respective column and exit the dialog box by clicking the **OK** button.

To switch to the next load case, click the **Create** button, and the number of loadings will be automatically increased by one. If you need to view or modify the data from any of the previously entered load cases, just select its number in the **Load case** list.

Once you click the **Add** button, an image of the current loading is displayed in the **Diagrams** field. If you select the **Diagrams for the current load case** mode from the list above the design model and click the **All** button, the diagrams of moments, longitudinal and shear forces will be displayed together. The buttons **Force N**, **Moment**, **Force Q** are used to obtain a separate diagram of the respective force, as well as the extreme values of each force.

Each element of the frame structure is checked for the following factors:

Check	Reference to SNiP II-25-80, KMK 2.03.08.98	Reference to SP 64.13330.2011	Reference to SP 64.13330.2017	Reference to DBN B.2.6-161:2017	Reference to EN 1995-1-1-2009
Strength under a longitudinal compressive force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.1	Sec. 6.1.4
Stability in the frame plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability out of the frame plane under a longitudinal force	Sec. 4.2	Sec. 6.2	Sec. 7.2	Sec. 9.3.3	Sec. 6.3.2
Stability of in-plane deformation	Sec. 4.14, 4.18	Sec. 6.14, 6.20	Sec. 7.14, 7.20	Sec. 9.4.4, 9.6.2, 9.6.3	Sec. 6.2.4, 6.3.3
Strength under the bending moment	Sec. 4.9	Sec. 6.9	Sec. 7.9	Sec. 9.4.1	Sec. 6.1.6
Strength under combined action of longitudinal force and bending moment	Sec. 4.17	Sec. 6.17	Sec. 7.17	Sec. 9.5.1, 9.5.2, 9.6.2	Sec. 6.2.3, 6.2.4
Strength under the lateral force	Sec. 4.10	Sec. 6.10	Sec. 7.10	Sec. 9.4.2	Sec. 6.1.7

12.4 Peculiarities of the Implementation of SP 64.13330

SP 64.13330.2017 suggests two methods for determining the design strength of timber (according to Sec. 6.1 and Sec. 6.2). **Decor** uses the recommendations of Sec. 6.1. Annex C of SP 64.13330.2011 contains an approach similar to Sec. 6.2, but despite the mandatory status of this Annex, there are no references to it in the text of SP 64.13330.2011. When the calculations are performed according to SP 64.13330.2011, the design strength values are determined on the basis of the Table 3 (see Sec. 5.2).

The durability factor, m_{dur}, has to be specified by the user. Its default value is m_{dur}=0,66. The automatic determination of this factor is not implemented, because the classification of loadings used in the Table C.1 of SP 64.13330.2011 and Table 4 of SP 64.13330.2017 does not correspond to the classification adopted in SP 20.13330.

When determining deflections, the value of the elastic modulus E = 10,000 MPa is used.

12.5 Design Codes the Requirements of Which are Implemented in Decor

Mode	References to sections of codes and standards				
	SNiP II-25-80, KMK 2.03.08.98	**SP 64.13330.2011**	**SP 64.13330.2017**	**DBN B.2.6-161:2017**	**EN 1995-1-1-2009**
Deflection and Strain Limits	Tables 15, 16	Tables 18, 19	Table 17	Table 10.2	Table 7.2
Densities	Annex 3	Annex E	Annex G		
Range of Timber	GOST 24454-80	GOST 24454-80			
Design Strength	Sec. 3.1, 3.2	Sec. 5.1,5.2	GOST 24454-		

Mode	References to sections of codes and standards				
	SNiP II-25-80, KMK 2.03.08.98	SP 64.13330.2011	SP 64.13330.2017	DBN B.2.6-161:2017	EN 1995-1-1-2009
			80		
Timber	Table 1, Table 2, Sec. 3.5	Table 1, Table 2, Sec. 5.3-5.5	Table 1, Table 2		
Limit Slenderness	Table 14, Table 1 of Annex 4	Table 17, Table F.1 of Annex F	Table 16, Table F.1 of Annex F		
Effective Lengths	Sec. 4.21	Sec.6.23	Sec.7.23	Sec. 9.3, 9.4	Sec. 6.3
Resistance of Connections	Sec. 3.1, 3.2, 5.2, 5.3, 5.11,5.13-5.15, 5.18*	Sec. 5.1, 5.2, 7.2, 7.3, 7.11, 7.13-7.15, 7.18	Sec. 6.1, 6.2, 8.2, 8.3, 8.11, 8.13-8/16, 8/20		
Resistance of Sections	Sec. 3.1, 3.2, 4.1, 4.2, 4.4, 4.9, 4.10, 4.12, 4.14-4.18	Sec. 5.1, 5.2, 6.1, 6.2, 6.4, 6.9, 6.10, 6.12, 6.14-6.18	Sec. 6.1, 6.2, 7.1, 7.2, 7.4, 7.9, 7.10, 7.12, 7.14-7.18	Sec. 9.2 – 9.7	Sec. 6.1 – 6.3
Continuous Purlins	Sec. 3.1, 3.2, 4.9, 4.10, 4.12, 4.14, 4.15, 4.18, 4.33	Sec. 5.1, 5.2, 6.9, 6.10, 6.12, 6.14, 6.15, 6.18, 6.35	Sec. 6.1, 6.2, 7.9, 7.10, 7.12, 7.14, 7.15, 7.18, 7.35	Sec. 9.4.1 – 9.4.4	Sec. 6.1.6, 6.1.7, 6.2.4
Cantilever Purlins	Sec. 3.1, 3.2, 4.9, 4.10, 4.12, 4.14, 4.15, 4.18, 4.33	Sec. 5.1, 5.2, 6.9, 6.10, 6.12, 6.14, 6.15, 6.18, 6.35	Sec. 6.1, 6.2, 7.9, 7.10, 7.12, 7.14, 7.15, 7.18, 7.35	Sec. 9.4.1 – 9.4.4	Sec. 6.1.6, 6.1.7, 6.2.4
Beams	Sec. 3.1, 3.2, 4.9, 4.10, 4.14, 4.15, 4.18, 4.33	Sec. 5.1, 5.2, 6.9, 6.10, 6.14, 6.15, 6.18, 6.35	Sec. 6.1, 6.2, 7.9, 7.10, 7.14, 7.15, 7.18, 7.35	Sec. 9.4.1, 9.4.2, 9.4.4	Sec. 6.1.6, 6.1.7, 6.2.4
Columns	Sec. 3.1, 3.2, 4.1, 4.2, 4.4, 4.9, 4.10, 4.12, 4.14-4.18	Sec. 5.1, 5.2, 6.1, 6.2, 6.4, 6.9, 6.10, 6.12, 6.14-6.18	Sec. 6.1, 6.2, 7.1, 7.2, 7.4, 7.9, 7.10, 7.12, 7.14-7.18	Sec. 9.2, 9.3, 9.4.1- 9.4.4, 9.5, 9.6	Sec. 6.1, 6.2.3, 6.2.4, 6.3.2, 6.3.3
Truss Member	Sec. 3.1, 3.2, 4.1, 4.2, 4.4	Sec. 5.1, 5.2, 6.1, 6.2, 6.4	Sec. 6.1, 6.2, 7.1, 7.2, 7.4	Sec. 9.2.1, 9.31, 9.3.3.	Sec. 6.1.2, 6.1.4, 6.3.2.
Trusses	Sec. 3.1, 3.2, 4.1, 4.2, 4.4, 4.33	Sec. 5.1, 5.2, 6.1, 6.2, 6.4, 6.35	Sec. 6.1, 6.2, 7.1, 7.2, 7.4, 7.35	Sec. 9.2.1, 9.31, 9.3.3	Sec. 6.1.2, 6.1.4, 6.3.2
Arches	Sec. 3.1, 3.2, 4.1, 4.2, 4.4, 4.8-4.10, 4.17 Sec. 5.1, 6.16 of SNiP II-23-81*	Sec. 5.1, 5.2, 6.1, 6.2, 6.4, 6.8-6.10, 6.17 Sec. 5.1, 6.16 of SNiP II-23-81*	Sec. 6.1, 6.2, 7.1, 7.2, 7.4, 7.8-7.10, 7.17 Sec. 5.1, 6.16 СНиП II-23-81*	Sec. 9.3.1, 9.3.3, 9.4.1, 9.4.4, 9.5.1, 9.5.2, 9.6.2, 9.6.3	Sec. 6.1.4, 6.1.6, 6.1.7, 6.2.3, 6.2.4, 6.3.2
Rafters	Sec. 3.1, 3.2, 4.1, 4.2, 4.8-4.10, 4.17	Sec. 5.1, 5.2, 6.1, 6.2, 6.8-6.10, 6.17	Sec. 6.1, 6.2, 7.1, 7.2, 7.8-7.10, 7.17	Sec. 9.3.1, 9.3.3, 9.4, 9.5.1, 9.5.2	Sec. 6.1.4, 6.1.6, 6.1.7, 6.2.3, 6.2.4, 6.3.2, 6.3.3.
Frames	Sec. 3.1, 3.2, 4.1, 4.2, 4.8-4.10, 4.17	Sec. 5.1, 5.2, 6.1, 6.2, 6.8-6.10, 6.17	Sec. 6.1, 6.2, 7.1, 7.2, 7.8-7.10, 7.17	Sec. 9.3.1, 9.3.3, 9.4, 9.5.1, 9.5.2, 9.6.2	Sec. 6.1.4, 6.1.6, 6.1.7, 6.2.3, 6.2.4, 6.3.2, 6.3.3

12.6 References

[1] I.M. Grin, V.V. Fursov, D.M. Babushkin, P.G. Galushko, V.I. Grin, Engineering and analysis of timber structures. Reference manual. Kiev, "Budivelnik" Press, 1988, 240 p.

13. Cross

CROSS enables to calculate Winkler coefficient C_1 (compression) using the results of geological surveys. The software implements a procedure developed by NIIOSP experts (a description of the procedure is published in the article "Prediction of settlements of shallow foundations and selection of a subsoil model for the slab analysis" [2].[1]

CROSS determines settlements (and thus the Winkler coefficient) allowing for the distributive ability of subsoil. This feature enables to use results obtained by the program to determine the stress-strain state of structures (for example using **SCAD** [1]) without specifying Pasternak coefficient C_2 (shear).

CROSS selects the values of the Winkler coefficients C_1 which result in the same settlements as in the case of using the linearly-deformable half-space model, i.e. a correspondence with the models recommended by SNiP is provided.

13.1 Coordinate System

The right-handed Cartesian coordinate system (**X**, **Y**, **Z**) is used. **Z**-axis is the longitudinal axis of a building or a structure directed from the drawing plane toward the observer. **Y**-axis is conceived as a vertical one and directed upward in the drawing, **X**-axis is the horizontal axis with the positive direction to the right.

13.2 Files Created by the Program

CROSS creates, reads, and saves results in files with the **crs** extension.

13.3 Structure of a Model and Initial Data

The software considers a construction site where the designed structure is located together with other objects (buildings either existing or being erected) that affect the main structure in the sense that loads upon the soil transferred by these objects may cause settlements of the designed foundation. It is assumed that the spots of the designed structure and its adjacent structures are represented as closed polygons (possibly, with openings). Each polygon transfers a load to the soil, the load being applied at the footing bottom level. It is also assumed that there are known results of geological surveys represented as information about properties of the soil in the drilled boreholes. The surface relief of the site is assumed to be sufficiently smooth and specified as a set of elevations of the borehole mouths. No other data of the geodetic survey are used. The data are entered using a coordinate grid with its spacing specified by the user.

It is recommended to follow these steps to create a model:

- specify overall dimensions of the construction site;
- define parameters of the coordinate grid;
- create the outline of the foundation slab;
- create the outlines of existing buildings (if necessary);
- specify openings (if there are any);
- round the angles (if necessary);
- specify loads;
- specify the footing bottom levels;
- specify locations of boreholes;
- enter soil data;

[1] Shallow foundation is a foundation with the ratio of its height to the width of the footing of less than four and which transfers the load to the subsoil primarily through the footing (see MGSN 2.07-01).

↳ specify the parameters of boreholes.

The results of the analysis are the values of the Winkler coefficient in any point of the foundation slab.

The technique and procedure for this analysis have been developed by V.G. Fedorovsky. Basic principles of this method are described in the paper by V.G. Fedorovsky, S.G. Bezvolev "Prediction of settlements of shallow foundations and selection of a subsoil model for the slab analysis" // Foundations and soil mechanics. 2000. N4. P. 10–18 [2].

13.4 Multi-tab Workspace

A characteristic feature of **Cross** is the possibility of simultaneous displaying of multiple independent windows each containing a different section in the work area. The windows can be invoked either from tabs at the bottom left corner of the work area, or by pointing a cursor at them. All the windows are controlled using a single toolbar. The operations are performed only in the currently active window.

If the **Workbook** checkbox in the **View** menu is checked, tabs with the filenames of sections opened in the windows of the workspace appear in the lower left corner. Clicking on the tab activates the corresponding window.

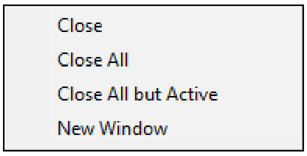


Figure 13.4-1. *Tabs menu*

Right clicking on the tab invokes a menu (Fig. 13.4-1), which enables to perform the following actions:

- close the respective window;
- close all windows;
- close all windows except for the one corresponding to the tab;
- create a new window.

13.5 Saving the Workspace

The settings for each window (workspace) can be saved, which will allow the next session with the program to begin with auto recovery of the settings of the previous session. Program settings are saved in a file, the name of which is specified in the **Save workspace** dialog box (Fig. 13.5-1). **File** menu contains the following operations with workspace customization files **Open Workspace, Save Workspace, Close Workspace** and **Save Workspace As**.

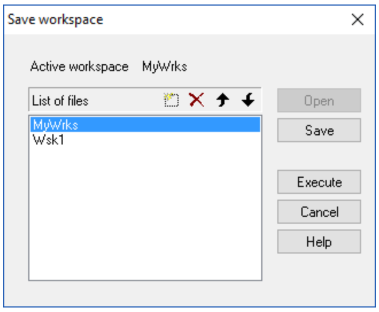


Figure 13.5-1. *The* **Save workspace** *dialog box*

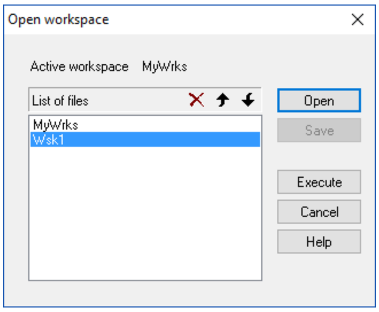


Figure 13.5-2. *The* **Open workspace** *dialog box*

Open Workspace item is used to open models of construction sites and settings saved in the file in the window. Name of the file containing the workspace parameters is selected from the list in the **Open workspace** dialog box (Fig. 13.5-2).

Close Workspace item enables to remove all the windows with models of construction sites that have earlier been saved in the file with the workspace parameters from the screen (only). If at this moment new windows are opened or the previously created models are modified, then a message appears prompting to save the changes. If the answer is affirmative, then all the changes in models saved earlier in a workspace file will not only be included in the files with models, but also in the file with workspace parameters. Models of construction sites in the "new" windows will not be saved in the file with the workspace parameters. Use **Save Workspace** or **Save Workspace As** items to save them.

If the workspace was not loaded by the **Open Workspace** item, **Save workspace** dialog box appears where you either have to select an existing name of a file, which will be used to save the workspace parameters, or to specify a new name using the respective button. **Save Workspace As** item is used in a similar way.

13.6 Controls

13.6.1 Main Window

CROSS (Fig. 13.6.1-1) contains a menu, a toolbar, a work area (with scrollbars if necessary), and a status bar.

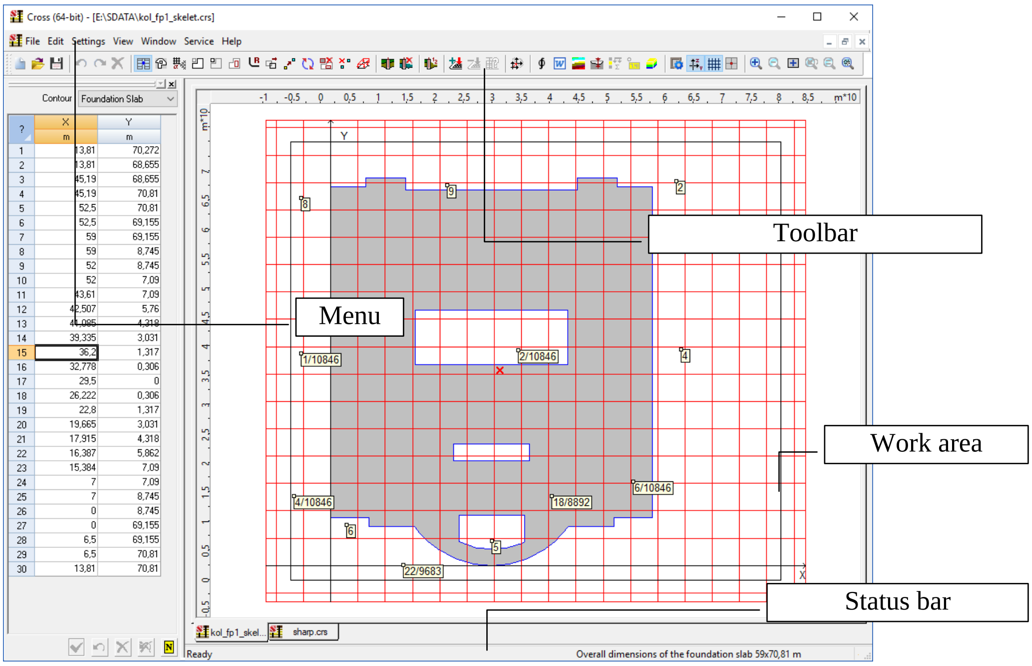


Figure 13.6.1-1. General view of the **CROSS** window

13.6.2 Settings

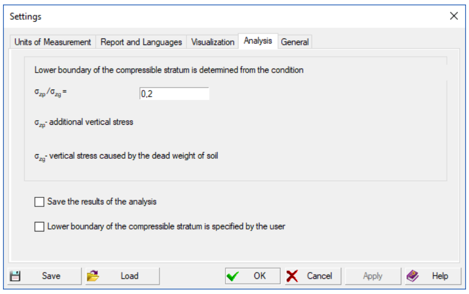


Figure 13.6.2-1. *The* **Analysis** *tab of the* **Settings** *dialog box*

The program contains setting fuctions which enable to specify the units of measurement of the main values and the rules of the report generation. These functions are concentrated in the multi-tab **Settings** dialog box, which can be invoked from the **Settings** menu or from the toolbar (the **Settings** button —). The settings are performed in the same way as described in Sec. 3.2.

The only difference is that the **Settings** dialog box contains the **Analysis** tab (Fig. 13.6.2-1), which enables to specify the ratio σ_{zp}/σ_{zg} between an additional vertical stress (σ_{zp}) and a vertical stress caused by the dead weight of soil (σ_{zg}). This ratio will determine the lower boundary of the compressible stratum. The default value for this ratio is 0.2 which complies with SNiP 2.02.01-83* Foundations of buildings and structures, though you can specify any number from the interval [0.1…0.5] in compliance with requirements of SP 50-101-2004 and SP 22.13330.2011.

This tab also contains the **Save the results of the analysis** checkbox. If it is checked, the results of the analysis will be saved whenever the model is saved (if the analysis has been performed). Though it increases the size of the model file dramatically, it enables not to perform the calculation again after restarting the program and re-loading the model, and you can analyze the results at once. In the case of the presence of bedrock the **Lower boundary of the compressible stratum is specified by the user** checkbox enables to perform the analysis of the compressible stratum not down to the bedrock, but to the level specified by the user.

13.6.3 Menu

Cross has the following pull-down menus in the upper part of the window: **File**, **Edit**, **Settings**, **View**, **Window**, **Service** and **Help**.

The **File** menu includes the following set of commands:

- **New** — creates a new site (“hot keys” combination — **Ctrl+N**);
- **Open** — opens a previously created site (“hot keys” combination — **Ctrl+O**);
- **Save** — saves the current site to disk (“hot keys” combination — **Ctrl+S**);
- **Save As…** — saves the site (file) under a new name;
- **Import DXF, DWG ...** — imports data on the geometry of a foundation slab from the files of the DXF or DWG format;
- **Send —** sends the file by electronic mail;
- **Calculate** — calculates Winkler coefficients;
- **Report** — generates a report;
- **Fields** — displays fields of Winkler coefficients;
- **Settlement...** — displays fields of distribution of the footing settlements;
- **Load Field** — displays fields of loads at the non-uniform load on the slab;
- **Save Picture** — saves the picture currently on the screen in a Windows metafile format;
- **Open Workspace ...** —opens models of construction sites in the windows;
- **Save Workspace** — saves the workspace parameters in a file;
- **Save Workspace As…** — saves the workspace parameters in a file selected by the user;
- **Close Workspace** — removes all the windows with models of construction sites that have earlier been saved in the file with the workspace parameters from the screen (only).

The **Edit** menu includes the following set of commands:

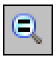
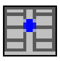
- **Undo** — undo the last action;
- **Redo** — redo the previously undone action;
- **Dimensions** — specifying the overall dimensions of the site where the designed structure is located together with other objects that affect it (this action and all the following ones in this menu duplicate the respective buttons of the toolbar);
- **Foundation Slab** — creating and modifying the outline of a foundation slab;
- **Existing Building** — creating and modifying the outline of a building that stands beside the designed structure and has an effect on it;
- **Opening** — creating the outline of an opening;
- **Delete** — deleting existing buildings and/or openings;
- **Round Angle…** — round a selected angle by an arc of the specified radius;
- **Move** — move vertices of the contours of the foundation slab or of the existing buildings;
- **Move Vertices** — edit the coordinates of vertices of the foundation slab or of the existing buildings;

- **Delete Vertices** — delete one or more vertices;
- **Assign Foundation Slab** — assign the spot of the existing building as the design foundation slab;
- **Load** — specify a load on the foundation slab (or on the existing building) and the footing bottom levels;
- **Create Area with Extra Load** — create an area on the foundation slab with the extra load;
- **Delete Area with Extra Load** — delete an area with the extra load;
- **Change Load on Area** — edit the value of a load in the additional area;
- **Add Point with Extra Load** — create points in which the load values are determined;
- **Delete Point with Extra Load** — delete additional points with the specified load;
- **Points with Extra Load** — edit the coordinates and values of loads in the additional points;
- **Create Borehole** — add a borehole;
- **Delete Borehole** — delete a previously created borehole;
- **Borehole Parameters …** — invoke a dialog box to specify the information on soils and the parameters of boreholes;
- **Origin …** — shift the origin;
- **Compressible Stratum Calculation Point** — specify a point where the compressible stratum level will be determined.

The **Settings** menu includes the following set of commands:

- **Settings …** — invoke the **Settings** dialog box where you can customize the program;
- **Grid Settings …** — specify the grid spacing;
- **Grid** — display the grid in the work area;
- **Coordinate Axes** — display the coordinate axes of the site;
- **Additional Points** — create points on fields for which to display markers with values of the Winkler coefficients;
- **Vertices** — display vertices;
- **Vertices Numbers** — display the numbers of vertices;
- **Load Fields** — displays fields of loads;
- **Loads Obtained from SCAD** — for models imported from SCAD this option enables to view the fields of imported loads;
- **Fields for Buildings** — checking this checkbox will display the fields of Winkler coefficients not only under the foundation slab but also in the area of the adjacent buildings;
- **Workbook —** invokes the tabs for switching between windows in a multi-tab mode.

The **View** menu includes the following set of commands:

- **Zoom** — zooms the site;
  -  **In** — zooms in the site;
  -  **Out** — zooms out the magnified site;
  -  **Rect —** zooms in the part of the site selected by the rectangle;
  -  **Undo —** returns to the previous scale;
  -  **Initial —** returns to the initial scale;
- **Toolbar —** show/hide the toolbar;
- **Status Bar —** show/hide the status bar;
-  **Table of Vertices —** show/hide the table of vertices.

The **Window** menu includes the following items: **New Window**, **Cascade**, **Tile** and **Arrange Icons**, which are standard commands of Windows environment for arranging the windows in a multi-tab mode.

The **Service** menu contains items for invoking a standard calculator of the Windows environment, a formula calculator, and a converter of units of measurement.

The **Help** menu provides the help information on using the application and on its functionality, information about the application, and it also contains an item that enables you to check for update.

13.6.4 Status Bar

Status bar (Fig. 13.6.4-1) contains three fields: **Overall dimensions**, coordinates, and **Distance**. The first field displays the specified overall dimensions of the foundation slab. The second field displays the coordinates of the current position of the cursor. The third field is used for displaying a distance between two points of a site in the measuring mode.

Figure 13.6.4-1. Status bar

13.6.5 Cursors

All actions are performed in the work area with a cursor. When moving the cursor over the screen or when performing some commands, the shape of the cursor changes. For example, when selecting an item from the menu or the toolbar the cursor takes the form of an arrow, when processing a command the cursor turns into an hourglass (busy cursor), when working in the work area the cursor is displayed as a cross.

A distance between two points of the site can be determined with the cursor. To do this, place the cursor over the first point and left-click. Drag the pointer to the second point while holding the button. The right part of the status bar will display the distance between the points (the accuracy of this indication depends on the precision specified in the **Units of Measurement** tab of the **Settings** dialog box). Coordinates of the current position of the cursor will be displayed in the status bar.

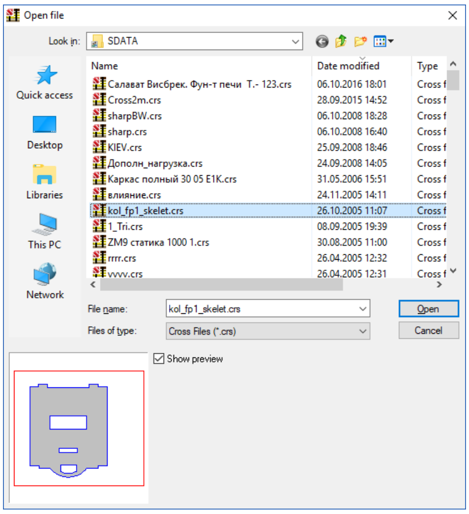
13.7 Toolbar

Pointing and left-clicking on a button in the toolbar invokes the corresponding command. Henceforward, this sequence will be called “clicking the button in the toolbar”.

13.7.1 New

This item is used to prepare a new site. As a result a new window appears where you can create a new site.

13.7.2 Open

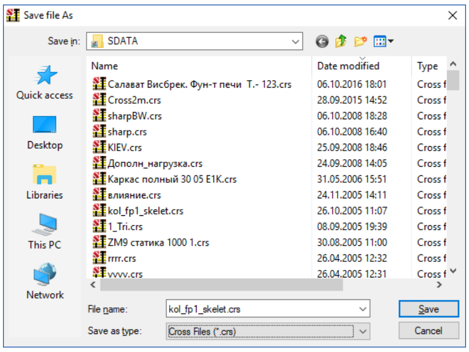


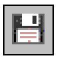
This item enables to open a previously created site. A standard Windows dialog box with a list of files with the .crs extension (Fig. 13.7.2-1) appears once the command is invoked. As in the previous case, the site is opened in a new window.

To preview the sites check the **Show preview** checkbox.

Figure 13.7.2-1. *The* **Open file** *dialog box*

13.7.3 Save





This item allows you to save the data on a site in a file. If the site has not been previously saved, a standard Windows dialog box appears where you have to enter a file name (Fig. 13.7.3-1).

Figure 13.7.3-1. *The* **Save file As** *dialog box*

13.7.4 Save As...

This item is used to save the data on the site in a file under a new name. Invoking the action will open a standard Windows dialog where you have to enter a file name (see Fig. 13.7.3-1).

13.7.5 Import

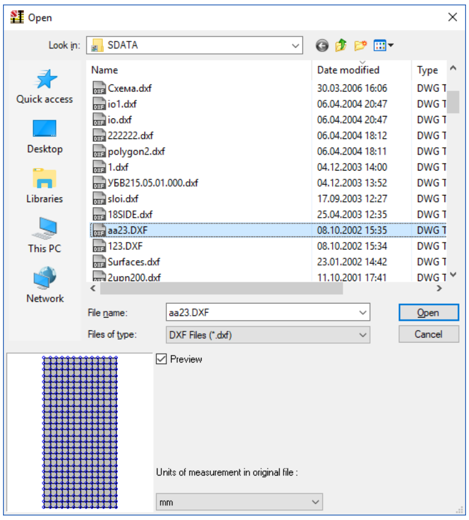


Figure 13.7.5-1. *The* **Open** *dialog box*

The description of geometry of a foundation slab can be imported from the AutoCAD system in the **DWG** or **DXF** file formats. The following types of graphic primitives are supported:

- 3DFACE
- SOLID
- TRACE
- LINE
- POLYLINE
- LWPOLYLINE
- ELLIPSE
- CIRCLE
- ARC

This mode is invoked by the **Import DXF, DWG** item of the **File** menu. In the dialog box with a list of files (Fig. 13.7.5-1) besides the preview option, there is a list where you have to select the units of measurement used when creating the drawing in AutoCAD. Moreover, data from other graphic formats can be imported as well.

The **Import** operation enables also to import the data on the geological structure (soils, boreholes) from geoXML files created by CREDO GEOLOGY. It is assumed that all information recorded in these files is in tones and meters. If there are data on several tests for a certain borehole, then the data on the first test will be imported. Adjacent identical soil layers are combined during the import.

13.7.6 Calculate

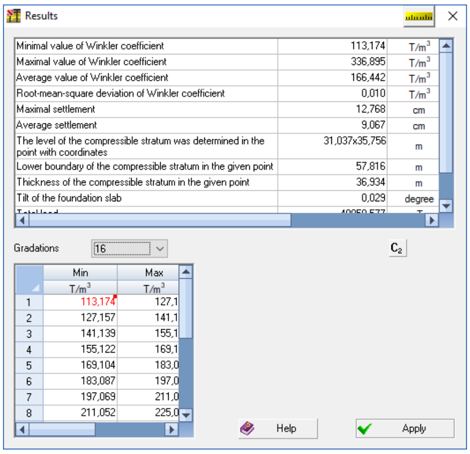


Figure 13.7.6-1. *The* **Results** *dialog box*

Clicking this button invokes the calculation of Winkler coefficients. Once the calculation is performed, the **Results** dialog box appears (Fig. 13.7.6-1), which provides the maximal and minimal values of the Winkler coefficient, the bottom level of the compressible stratum, and the thickness of the compressible stratum at the point of its determination (if there is no bedrock), the root-mean-square value of the Winkler coefficient, the root-mean-square deviation of the Winkler coefficient, the total load on the foundation slab, the maximal settlement, the root-mean-square settlement, and the tilt of the foundation slab. Moreover, in this dialog you can select the number of color ranges which will be used to display the fields, and you can also change the boundaries of intervals if necessary.

The root-mean-square deviation enables to make a decision in each particular case on whether the root-mean-square value of the Winkler coefficient can be used or various values over the area of the slab should be specified.

Moreover, this dialog box also provides the volume of excavated soil without consideration of additional earthworks

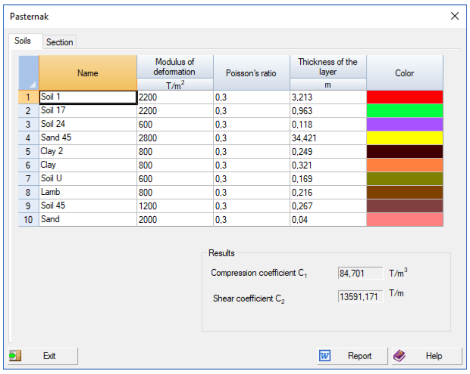


Figure 13.7.6-2. *The* **Shear Coefficient** *dialog box* **a**

necessary to provide the operation of equipment (e.g., excavation ramps) and without consideration of the slope of the excavation walls.

The dialog box (Fig. 13.7.6-2) has also the button $\boxed{C_2}$, which enables to determine Winkler coefficient (C_1) and Pasternak coefficient (C_2). This calculation is based on the assumption that the soil stratum is uniform in plan. The calculation is based on the model described in Section 13 devoted to **Pasternak**.

The data are generated as follows. The volume of each type of soil under the foundation slab is calculated. Each volume is divided by the area of the foundation slab and the obtained value is treated as the thickness of a layer of the horizontally layered soil deposit. In the result of the calculation we obtain the average value of the Pasternak coefficient. It should be noted that this model of soil behavior is different from the model is used in **CROSS**, so do not be surprised if the obtained values of the Winkler coefficient differ significantly from the values calculated by **CROSS**, even if the soil stratum is uniform in its plan.

13.7.7 Generating a Report

Once the operation is activated, a report containing the results of the analysis is created (see Sec. 2.5).

13.7.8 Fields

This action enables to generate fields of Winkler coefficients (Fig. 13.7.8-1).

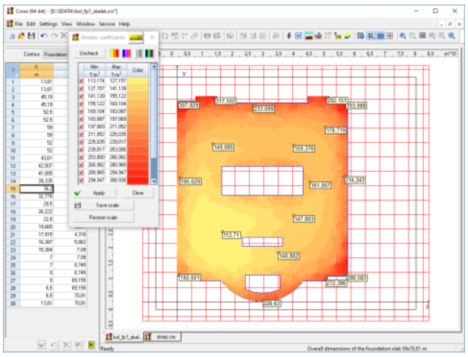


Figure 13.7.8-1. *Foundation slab with fields of Winkler coefficients*

The **Winkler coefficients** dialog box (Fig. 13.7.8-2) has the controls of the color scale. They enable to control the visualization depending on the character of results and technical capabilities of the computer. There are checkboxes which correspond to each range of results, and thus the color too. Checking the checkbox makes the results included in the respective range visible on the model. By unchecking a checkbox, you can choose not to display some of the results in color if those are of little importance for the current analysis.

To display the results of the color scale setup on the screen, click the **Apply** button at the bottom of the dialog box.

A row of buttons at the top is used to change the color gamut of the scale. The gamut changes on the screen once the **Apply** button is clicked.

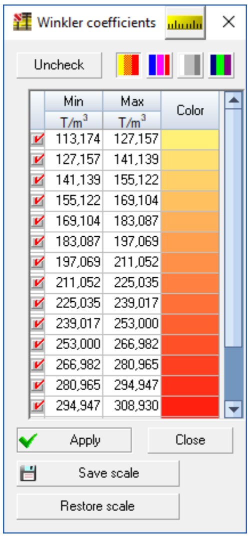


Figure 13.7.8-2. *The* **Winkler coefficients** *dialog box*

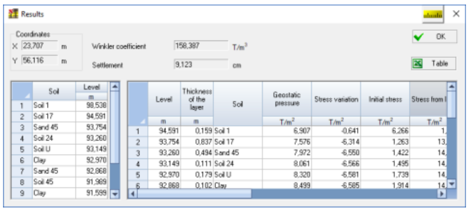


Figure 13.7.8-3. *A dialog box with the results of calculation in a particular point*

The **Check/Uncheck** button is used to simultaneously check or uncheck all the checkboxes of the color scale. This button is handy in cases when you need to choose only a few values of the whole range. To do it, first, uncheck all the checkboxes; and then check the checkboxes corresponding to the respective values and click the **Apply** button.

The color scale dialog box can occupy any position on the screen. It can be closed (by the **Close** button) or moved out of the work area (if the program window occupies only a part of the screen).

Place the cursor within the foundation slab and right-click to change the number of color ranges which will be used to display the fields or to change the interval boundaries. This will open the **Winkler coefficients** dialog box (Fig. 13.7.8-2).

Colors suggested for the scales are not obligatory. You can change any color in the scale or even the whole gamut, and save these changes for all subsequent working sessions. To do it, place the pointer over a color box with the color you want to change, and double-click the left mouse button. This will open a standard Windows dialog for selecting the color. Once the color is selected, click the **OK** button. The new color will appear in the color scale. Click the **Save scale** button to save the new color gamut. After this action, the new color gamut will be always displayed instead of the previous one.

It should be noted that when moving the cursor over the foundation slab, the value of the Winkler coefficient at the current position of the cursor is displayed in the status bar (Fig. 13.7.8-1).

Clicking the left mouse button will open the **Results** dialog box (Fig. 13.7.8-3) for the current position of the cursor. This dialog displays the coordinates of the point, a table describing the structure of the soil stratum, the Winkler coefficient, and the footing settlement. The information on the settlement is given with the assumption that the foundation slab has zero stiffness.

13.7.9 Additional Points

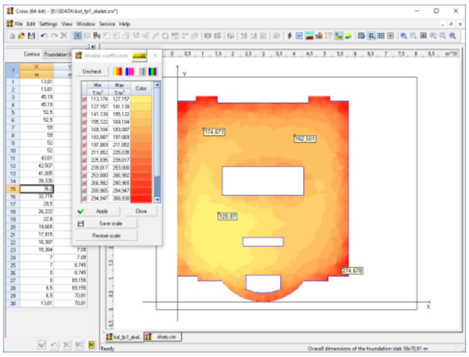


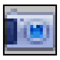
Figure 13.7.9-1. *Fields with additional markers*

If the **Additional Points** button is selected, then left-clicking will not open the dialog box of results (Fig. 13.7.8-3). Instead, values of the Winkler coefficient will appear on fields in the points indicated by the pointer (Fig. 13.7.9-1). Clicking the right mouse button will open a menu where you can select one of the following actions: remove additional points or invoke the dialog box to change the parameters of the color scale.

13.7.10 Settlements

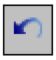
This mode enables to generate fields of distribution of the footing settlements. The rules for output are the same as those for the generation of the fields of Winkler coefficients.

13.7.11 Save Picture



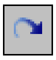
This action enables to save the picture currently on the screen as a Windows metafile (.wmf) which can be used afterwards to create a report document. Clicking the button will open a standard MS Windows dialog box where you have to specify a filename for the picture.

13.7.12 Undo



This item enables you to undo the previous action. The undo history is unlimited.

13.7.13 Redo



This item enables you to redo the previously undone action.

13.7.14 Dimensions

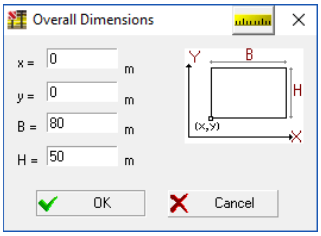


Figure 13.7.14-1. *The* **Overall Dimensions** *dialog box*

A foundation slab is created on the coordinate grid the overall dimensions of which are limited by those of the site. The dimensions of the site and its snap are specified in the **Overall Dimensions** dialog box (Fig. 13.7.14-1) in the units of measurement defined in the respective tab of the **Settings** dialog box.

Once the dialog is closed, a rectangle limiting the overall dimensions of the site will appear in the work area (Fig. 13.7.14-2).

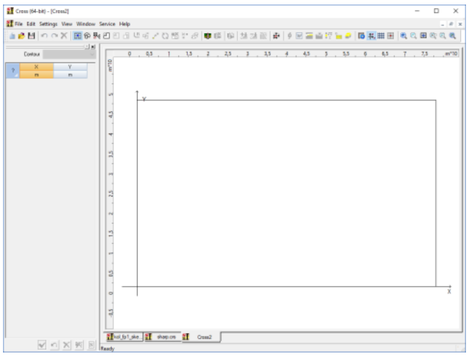


Figure 13.7.14-2. *Overall dimensions of the site displayed in the work area*

13.7.15 Foundation Slab

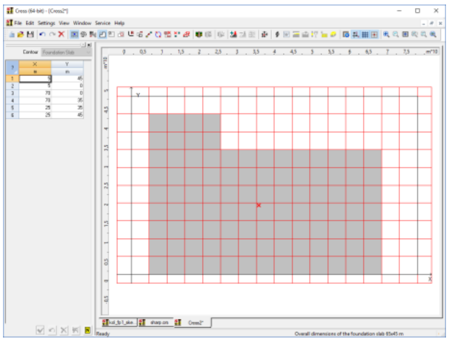
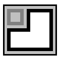


Figure 13.7.15-1. *A foundation slab displayed in the work area*



The external contour of the foundation slab can be created by consecutively adding the vertices of the polygonal contour by the cursor. Each vertex is created by left-clicking. The contour is closed by double-clicking the left mouse button. The last point is connected to the first one and the outline of the slab is displayed on the screen (Fig. 13.7.15-1).

The vertices can be placed arbitrarily or snapped to the nearest grid node. The type of snap is specified in the **Settings** menu. If there is no snap, the current coordinates of the cursor will be displayed in the second field of the status bar. If the **Snap to Grid** option is enabled, the coordinates of the grid node nearest to the cursor will be displayed in the field of the status bar.

13.7.16 Edit the Slab Outline

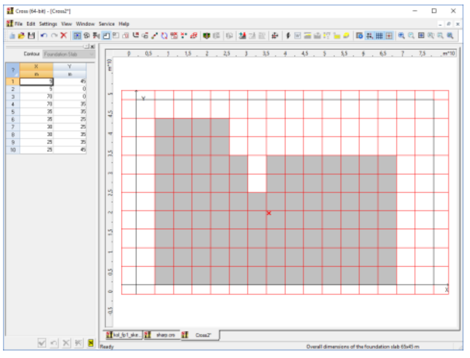
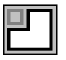


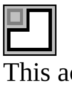
Figure 13.7.16-1. *Slab with an edited outline*



If the slab has already been created, clicking on the **Foundation Slab** button for the second time enables you to edit its outline. Place the cursor over any point of the contour to start the editing. After the cursor changes its shape (to a cross for an arbitrary point or to a cross with a target for a vertex), press the left mouse button and "drag" the selected point to a new position. The new vertex is fixed by double-clicking the left mouse button. A slab with an edited outline is shown in Fig. 13.7.16-1.

When moving the vertices the intersection of the sides of the slab outline and the intersection of sides of the opening with those of the outline are not allowed.

13.7.17 Existing Building



This action enables to create the outline of an existing building (or buildings). The procedure in this case is just the same as that used to create the outline of a foundation slab.

13.7.18 Opening

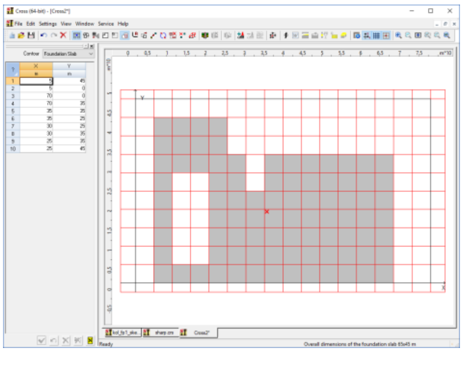


Figure 13.7.18-1. *An example of a foundation slab with openings*

This action enables to create openings in a foundation slab or in existing buildings. The openings are specified in the form of a closed polygon. The sequence of operations for creating and editing an opening does not differ from that for the outline of a foundation slab.

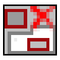
An example of a foundation slab with openings is shown in Fig. 13.7.18-1.

When creating polygonal openings, their intersection with the outline of the foundation slab and with other openings is not allowed.

13.7.19 Assign Foundation Slab

In some cases it is necessary to design a few buildings standing near each other at the same time (for example, a city block). A situation may arise when it is unknown which building will be erected first. Consequently, during the creation of a model it is not clear which building (in terms of the **CROSS** software) should be treated as a foundation slab and which one as an existing building. This action enables to re-assign a foundation slab, that is, by invoking the action and pointing at an existing building you can assign the selected building to be the new foundation slab. The previous foundation slab will then become an "existing building".

13.7.20 Delete



This action enables to delete an opening (either in a foundation slab or in an existing building) or even an existing building itself. To perform this action, place the cursor over the object you want to delete and click the left mouse button.

13.7.21 Round Angle

An angle can be rounded by inscribing a circular arc of a given radius in it. After you invoke this command, place the cursor over a vertex of the contour (external or that of an opening) and when the cursor takes the form of a cross with a target, click the left mouse button. In the invoked **Radius** dialog box (Fig. 13.7.21-1) specify a radius and click the **OK** button. A slab with rounded angles is shown in Fig. 13.7.21-2.

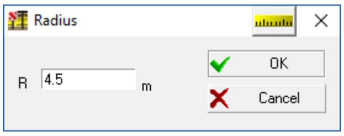


Figure 13.7.21-1. *The* **Radius** *dialog box*

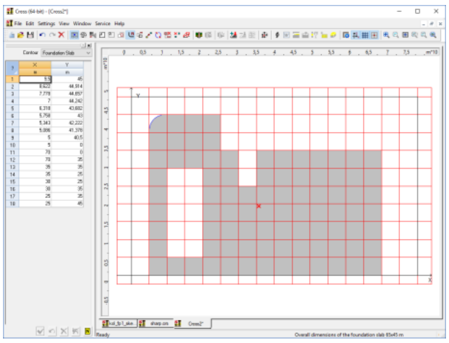


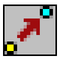
Figure 13.7.21-2. *An example of a slab with rounded angles*

13.7.22 Move

This command enables to move the contour to a new position. You can move the whole building or the selected opening. The type of the object you want to move is selected in the menu which appears once you invoke this command. The whole building can be moved (shifted) only within the given boundaries. Thus if the building occupies the entire boundary, change the overall dimensions before moving the contour. Follow these steps to perform this operation:

- invoke the command and select the type of the object you want to move;
- place the cursor inside the contour you want to move and left-click;
- after the cursor changes its shape move the contour to a new position;
- left-click to confirm this new position.

13.7.23 Move Vertices



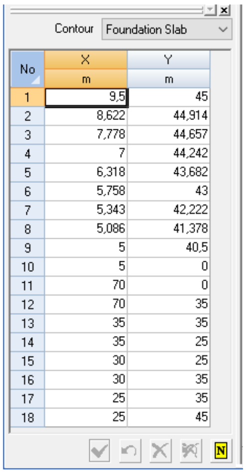
This command enables to move one vertex or a group of vertices selected by a cursor, rectangular or polygonal marquee. Follow these steps to perform this operation:

- invoke the command and choose a cursor for selecting vertices;
- if you have to move one vertex, select it with a cursor and left-click;
- move the vertex to a new position and left-click to confirm it;
- if you have to move a group of vertices, select them with a rectangular or polygonal marquee;
- move the cursor inside the marquee and when it changes its shape, move the marquee together with the selected vertices to a new position;
- left-click to confirm this new position.

When moving the vertices the intersection of the sides of the slab outline and the intersection of sides of the polygonal openings with those of the slab outline are not allowed.

13.7.24 Table of Vertices

This command enables to edit the coordinates of the vertices in a tabular form. Click on the **Table of Vertices** button and this table (Fig. 13.7.24-1) will appear on the left from the work area. The dialog box includes a list of contours in the order of their creation and the table with the coordinates of vertices selected from the contour list.



No	X (m)	Y (m)
1	9,5	45
2	8,622	44,914
3	7,778	44,657
4	7	44,242
5	6,318	43,682
6	5,758	43
7	5,343	42,222
8	5,086	41,378
9	5	40,5
10	5	0
11	70	0
12	70	35
13	35	35
14	35	25
15	30	25
16	30	35
17	25	35
18	25	45

Figure 13.7.24-1. *The table of vertices*

Perform the following steps to edit the position of vertices:

- select a contour from the list;
- press the button [N] to display the numbers of vertices;
- change the coordinates of a vertex in the table of coordinates;
- press the **Apply Modifications** button.

This dialog also contains the following buttons: a button for undoing the performed operation, a button for highlighting the vertices you want to delete and a button for removing these highlights.

To delete the vertices select the respective rows in the table and click on the **Delete Vertices** button. All these vertices will be highlighted in the section and deleted once you press the **Apply Modifications** button.

When moving the vertices the intersection of the sides of the slab outline and the intersection of sides of the polygonal openings with those of the slab outline are not allowed.

13.7.25 Delete Vertices

This command enables to delete one vertex or a group of selected vertices. Follow these steps to perform this operation:

- invoke the command;
- select the vertices you want to delete with a rectangular or polygonal marquee;
- click the left mouse button.

13.7.26 Load

This command is used to specify a load applied at the footing bottom level, and the value of the footing bottom level itself. Follow these steps to perform this operation:

- invoke the action;
- select the object (foundation slab or the nearby building) you want to apply the load to and left-click;
- specify the value of the load and the footing bottom level in the **Load** dialog box (Fig. 13.7.26-1).

The subsoil subsidence *W* (see Fig. 13.7.26-2) depends on the applied load *P* nonlinearly; therefore, the Winkler coefficient is generally not a constant value.

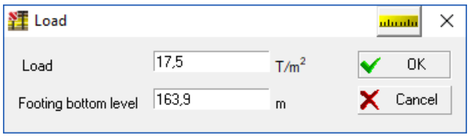


Figure 13.7.26-1. *The* **Load** *dialog box*

In the engineering practice, the nonlinear relationship OBA is usually replaced by the linear one OB (a “chord modulus” analysis). Tangent of the inclination angle of this linear plot characterizes the Winkler coefficient.

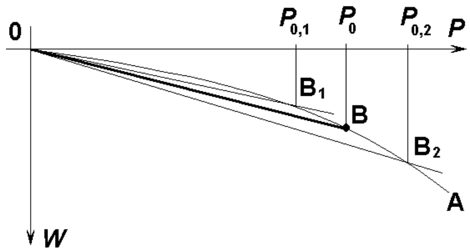


Figure 13.7.26-2.

Therefore the applied load P_0 should take the most probable expected value (for example, the load from the dead weight plus long-term loads and a sustained part of short-term loads). The actual structure may be subjected to other loads as well, but if the main part of the load has been taken into account then small variations between $P_{0,1}$ and $P_{0,2}$ will not have a considerable effect on the Winkler coefficient (the inclination of the OB_1 and OB_2 lines is hardly different from that of OB).

13.7.27 Create Area with Extra Load

A non-uniform load on the foundation slab is modeled by introducing an area or a point with an extra load. Once this command is invoked, use the mouse pointer to create a polygonal area of load of any shape. Double-click the left mouse button to finish creating the area, and this will open the **Load** dialog box where you have to specify the value of the load.

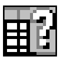
Areas with extra load can be created by invoking this action from the main menu or by clicking the respective button of the toolbar. The entered load value will be added to the value of the uniform load on the foundation slab specified earlier using the **Load** action.

Note. If a non-uniform load is acting on the foundation slab and there is no bedrock, the lower boundary of the compressible stratum will be determined using the value of the average load on the foundation slab.

13.7.28 Delete Area with Extra Load

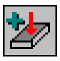
In order to delete an area with extra load, invoke the **Delete Area with Additional Load** command, point at an area you want to delete, and click the left mouse button.

13.7.29 Change Load on Area



A load applied to an area can be modified. To do it, invoke the action, point at the respective area, click the left mouse button, and specify a new value for the load in the dialog box that opens.

13.7.30 Add Point with Extra Load

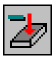


In cases when a load in a certain convex area changes according to a linear law, you can enter the points bounding this loaded area. The load at points inside the area is calculated by linear interpolation of the load values in the entered points and added to the load specified earlier. These points (three at least) are entered using the **Add**

Point with Extra Load action, which can be invoked from the **Edit** menu. After entering each point, you have to specify a load value for it.

It should be noted that the load in additional points must be specified in *units of pressure* — please do not confuse these points with the points of application of the concentrated forces.

13.7.31 Delete Point with Extra Load



Points with extra load can be deleted using the command which can be invoked from the **Edit** menu.

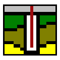
13.7.32 Points with Extra Load

The **Points with Extra Load** function enables to change the load values or coordinates of points in a table.

13.7.33 Load Field

To verify that the loads on the slab have been specified correctly, you can use the **Load Field** action and obtain a picture of load fields.

13.7.34 Create Borehole



This command is used to specify a location of a borehole on the construction site. To enter a new borehole, point at its location with the cursor and fix it by left-clicking. The work area will display a number of this borehole (or its name if it has been specified). If the **Snap to Grid** mode is active, the boreholes will be created at the nodes of the grid nearest to the position of the pointer. You can modify the coordinates of a borehole by clicking the **Borehole Parameters** button.

13.7.35 Delete Borehole

To delete a borehole, place the pointer over it and click the left mouse button.

13.7.36 Borehole Parameters

Invoking this action opens the **Borehole Parameters** dialog box (Fig. 13.7.36-1) where you have to specify the following information for each borehole:

- coordinates of the borehole (they were specified when the borehole was created, but can be changed in this dialog);
- name of the borehole;
- information on soils;
- levels of soil layers (upper) for each borehole.

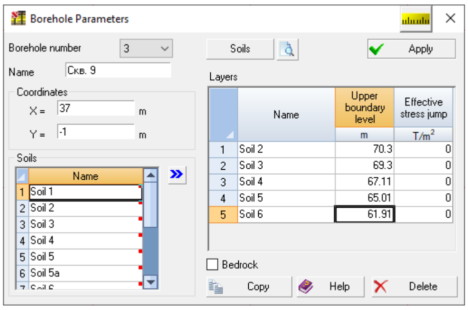


Figure 13.7.36-1. *The* **Borehole Parameters** *dialog box*

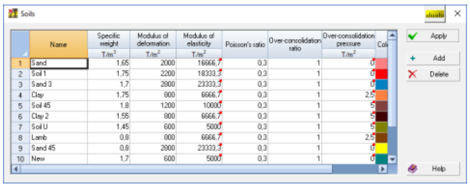


Figure 13.7.36-2. *The* **Soils** *dialog box*

If there is bedrock under the soil layers, you have to enable the respective checkbox. The calculation of settlements with depth will be limited by the level of the deepest soil layer. The name of the latter is replaced by "rock" in the table. Otherwise, the program will determine the depth of the compressible stratum assuming that the deepest layer has an infinite depth.

Before specifying the borehole parameters it is necessary to describe the characteristics of all soils. These characteristics are specified in the **Soils** dialog box (Fig. 13.7.36-2). First click the **Add** button before entering the parameters for each soil. You have to specify the following information in the new line of the soil table:

- name of the soil;
- specific weight (allowing for buoyancy);
- modulus of deformation;
- modulus of elasticity (if this parameter is set to zero, it will be calculated automatically as D/0.12 where D is the modulus of deformation);
- Poisson's ratio;
- over-consolidation ratio;
- over-consolidation pressure.

In the last column you can select a color which will be used to display the given soil in sections.

It should be noted that when the considered ground layer contains the groundwater level, the soil of the same type, but dry or water-saturated, should be referred to different layers. In this case the decrease in the specific weight of water-saturated soil due to the buoyancy acting on mineral particles should be taken into account:

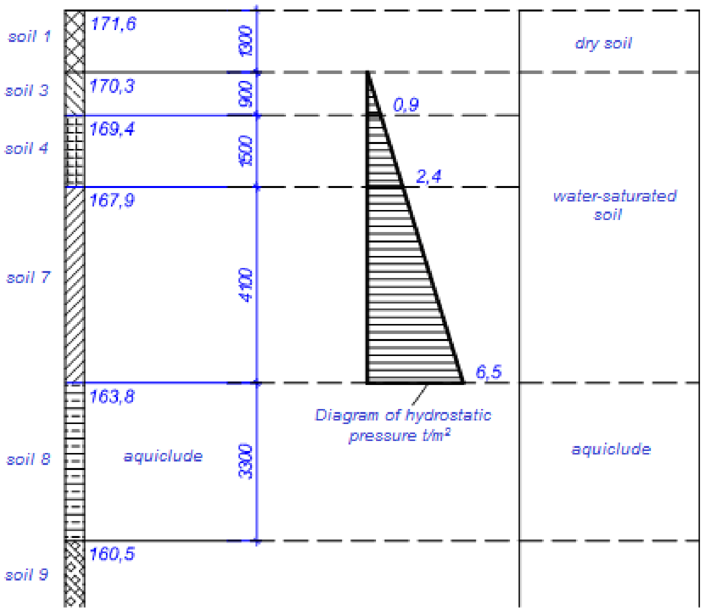
$$\gamma_{sb}=(\gamma_s-\gamma_w)(1+e)$$

where γ_{sb} — buoyant specific weight of soil, γ_s — specific weight of soil particles, γ_w=10 kN/m^3 — specific weight of water, e — soil void ratio.

If the aquiclude lies at some depth below the water-saturated soil level, the weight of the water column acts on its upper boundary, and a (positive) effective stress jump equal to the hydrostatic pressure at this level occurs.

In cases when water-saturated layers are located under the aquiclude and at the same time groundwater under and above the aquiclude are connected somewhere outside the borehole (aquiclude outcrop), a **negative** effective pressure jump occurs at the lower boundary of the aquiclude, the value of which corresponds to the hydrostatic pressure at the depth of the lower boundary of the aquiclude. This jump is absent when there are two independent groundwater horizons (above and below the aquiclude).

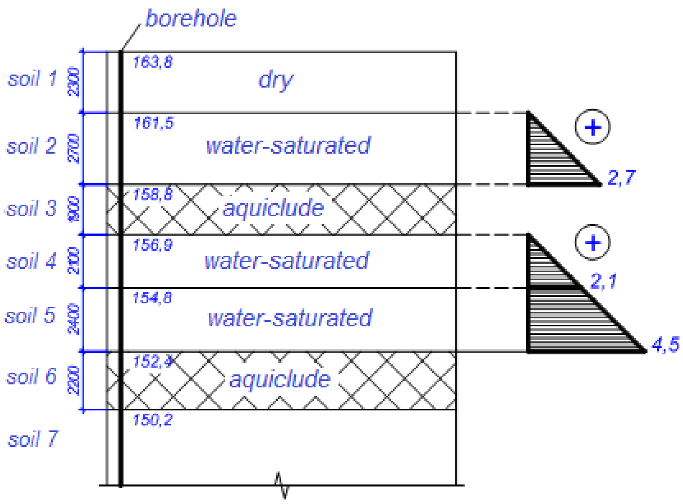
Let's illustrate this by several examples given below.



Layers

	Name	Upper boundary level	Effective stress jump
		m	T/m²
1	Soil-1	171,6	0
2	Soil-3	170,3	0
3	Soil-4	169,4	0
4	Soil-7	167,9	0
5	Soil-8	163,8	6,5
6	Soil-9	160,5	0

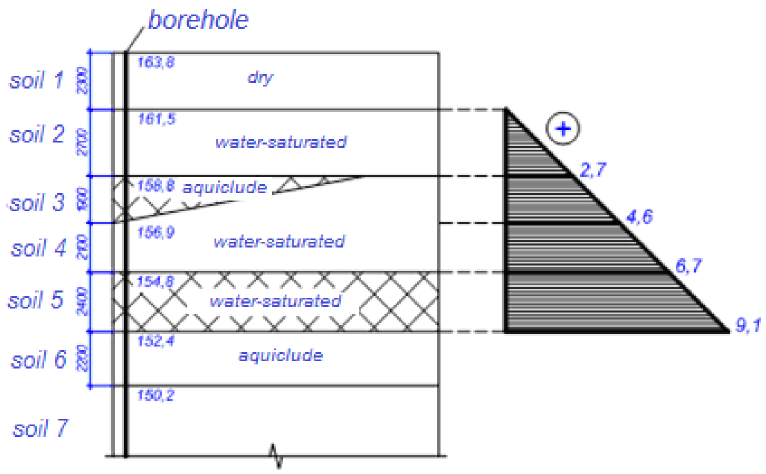
Specification of the effective stress jump



Layers

	Name	Upper boundary level	Effective stress jump
		m	T/m²
1	Soil-1	163,8	0
2	Soil-2	161,5	0
3	Soil-3	158,8	2,7
4	Soil-4	156,9	0
5	Soil-5	154,8	0
6	Soil-6	152,4	4,5
7	Soil-7	150,2	0

Two independent groundwater horizons (above and below the aquiclude)



Layers

	Name	Upper boundary level	Effective stress jump
		m	T/m²
1	Soil-1	163,8	0
2	Soil-2	161,5	0
3	Soil-3	158,8	2,7
4	Soil-4	156,9	-4,6
5	Soil-5	154,8	0
6	Soil-6	152,4	9,1
7	Soil-7	150,2	0

Groundwater under and above the aquiclude are connected outside the borehole (aquiclude outcrop)

One of the main parameters that describe the soil behavior is the structural strength p_c. The p_c value can be represented as

$$p_c = R\sigma + \Delta p_c ,$$

where R is the over-consolidation ratio (related to the age of the soil stratum), Δp_c is the over-consolidation pressure (related to maximal pressures that have been acting on the soil during the whole history of its formation), σ is the geostatic pressure. The over-consolidation ratio and the over-consolidation pressure are constant, and they can be treated as soil parameters.

If there are no experimental field data about the values of R and Δp_c, it is recommended to use the over-consolidation coefficient of 1.0, and the over-consolidation pressure of:

5 t/m^2 — for clayey soil;
2,5 t/m^2 — for sandy loam;
0 t/m^2 — for sand.

If any of the soils has to be deleted, select the respective line with the mouse pointer and click the **Delete** button.

Having finished entering the soil properties, click the **Apply** button to close the **Soils** dialog box and return to the **Borehole Parameters** dialog.

Follow these steps to describe the sequence of layers in a borehole:

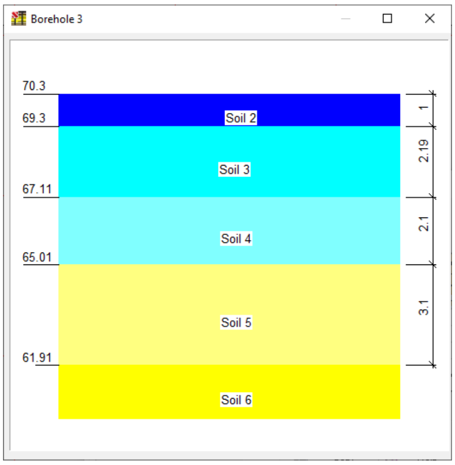
- select the number of the borehole from the **Borehole number** list;
- select the line with the soil properties of the layer in the **Soils** table (Fig. 13.7.36-1) and click the >> button to move it to the **Layers** table;
- specify the parameters of the layer (soil of the same type can occur in a few different layers).

The properties of a layer include: the level of the layer and (if necessary) the effective stress jump that can be caused, for example, by water saturation of the layer. It should be noted that the levels can be specified relatively to any reference point (e.g. the bedrock or the ground surface).

The easiest way to specify the borehole parameters is as follows: enter only one borehole and specify its parameters, then enter the rest of the boreholes. Their parameters will receive the same values as those of the first borehole by default, and you will only have to edit them. Moreover, clicking the **Copy** button opens the **Copy borehole** dialog box where you can select an identical borehole and copy the levels of its layers.

Кнопка **Предварительный просмотр** позволяет получить изображения структуры грунтового массива для выбранной скважины:

The **Preview** button allows you to get images of the soil massif structure for the selected borehole:



13.7.37 Compressible Stratum Calculation Point

If there is no bedrock, the lower boundary of the compressible stratum is calculated at the central point of the slab. You can invoke the **Compressible Stratum Calculation Point** command from the **Edit** menu and indicate a different location of the point with the cursor (it will be indicated by a red cross) .

If the shape of a foundation slab is close to a rectangle, it is recommended to place the compressible stratum calculation point in the middle of a line that connects one of the vertices of the rectangle with the intersection point of its diagonal.

13.7.38 Cancel

Clicking this button will cancel the currently invoked action, and you can use such functions as the distance measurement, construction of sections etc.

13.7.39 Section

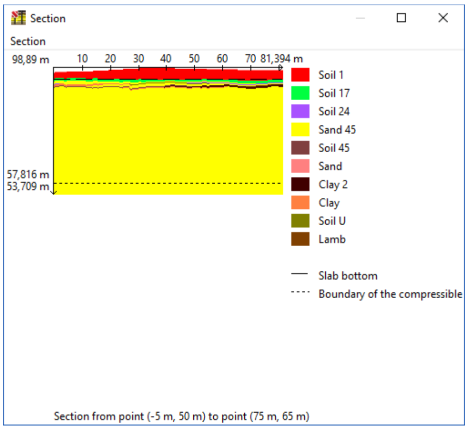


Figure 13.7.39-1. *The* **Section** *window*

To check the specified parameters of boreholes, you can use the mode for creating geological sections (Fig. 13.7.39-1). The section is generated along a linear segment drawn in any place of the site. Follow these steps to create a section:

- use the **Cancel** button to stop the current action;
- place the cursor at the beginning of the segment;
- press the **Ctrl** key and, at the same time, click and hold the left mouse button;
- holding both the mouse button and the key, drag the mouse pointer to the end point of the segment.

A window will appear with a geological section along the specified direction. The location of the slab bottom will be shown with a dashed line, and the boundary of the compressible stratum will be shown with a dotted line.

The boundary line will be visible only in the case if it lies not deeper than 10 meters from the upper level of the last soil layer.

You can use the **Save** menu item to save the picture as a Windows metafile.

The following algorithm is used to create a section:

A convex envelope is constructed for all points where the boreholes are specified. This convex envelope is triangulated. Further, one of three options is possible for every point inside the construction site:

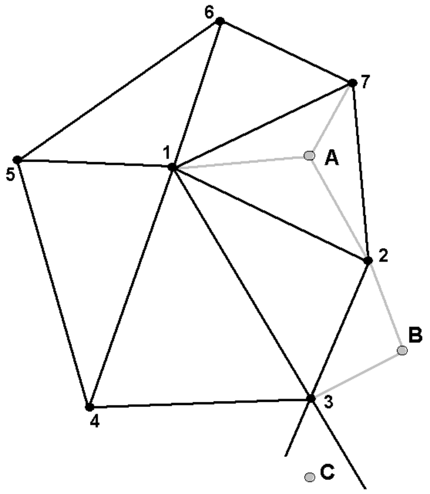


Figure 13.7.39-2.

A the point lies inside one of the triangles (an example is the point **A** in Fig. 13.7.39-2);

B the point is outside the convex envelope, and the distance from this point to the convex envelope is equal to the length of a perpendicular dropped to the nearest side of the convex envelope (an example is the point B in Fig. 13.7.39-2);

C the point is outside the convex envelope, and the distance from this point to the nearest side of the convex envelope is equal to the distance to one of the vertices (an example is the point **C** in Fig. 13.7.39-2);

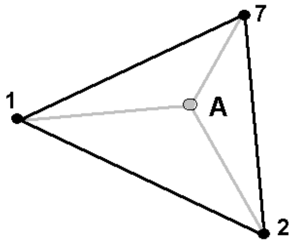


Figure 13.7.39-3.

In case **A**, the thickness of each soil layer is calculated by linear interpolation between the vertices of the triangle, proportionally to the areas of the triangles opposite to their respective vertices (Fig. 13.7.39-3).

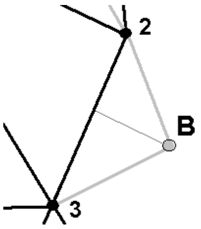


Figure 13.7.39-4.

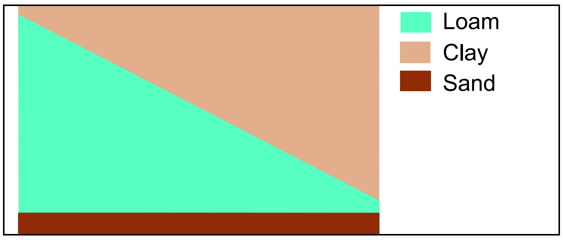
In case **B**, the thickness of each soil layer is calculated by linear interpolation of the lengths of segments the side is divided into by the perpendicular dropped from the point **B** (Fig. 13.7.39-4).

In case **C**, it is assumed that the structure of the soil stratum in the given point is the same as that in the nearest borehole (borehole 3 in Fig. 13.7.39-2).

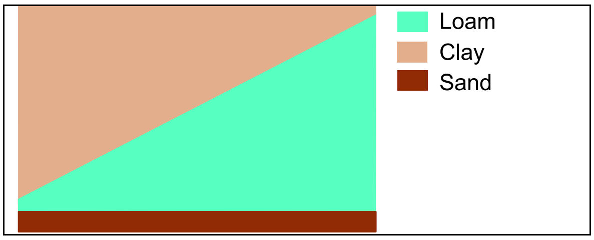
It should be noted that it is impossible to restore the structure of the soil stratum knowing only the structure of the boreholes. It can be seen from the following simple example. Suppose there are two boreholes of the following structure:

Borehole 1		Borehole 2	
Soil	**Upper boundary level**	**Soil**	**Upper boundary level**
Loam	0	Sand	0
Clay	-10	Loam	-10

A geological section can take the following form:



or



The ambiguity like this can be eliminated by specifying the same package of soils for each borehole. That is, each borehole has the same set of soils in the same order (only their upper boundary levels are different), and the lack of a soil layer (for example, i^{th}) in a particular borehole is indicated by specifying the same level for both i^{th} and $(i+1)^{th}$ soil layers. For the example above, the structure of the boreholes should be described in the following way (supposing the geological section on the right is the correct one).

Borehole 1		Borehole 2	
Soil	**Upper boundary level**	**Soil**	**Upper boundary level**
Sand	0	Sand	0
Loam	0	Loam	-10
Clay	-10	Clay	-10

13.7.40 3D Visualization of the Soil Stratum

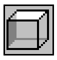
In order to obtain a 3D visualization of the soil stratum, press the respective button —  in the toolbar. A **3D** window appears (Fig. 13.7.40-1), which includes three scalable fields:

- table of soil types;
- fragmentation window;
- window with the 3D visualization of the soil stratum.

The program includes the following types of visualization of a three-dimensional model:

with *isolated layers*, where each layer has "no thickness" and is presented as a surface formed by the levels of its upper boundary (this mode is active by default);

with *3D layers* — , where each layer forms a solid mass between the levels of the upper boundary of the current and the next by depth layer.

Moreover, it is possible to display the soil stratum with the transparency effect —  (both solid and with isolated layers), to take illumination into account —  and the display the soil stratum together with boreholes — .

Display control buttons are located in the toolbar of the **3D** window.

The soils table has three columns. The first column has checkboxes which enable to disable the display of one or several soils in the 3D visualization.

The second column has color fields which correspond to the colors for displaying the layers. To change the color of the layer, place the cursor over the respective color marker and double-click the left mouse button to invoke a standard Windows dialog – **Colors**.

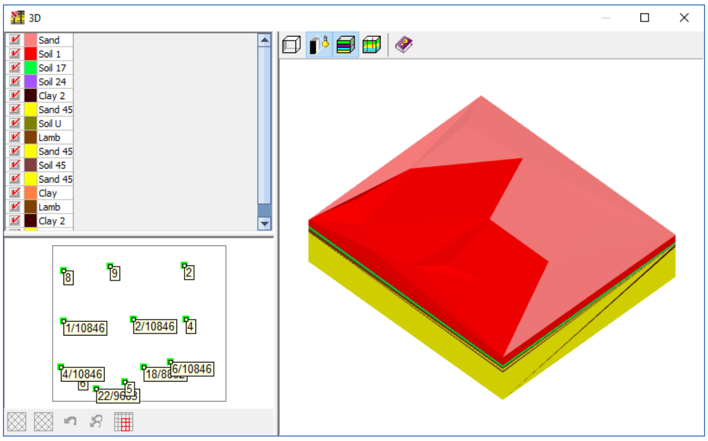


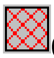
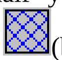
Figure 13.7.40-1. *The* **3D** *window*

The third column provides the name of the layer assigned to it when specifying the initial data.

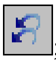
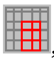
Fragmentation window enables to create any section of the soil stratum. To do this, follow these steps:

- place the cursor over the start point of the section line;
- drag the "rubber band" in the direction of the section;
- press the left mouse button again to fix the line.

It should be noted that the position of the start point and end point on the section line does not matter. Only the direction is taken into account when creating a section.

Once the last step is performed the site which includes the soil stratum will be divided by the section line into two parts – red and blue. A "half" you are interested in can be selected by clicking a button of the respective color in the toolbar — (red) and (blue), and a cutting plane displaying the soil stratum will be shown in the 3D visualization. There can be more than one cutting plane at the same time.

Besides these buttons the toolbar of the fragmentation window also contains:

- the **Undo last cut** button — ;
- the **Undo all cuts** button (returns to the initial state) — ;
- the **Navigator mode** button — , which shows the current state of the soil stratum after a series of cuts in the fragmentation window.

To change the sizes of the fields of the **3D** window, move the cursor to the border separating the fields, and after the cursor changes its shape, click and hold the left mouse and "drag" the selected border.

Click the **Close** button in the title bar of the **3D** window to exit it.

13.7.41 Distance Measurement

A distance between two points of the site can be determined with the cursor. To do this, place the cursor over the first point and left-click. Drag the pointer to the second point while holding the button. The right part of the status bar will display the distance between the points (the accuracy of this indication depends on the precision specified in the **Units of Measurement** tab of the **Settings** dialog box). Coordinates of the current position of the cursor will be displayed in the status bar.

It should be noted that in order to be able to perform this action, you have turn off all the edit modes.

13.7.42 Origin

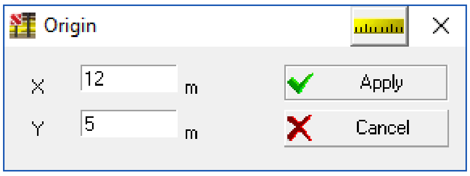


Figure 13.7.42-1. *The* **Origin** *dialog box*

This action is invoked from the **Edit** menu and enables to move the origin to a point with specified coordinates (Fig. 13.7.42-1).

The grid is rotated about its origin, therefore moving the origin can be also useful when creating the contours.

13.7.43 Grid Spacing

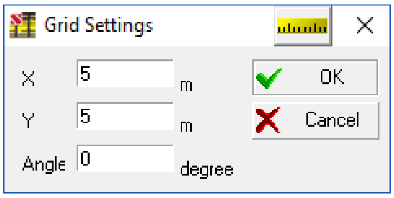


Figure 13.7.43-1. *The* **Grid Settings** *dialog box*

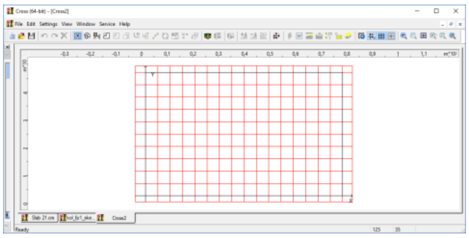


Figure 13.7.43-2. *A grid displayed in the work area*

Parameters of a coordinate grid are specified in the **Grid Settings** dialog box (Fig. 13.7.43-1), which opens once you invoke the respective command. The text fields of this dialog enable you to specify horizontal (along the **X** axis) and vertical (along the **Y** axis) grid spacing, and an angle of the grid in degrees with respect to the horizontal axis. The grid is rotated about the origin.

It should be noted that the grid spacing and its angle can be changed as many times as needed during the creation of the contours. This allows you to customize a grid in accordance with dimensions or position of the contours created.

The grid will be displayed once its parameters are entered (Fig. 13.7.43-2).

13.7.44 Grid

This button enables to display the grid in the work area. The spacing of the grid can be specified by invoking the respective action from the **Settings** menu or by clicking the  button in the toolbar.

13.7.45 Fields for Buildings

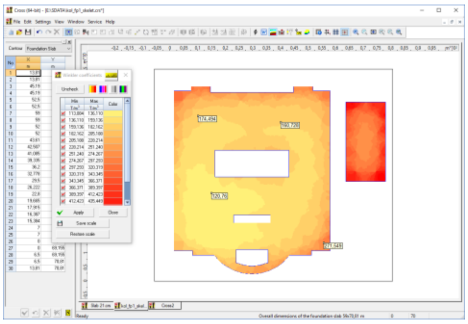
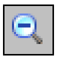


Figure 13.7.45-1. *Field of the Winkler coefficients for the foundation slab and the adjacent building*

The program enables to obtain fields for the existing buildings if the respective checkbox is checked in the **Settings** menu. As a rule, the information about the distribution of Winkler coefficients for the existing buildings is not important for the design. Nonetheless, in some cases this data may be useful.

If the **Fields for Buildings** checkbox is checked, a full picture of the distribution of Winkler coefficients for all buildings will be obtained (Fig. 13.7.45-1).

13.7.46 Zooming the Site View

A view of the site can be zoomed in. Every time you click the **Zoom In** button  — the site view is magnified by 10%. Maximum zoom is 200%. If the site view has been zoomed in, scroll bars appear at the right and bottom edges of the **Work area** allowing you to change the position of the site in the work area. The view can be zoomed out by the **Zoom Out** button . Each time you click on it the magnification is decreased by 10% until you receive the initial view.

13.7.47 Loupe Tools

This visualization command allows you to obtain a view of the selected by a rectangular marquee part of a section enlarged to the required scale. The following procedure is recommended:

- click the button  in the toolbar, **Loupe Tools** dialog box with the default scale (or the one used the last time) will appear on the screen;
- a rectangular marquee will appear once you start moving the cursor over the work area. The sizes of the marquee correspond to the scale value specified in the **Loupe Tools** dialog box (Fig. 13.7.47-1);
- move the cursor together with the marquee to the part of the site you want to view more closely, so that this area is in the middle of the marquee, and confirm this position by clicking the left mouse button;
- use the slider or the buttons «+» and «-» in the **Loupe Tools** dialog box to set the scale;
- continue working with the object.

Working with the loupe tools does not interrupt any commands, which allows you to continue working with the object after fixing the position of the marquee.

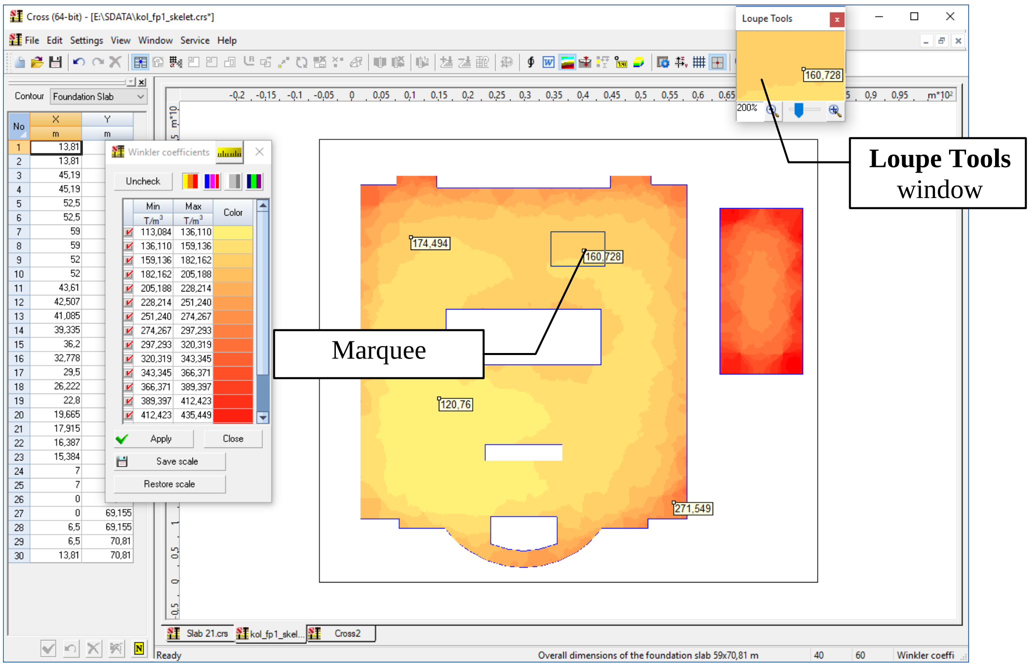
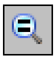


Figure 13.7.47-1. *Working with a site view in the* ***Loupe Tools*** *mode*

13.7.48 Zoom Rect

After invoking this action, you can select a part of the section by the marquee zoom and after left-clicking it will be zoomed in.

13.7.49 Zoom Initial



This command enables to obtain the initial view.

13.7.50 Help

Clicking the **Help** button invokes the standard Windows function for obtaining the help information on using the application and on its functionality.

13.7.51 About

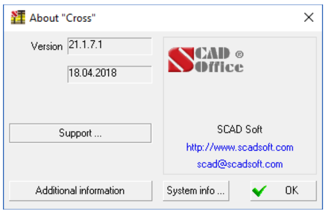
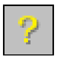


Figure 13.7.51-1. *The* **About** *information window*



Once you click on this button an **About** information window (Fig. 13.7.51-1) is invoked. It contains the information on the version and the developer of the software.

Moreover, this dialog box contains the **Additional information** button, which opens a list of modules that make up the program, and the **About the system** button, which invokes a standard Windows dialog box with the system information.

The **Support...** button enables to create a letter to the technical support service. Files with information about the operating system will be automatically attached to the letter.

13.8 References

[1] V.Karpilovsky, E.Kriksunov., A.Maliarenko, M.Mikitarenko, A.Perelmuter, M.Perelmuter, SCAD Office System SCAD. — M: ASV Publishing House, 2007.— 592 p.

[2] V.G. Fedorovsky, S.G. Bezvolev, Prediction of settlements of shallow foundations and selection of a subsoil model for the slab analysis// Foundations and soil mechanics. — 2000. — N4. — p. 10–18.

14. Pasternak

Suppose we have a horizontally layered soil deposit consisting of a finite number of layers, each layer being linearly deformable and having a constant thickness. Soil stiffness parameters in this case can be found using a technique proposed by M.I. Gorbunov-Posadov, V.Z. Vlasov, and P.L. Pasternak (see [1], [2], [3], [4], [5]). **Pasternak** enables you to determine Winkler coefficient C_1 (compression) and Pasternak coefficient C_2 (shear) using the said technique.

14.1 Working with the Program

The program's window is a two-tab dialog box. The first tab contains a table where the user needs to enter data for each layer of the soil profile.

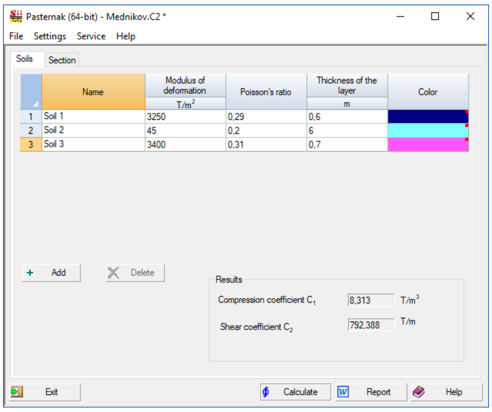


Figure 14.1-1. *The* **Soils** *tab*

Click the **Add** button to enter the properties of each soil layer (Fig. 14.1-1).

This will add a new line to the soils table, where you have to specify:

- name of the soil;
- modulus of deformation (E_0)[10];
- Poisson's ratio (ν);
- thickness of the layer (h).

Specify a color in which this soil will be displayed in the soil profile diagram in the last column of the table.

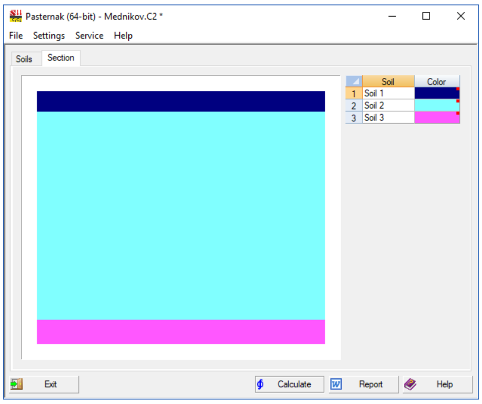


Figure 14.1-2. *The* **Section** *tab*

The second tab, **Section**, (Fig. 14.1-2) enables you to obtain a visualization of the layered soil profile.

The upper part of the dialog box contains a menu which enables to save the created model in a file (witrh the **.c2** extension), open a previously created model, or save the model under another name. The **Settings** menu enables to invoke the multi-tab **Settings** dialog box where you can customize the program (see below). The **Service** menu contains items for invoking a standard calculator of the Windows environment, a formula calculator, and a converter of units of measurement (the latter two are described in the **Appendix**). The **Help** menu provides the help information on using the application.

---

[10] Subgrade compliance (and subsoil parameters) are expressed through the soil deformation modulus if the load on the subgrade acts for a long time and the inelastic soil settlement takes place. If it is subjected to short-term dynamic loads, the subgrade compliance is defined by the modulus of elasticity of the soil. The name of the respective column (the modulus of deformation) does not change in the program, and the user has to decide on the basis of the load type whether to use the modulus of elasticity or the modulus of deformation.

14.2 Calculation

The result of the calculation includes Winkler coefficient C_1 (compression) and Pasternak coefficient C_2 (shear). The *effective* moduli of deformation are used in the analysis. These moduli can vary depending on the assumptions made about lateral strains or stresses. Therefore, the relationships implemented in the software are given below.

For each layer, the effective deformation modulus is

$$E = E_0 \frac{1-\nu}{(1+\nu)(1-2\nu)},$$

the shear modulus is

$$G = \frac{E_0}{2(1+\nu)}.$$

Let's denote the overall thickness of the layered soil deposit as $H = \sum_{i=1}^{n} h_i$ then

$$C_1 = \left(\int_0^H \frac{dz}{E(z)} \right)^{-1};$$

$$C_2 = \left(\int_0^H \frac{dz}{E(z)} \right)^{-2} \int_0^H \frac{1}{E(z)} \left(\int_0^z G(t) \left(\int_t^H \frac{ds}{E(s)} \right) dt \right) dz,$$

where $E(z), G(z)$ are effective deformation and shear moduli at the depth of z respectively.

14.3 References

[1] V.Z.Vlasov, N.N.Leontiev, Beams, Plates, and Shells on Elastic Foundation, Moscow: Fizmatgis Publishers, 1960 — 491 p.

[2] M.I.Gorbunov-Posadov, Beams and plates on an elastic base. Mashstroyizdat Publishers, 1949.

[3] I.A.Mednikov, Coefficients of subgrade reaction of a linear multilayer base, Foundations, beds, and soil mechanics, 1967, – No.4.

[4] P.L.Pasternak, On a New Method of Analysis of an Elastic Foundation by Means of Two Constants. Moscow: Gosstroyizdat Publishers, 1954 — 56 p.

[5] Y.H. Huang, Stability Analysis of Earth Slopes. M.: "Stroyizdat" Publ., 1988.

15. Slope

The analysis of slope stability is one of the most important problems in engineering geology. There are numerous methods developed for solving it within the frames of the limit equilibrium theory [3]. As a rule, these methods are based on the following assumptions.

The failure mechanism is assumed to be a slip of a soil mass with respect to a stable part of the slope. The boundary between the sliding soil mass and the stable soil is called a slip surface.

The shear resistance along the slip surface is calculated under static conditions. The Coulomb failure criterion is satisfied along the whole surface.

The actual shear stress obtained by the calculation is compared with the ultimate shear resistance, and the result of the comparison is expressed as a factor of safety *K*. For a particular slip surface, the factor of safety *K* is such a number that if the strength parameters (the internal friction angle and the effective cohesion) along the whole surface are made *K* times smaller, then the potential sliding soil mass will be in the limit equilibrium state. The factor of safety of a slope is a minimum factor of safety for all potential slip surfaces which satisfy given restrictions (the restrictions are usually defined by the method of analysis).

The real slip surface is three-dimensional. However, most methods of analysis, including the **Slope** software, assume a plane strain hypothesis where the slip surface is cylindrical with its generatrices parallel to the slope surface and the problem comes down to finding a critical directrix called a slip line. This approach is based on the hypothesis that ignoring the spatial character of the phenomenon has little effect on the stability factor of safety and affects only the strength factor of safety.

Various, usually fairly limited, classes of potential slip lines are used (such as circular arcs or logarithmic spirals). However, it is obvious that the limitations of the slip surface configuration must be minimal for essentially nonhomogeneous slopes and complicated groundwater conditions taken into account by the software. The algorithm of analysis implemented in **Slope** is based on a procedure suggested in [1], [2] which uses the method of variable level of shear strength mobilization (MVLM). Moreover, the following classic methods of the slope stability analysis are also implemented in the software:

- Fellenius;
- Bishop simplified;
- Spencer;
- Corps of Engineers №1;
- Lowe and Karafiath;
- Janbu simplified;
- Janbu corrected.

The description of these standard methods is given in most books on slope stability analysis (see, for example, [3], [4]).

It should be noted that the Fellenius, Bishop simplified and Spencer methods allow to analyze the circular slip surfaces only.

15.1 Controls

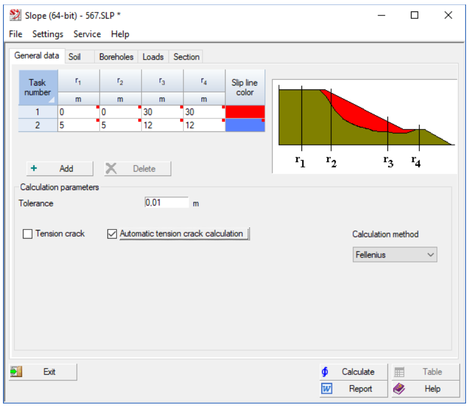


Figure 15.1-1. *The* **General Data** *tab*

Principles and means of control implemented in the software are uniform and provide a consistent interactive environment. The program uses a common multi-tab technique (Fig. 15.1-1). To activate a window, click on its tab, кроме того, используется меню.

Initial data is entered in the following tabs: **General Data**, **Soil**, **Boreholes** and **Loads**. The **Section** tab enables to perform the input data control by the geological section of the slope and to display the results of the analysis (the slip surface).

15.2 Preparing Initial Data

15.2.1 General Data Tab

The slope stability analysis and the determination of the most probable slip surface are based on the given intervals of the beginning and the end of the landslide. This tab is used to specify the sizes of the sliding soil mass. In general case, these include four numbers: r_1, r_2, which define the interval of admissible abscissas of the landslide beginning, and r_3, r_4, similar data for the end of the landslide. Since the slope may be directed either from left to right or vice versa, the 'beginning' and the 'end' of the landslide are conditional notions which may swap over when the slope goes from right to left (Fig. 15.2.1-1). Irrespective of the slope direction, the following rules must be observed when specifying the values for r: $r_4 > r_1$, $r_2 \geq r_1$, $r_4 \geq r_3$. It should be noted that the program enables to specify several variants of the r values defining several possible slip lines (and corresponding factors of safety) at different intervals of the beginning/end of the slope. The user can add/delete the variants of calculation using **Add/Delete** buttons located on the first tab.

The direction of the slope is defined by the characteristics of the boreholes (specified on the **Boreholes** tab).

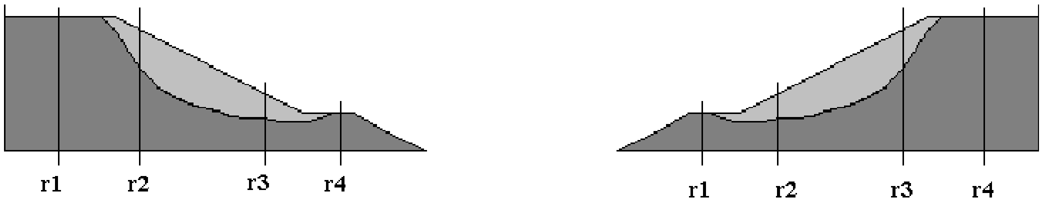


Figure 15.2.1-1. *Irrespective of the slope direction, the rules for specifying r*: $r_4 > r_1$, $r_2 \geq r_1$, $r_4 \geq r_3$

Besides the sizes of the sliding soil mass, the **General Data** tab (Fig. 15.2-1) is used to specify parameters of the analysis including the following data:

Tolerance – the accuracy for determining the points of the slip surface;

Tension Crack – the checkbox indicating the presence of a tension crack at the beginning of the landslide. If it is checked, you should also choose the marker indicating the location of the boundary of the landslide beginning (**Left** for the left-to-right slope, **Right** for the right-to-left slope). The depth of the tension crack is specified in the **Depth of Tension Crack** text field. It should be noted that the tension crack must be located at the beginning of the landslide (at a higher point of the slip line). The application does not verify the specified location of the tension crack (the *tension crack on the left/right* alternative).

Automatic Tension Crack Calculation – the checkbox indicating that the possibility of appearance (or propagation if the **Tension Crack** checkbox is checked) of a vertical tension crack at the upper part of the slip line must be taken into account in the analysis.

User can select a method of analysis from the drop-down list.

15.2.2 Soil Tab

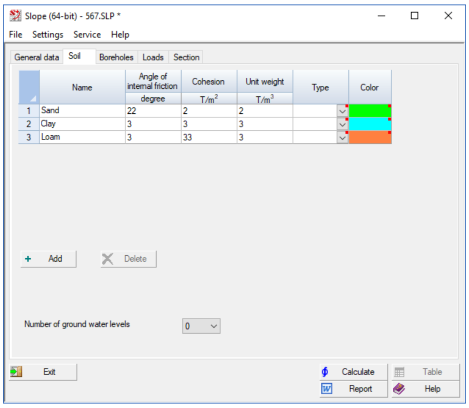


Figure 15.2.2-1. *The* **Soil** *tab*

This tab (Fig. 15.2.2-1) is used to specify soil properties. The order in which the soils are defined is of no importance. The following soil data are required for the analysis (the parentheses contain the default units of measurement):

- angle of internal friction (degrees);
- effective cohesion (tons per square meter);
- unit weight of dry soil (tons per cubic meter);
- saturated unit weight (tons per cubic meter).

Moreover, the respective item should be chosen from the list in the **Type** column for the waterproof soils. Some soil layers can have such structural properties (for example, reinforcement), that do not allow the slip line to intersect them. In this case the "No slipping" item should be chosen in the **Type** column. These soils will also be considered as waterproof. Each soil type can be assigned a particular color, which makes it easier to monitor the structure of the soil body on the **Section** tab.

The number of groundwater levels (0 to 3) is specified by choosing an appropriate number from the respective list. Each water level is specified on the **Boreholes** tab.

15.2.3 Boreholes Tab

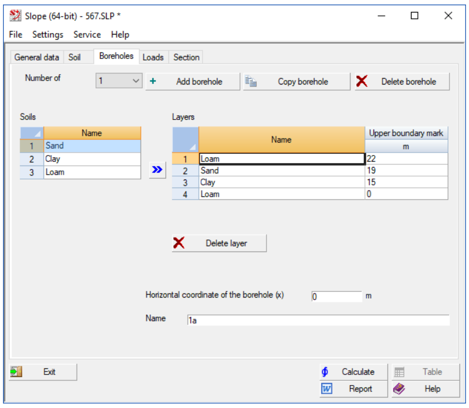


Figure 15.2.3-1. *The* **Boreholes** *tab*

The geological section of a slope is defined by specifying a set of boreholes on the respective tab (Fig. 15.2.3-1). The upper boundary marks of the uppermost soil layers define the direction and angle of the slope. The marks can be specified in any coordinate system, provided it is the same for all boreholes.

To specify the characteristics of the boreholes, follow these steps:

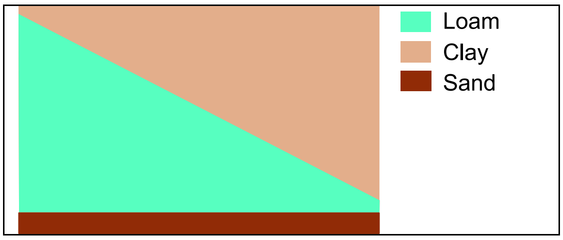
- click the **Add Borehole** button (the **Number of Borehole** list will display its current number);
- in the **Soils** table, choose a row with the name of the top soil layer of the current borehole and click the button » to move it to the **Layers** table;
- repeat the above action for each soil of the current borehole in order from the surface down;
- specify upper boundary marks for all soil layers;
- if the number of groundwater levels is not equal to zero, specify a level mark (in meters) for each level in the **Groundwater Levels** table;
- specify a horizontal coordinate of the borehole with respect to the coordinate origin of the slope (the horizontal coordinates of the boreholes must not coincide);
- specify a name for the borehole (this name will be displayed on the slope section and in the borehole description table of the report);
- return to the first step and repeat all the steps described above for each borehole.

The order in which the boreholes are described is of no importance because the snap of each borehole to the X axis is taken into account.

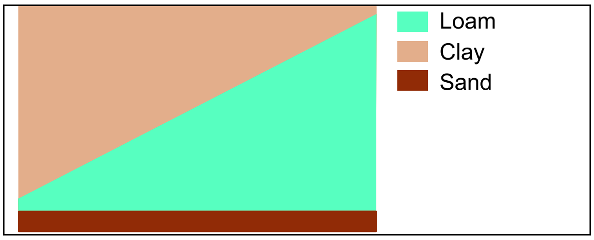
It should be noted that it is impossible to fully restore the structure of the soil mass knowing only the structure of the boreholes. It can be seen from the following simple example. Suppose there are two boreholes of the following structure:

Borehole 1		Borehole 2	
Soil	**Upper boundary mark**	**Soil**	**Upper boundary mark**
Loam	0	Sand	0
Clay	-10	Loam	-10

The geological section can be something like this:



or

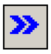


The ambiguity like this can be eliminated by specifying the same package of soil types for each borehole. That is, each borehole has the same set of soils in the same order (only their upper boundary marks are different), and the lack of a soil layer (for example, *i*-th) in a particular borehole is indicated by specifying the same level mark for both *i*-th and (*i*+1)-th soil layers. For the example above, the structure of the boreholes should be described in the following way (supposing the geological section on the right is true).

Borehole 1		Borehole 2	
Soil	**Upper boundary mark**	**Soil**	**Upper boundary mark**
Sand	0	Sand	0
Loam	0	Loam	-10
Clay	-10	Clay	-10

The geological section of a slope is defined by specifying a set of boreholes on the respective tab (Fig. 15.2.3-1). The upper boundary marks of the uppermost soil layers define the direction and angle of the slope. The marks can be specified in any coordinate system, provided it is the same for all boreholes.

To specify the characteristics of the boreholes, follow these steps:

- click the **Add Borehole** button (the **Number of Borehole** list will display its current number);
- in the **Soils** table, choose a row with the name of the top soil layer of the current borehole and click the button  to move it to the **Layers** table;
- repeat the above action for each soil of the current borehole in order from the surface down;
- specify upper boundary marks for all soil layers.

15.2.4 Loads Tab

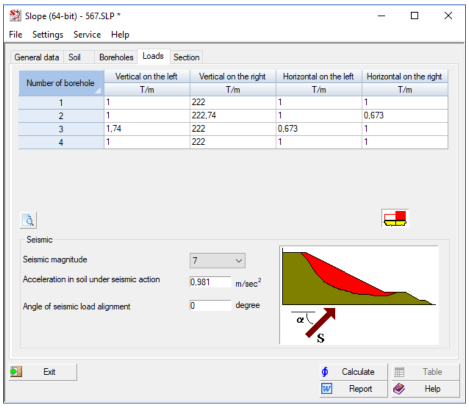


Figure 15.2.4-1. *The* **Loads** *tab*

This tab (Fig. 15.2.4-1) is used to specify loads (pressure) acting on a certain part of the slope. The application treats the problem as a two-dimensional one. It actually deals with a flat layer 1 m thick, cut out of the real slope. Therefore, the loads should be specified per one running meter of the slope width. The loads are approximated by piecewise linear functions of x coordinate and are bound to the boreholes for the convenience of description. The application deals with distributed loads directed vertically and/or horizontally at the top point of the boreholes. The default values of the load on the left and on the right of each borehole are equal to zero. Thus, if you need to specify a distributed load on a part of the slope between two adjacent boreholes, you should indicate at which borehole it begins and where it ends. The control of the specified loads is performed on the **Loads** dialog box invoked by the **Preview the Load Distribution Diagram** button.

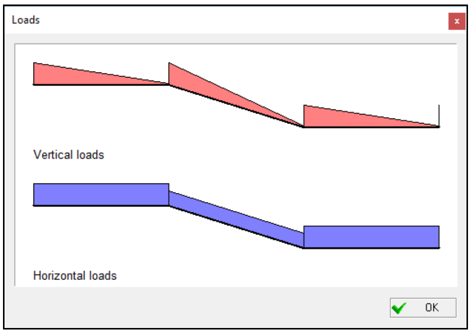


Figure 15.2.4-2. *The* **Loads** *dialog box*

For example, the trapezoid load between boreholes C2 (30 t/m) and C3 (40 t/m) specified in the table in Fig. 15.2.4-1, and displayed in the preview window, **Loads** (Fig. 15.2.4-2), is defined by the user as follows:

the *Vertical On The Left* column specifies zero for the C2 borehole; it means that there is no load to the left of the C2 borehole;

the *Vertical On The Right* column specifies 30 t/m for the C2 borehole, i.e. there is a load to the right of the C2 borehole, its initial value being 30 t/m;

the *Vertical On The Left* column specifies 40 t/m for the C3 borehole, i.e. the final value of the load on the C2-C3 segment will be 40 t/m;

the *Vertical On The Right* column specifies zero for borehole C3, which means that there is no load to the right of the C3 borehole.

In the same way, a uniformly distributed horizontal load (12 t/m) is specified between the same boreholes.

When specifying loads, follow the sign convention presented in Fig. 15.2.4-3.

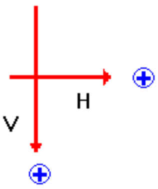


Figure 15.2.4-3. *The sign convention for specifying loads applied to a slope*

To calculate the factor of safety taking into account the seismic action, choose from the list a seismic magnitude of the site, specify the acceleration in soil under the seismic action (if its value is different from the standard one) and the angle of the seismic load application. If the calculation without the seismic load is also required, it suffices to set one of the parameters equal to zero (such as the acceleration in the soil).

The seismic load is taken into account under the assumption that the whole soil mass moves synchronously at a user-specified acceleration, and the direction of the acceleration vector conforms to the given angle of the seismic load application. Volumetric inertial forces appearing in this process are equal to the product of the specific weight and the acceleration and are included in the analysis.

15.3 Performing the Calculation and Displaying Its Results

To calculate the factor of safety, click the **Calculate** button. The calculation consists of two steps: refinement of the initial approximation, calculation of the factor of safety and plotting the slip surface. The slip surface is displayed on the slope section on the **Section** tab (Fig. 15.3-1).

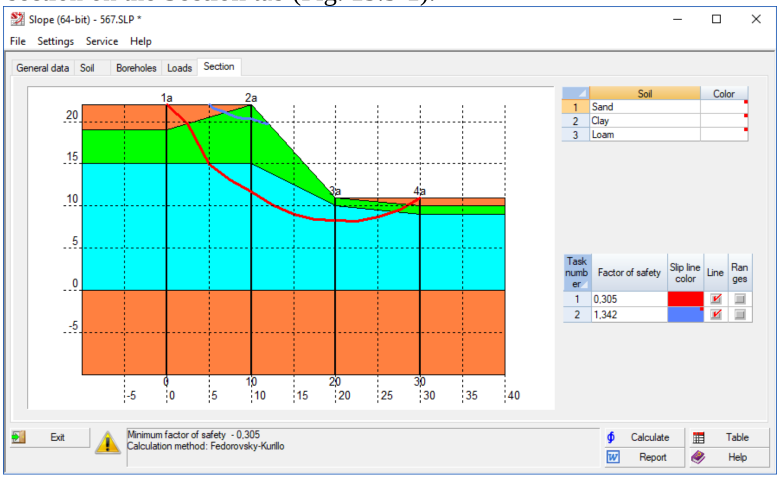
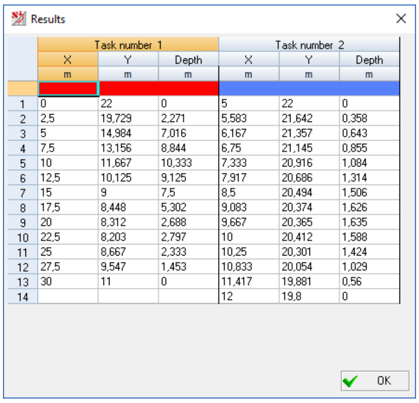


Figure 15.3-1. *The* **Section** *tab displaying a slip surface*

If several variants of calculation have been selected on the **General Data** tab, the respective table of the **Section** tab will contain factors of safety for all solved problems. You can display or hide the slip lines and the intervals for searching their beginning and end using the respective checkboxes, and you can also change the colors of these slip lines.

Moreover, you can obtain the tabular representation of the slip lines by clicking the **Table** button:


Results

	Task number 1			Task number 2		
	X	Y	Depth	X	Y	Depth
	m	m	m	m	m	m
1	0	22	0	5	22	0
2	2,5	19,729	2,271	5,583	21,642	0,358
3	5	14,984	7,016	6,167	21,357	0,643
4	7,5	13,156	8,844	6,75	21,145	0,855
5	10	11,667	10,333	7,333	20,916	1,084
6	12,5	10,125	9,125	7,917	20,686	1,314
7	15	9	7,5	8,5	20,494	1,506
8	17,5	8,448	5,302	9,083	20,374	1,626
9	20	8,312	2,688	9,667	20,365	1,635
10	22,5	8,203	2,797	10	20,412	1,588
11	25	8,667	2,333	10,25	20,301	1,424
12	27,5	9,547	1,453	10,833	20,054	1,029
13	30	11	0	11,417	19,881	0,56
14				12	19,8	0

OK

A report can be generated with the results of the calculation.

15.4 References

[1] V.G. Fedorovsky, S.V. Kurillo, Method of a variable level of shear-strength mobilization for calculation of the strength of soil masses, Soil Mechanics and Foundation Engineering, 1998, No. 4-5, p. 18-22.
[2] V.G. Fedorovsky, S.V. Kurillo, A method for analyzing the stability of slopes // Foundations and soil engineering, 1997, No. 6, pp. 95-106.
[3] Y.H. Huang, Stability Analysis of Earth Slopes. M.: "Stroyizdat" Publ., 1988.
[4] M.G. Anderson, K.S. Richards, "Slope Stability: Geotechnical Engineering and Geomorphology", 1987, John Wiley and Sons, N.Y.

The foundation is needed not because it is nice to live in a basement.
V.Boss "Lectures on mathematics: from Diophantus to Turing". V. 6 "

16. ZAPROS

ZAPROS enables you to perform the analysis of soil and foundation members according to SNiP 2.02.01-83*, KMK 2.02.01-98 and SP 50-101-2004, SNiP 2.02.03-85, KMK 2.02.03-98 and SP 50-102-2003, SP 22.13330 and SP 24.13330.2011 or DBN B.2.1-10-2009. The application also provides reference information commonly used in geotechnical engineering.

This version of the software enables you to perform the deformation analysis of rectangular foundations, determine the tilts of the foundations of this type, calculate the subsoil parameters and the ultimate pressure in the deformation analysis (design soil resistance).

Principles and means of control implemented in the software are uniform and provide a consistent interactive environment. The program uses a common multi-tab technique. To activate a window, click on its tab.

16.1 Main Window

When the application is started, the first thing that appears on the screen is its main window (Fig. 16.1-1) with a set of buttons for selecting a working mode.

Figure 16.1-1. *The main window*

Design codes for foundations and piles can be selected from the respective lists. Information on the selected code is displayed in the bottom left corner of the active mode window. If the selected design code for piles is DBN B.2.1-10-2009, then pile field tests will be performed according to DSTU B B.2.1-27:2010.

The modes are invoked by corresponding buttons and can be either reference (included in the **Information** group) or design (included in the **Analysis** group).

The **Information** group includes the following modes:

Ultimate Soil Deformations — enables to browse limit values of the relative difference of settlements, tilt, and an average or maximum settlement for various types of structures given in Annex 4 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Annex F of SP 50-101-2004, Annexes E, G, L of SP 22.13330.2011, Annexes G, E, K of SP 22.13330.2016 or Annex I of DBN B.2.1-10-2009).

Design Soil Resistance — presents information on the design resistance of various kinds of soils given in Annex 3 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Annex A of SP 50-101-2004, Annex C of SP 22.13330.2011, Annex B of SP 22.13330.2016, Annex F of DBN B.2.1-10-2009);

Soil Properties — presents information given in Annex 1 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Annex D of SP 50-101-2004, Annex B of SP 22.13330.2011, Annex A of SP 22.13330.2016, Annex C of DBN B.2.1-10-2009);

Service Factors — provides information from Table 3 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Table 5.2 of SP 50-101-2004, Table 5.4 of SP 22.13330, Table F.7 of DBN B.2.1-10-2009).

The **Foundations** group contains the following modes:

Foundation Tilt — determines the tilt of a rectangular foundation caused by loads acting on it;

Foundation Settlement — performs the deformation analysis of subsoil under pad and strip foundations, rectangular in their plane, and rigid slabs;

Subsoil Parameters — determination of the stiffness parameters of the subsoil consisting of a finite number of layers, each layer being linearly deformable and having a constant thickness;

Ultimate Pressure in the Deformation Analysis — calculates the ultimate pressure under the foundation base (design resistance of soil).

The **Piles** group includes two information modes — **Service Factors for Piles** and **Nomenclature for Piles**, together with the following design modes:

Load-Bearing Capacity of Pile — calculates the load-bearing capacity of a pile subjected to a vertical load;

Pile Analysis — calculates the stability factor of safety of the subsoil, the minimal and maximal bending moment and the shear force in a given section of the pile, and some other pile characteristics;

Settlement of the Pile — calculates the settlement of a pile subjected to a vertical load.

The **Pile Field Tests** group consists of:

Dynamic Testing of Piles — calculation of the load-bearing capacity of piles obtained from the dynamic testing;

Testing with a Sample Pile — determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the soil testing with a sample pile;

Testing with a Probe Pile — determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the soil testing with a probe pile;

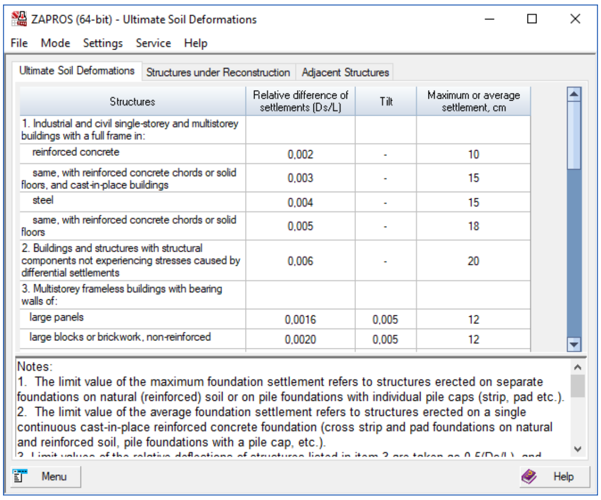
Cone Penetration Test — determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the cone penetration test.

When you invoke any of these modes, a multi-tab dialog box appears where you can enter data and browse the results.

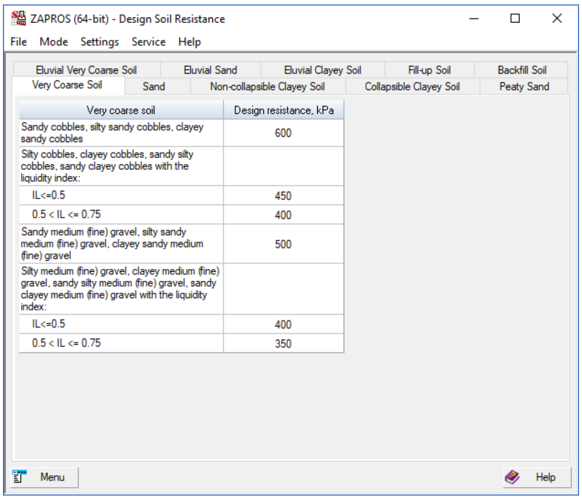
16.2 Information Modes

Reference modes provide the data given in SNiP, SP and DBN. All the values in tables are given in the same units of measurement as in SNiP, SP and DBN and do not depend on the application settings.

16.2.1 Ultimate Soil Deformations



The **Ultimate Soil Deformations** *dialog box*



The **Design Soil Resistance** *dialog box*

Figure 16.2.1-1

This mode (Fig. 16.2.1-1) provides the data given in Annex 4 of SNiP 2.02.01-83*, KMK 2.02.01-98, Annex F of SP 50-101-2004, Annexes E, G, L of SP 22.13330.2011 or Annex I of DBN B.2.1-10-2009.

16.2.2 Design Soil Resistance

Information given in this mode (Fig. 16.2.2-1) includes data from Tables 1 - 6 of Annex 3 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Annex A of SP 50-101-2004, Annex C of SP 22.13330.2011, Annex B of SP 22.13330.2016, Annex F of DBN B.2.1-10-2009), namely, the design resistance of sand, very coarse, clayey, collapsible, fill-up and backfill soils.

16.2.3 Soil Properties

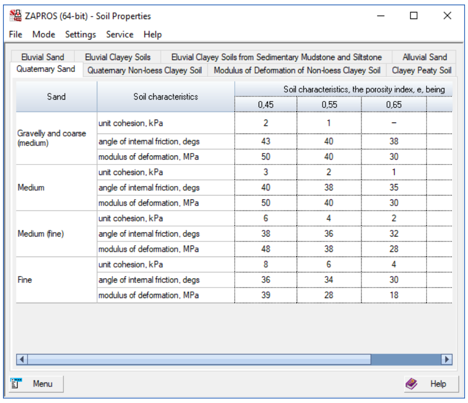


Figure 16.2.3-1. *The* **Soil Properties** *dialog box*

The information includes characteristic values of strength and strain properties of soils (Fig. 16.2.3-1), given in Tables 1- 3 of Annex 1 of SNiP 2.02.01-83*, KMK 2.02.01-98, Annex D of SP 50-101-2004, Annex B of SP 22.13330.2011, Annex A of SP 22.13330.2016, Annex C of DBN B.2.1-10-2009.

16.2.4 Service Factors

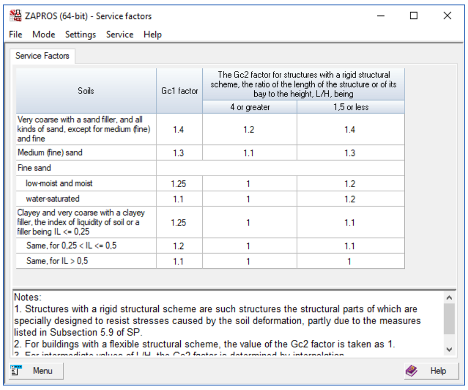


Figure 16.2.4-1. *The* **Service Factors** *dialog box*

This mode (Fig. 16.2.4-1) provides data from Table 3 of SNiP 2.02.01-83*, KMK 2.02.01-98 (Table 5.2 of SP 50-101-2004, Table 5.4 of SP 22.13330, Table F.7 of DBN B.2.1-10-2009).

16.3 Foundations

16.3.1 Foundation Tilt

General Information

This mode enables you to determine the tilt of a foundation rectangular in plan and subjected to loads from walls and columns, loads on adjacent areas, and pressure from adjacent foundations according to the requirements of SNiP 2.02.01-83*, KMK 2.02.01-98 and recommendations of "Guide to the design of foundations of structures (to SNiP 2.02.01-83)" by Gersevanov Research Institute for Foundations and Underground Structures, USSR State Committee for Construction (1986, Sec. 2.233-2.245, 2.212-2.218) [7], and SP 50-101-2004, SP 22.13330, DBN B.2.1-10-2009.

The tilt caused by the loads on the foundation is calculated both with and without taking into account the passive lateral earth pressure on the side surface of the pedestal (Section 17.2.241 of "Guide..." recommends to allow for the passive lateral earth pressure on the side surface of the pedestal for foundations the underground height of which exceeds 5 m; SP 50-101-2004 and DBN B.2.1-10-2009 do not consider the passive lateral earth pressure). The following values are determined as well:

- depth of the compressible stratum;
- bending moments at the footing bottom level;
- edge pressure under the foundation base (maximal and minimal);
- corner pressure under the foundation base (maximal and minimal);
- relative area of separation (for the case of uniaxial bending when calculating according to SP 22.13330 without taking into account the passive lateral earth pressure);
- coefficient of the non-uniform compression of soil under the foundation base;
- depth of the foundation rotation center;
- ordinates of the diagram of the passive lateral earth pressure on the side surface of the pedestal in eleven sections.

The analysis always uses the standard weighted mean value of the specific weight of the foundation, soil and floor above the footing bottom of 2 t/m^3.

All results are provided for two mutually perpendicular planes.

The mode can be used to analyze pad and strip foundations of industrial and residential buildings, or various other kinds of structures. The stiffness of structures above the foundation is not taken into account. The sizes of the foundation base are not limited. The subsoil can consist of layers of soft (non-rocky) soil of an uneven depth.

Data Preparation

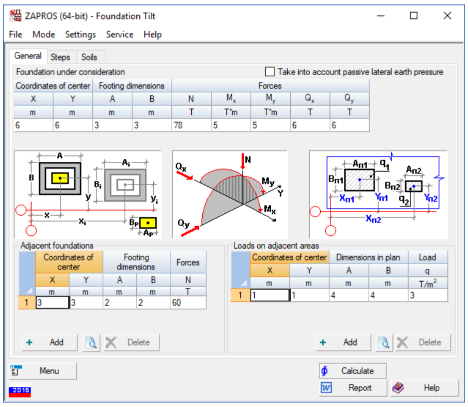


Figure 16.3.1-1. *The* **General** *tab of the* **Foundation Tilt** *dialog box*

Initial data for the analysis is entered in the **Foundation Tilt** multi-tab dialog box (Fig. 16.3.1-1), which consists of the following tabs:

General — allows to specify characteristics of the foundation under consideration and forces acting at the top of it, together with the characteristics of adjacent foundations and values of normal forces acting on the adjacent foundations at their tops. This tab is also used to specify loads on adjacent areas defined as rectangular areas. To define each area, specify coordinates of its center, sizes of the rectangle, and the value of the distributed load applied at the top of the foundation. The weight of both soil and the foundations is automatically taken into account. To determine the tilt caused by the adjacent foundations and the loads on the adjacent areas, specify at least one load. The information entered can be checked using the **Preview** buttons, .

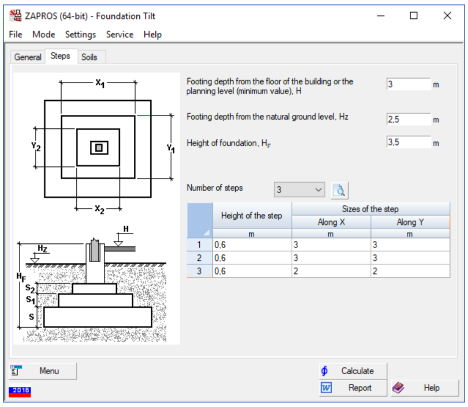


Figure 16.3.1-2. *The* **Steps** *tab*

Steps — enables to specify geometric characteristics of the foundation and its depth from the planning level (or floor) and the ground level (Fig. 16.3.1-2). The geometric characteristics include the foundation height, the number/height/sizes of the steps (for strip foundations, the number of steps is taken as one, and only the height of the first step has to be specified). In cases when the passive lateral earth pressure is not taken into account, no data on the steps is needed.

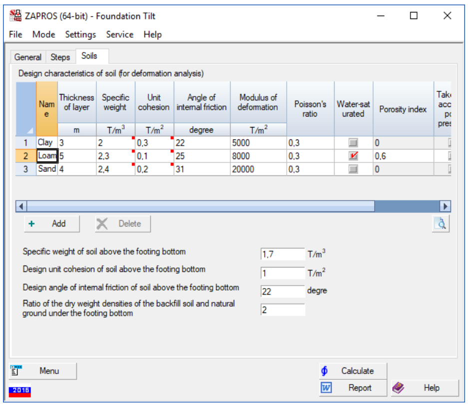


Figure 16.3.1-3. *The* **Soils** *tab*

Soils — enables to specify design properties of soil (for deformation analysis) under the footing bottom and properties of soil above the footing bottom (Fig. 16.3.1-3). If soil is water-saturated, you have to specify the specific weight of soil particles here, otherwise - the specific weight of soil. If the analysis is performed according to SP 22.13330.2016, the specific weight of soil should always be specified.

According to SP 22.13330 the groundwater pore pressure has to be taken into account. If the water-saturated soil layer satisfies the requirements of Sec. 5.6.40 of SP 22.13330, the user can check the checkbox in the "**Take into account pore pressure**" column. In this case the pore pressure at the layer border will be taken into account at the analysis of the vertical effective stress from the soil dead weight. When the calculations are performed according to SP 22.13330.2016, the pore pressure is always taken into account. The value of pore pressure is calculated according to Sec. B.1.2 SP 23.13330.2011.

When the calculations are performed according to SP 22.13330.2011, it is not recommended to check the "**Take into account pore pressure**" checkbox, because these codes require using the specific weight of saturated soils accounting for the buoyancy effect of water. And adding the pore pressure will double actual buoyancy effect of water.

The only document that provides recommendations for the analysis of the foundation tilt taking into account the passive lateral earth pressure is the Guide to SNiP 2.02.01-83 [7]. Therefore, regardless of the selected design code, the analysis in this case is performed according to the methodology given in this Guide.

To check the information entered in the listed tabs, use the **Preview** buttons. When the parameters of the foundations are being analyzed, the data check window displays the footing bottom sizes A and B of each foundation (including the current one) and their respective normal force values (Fig. 16.3.1-4).

When the loads on the adjacent areas are being analyzed, the load fields are displayed against the background of the foundations, and the dimensions of a rectangle limiting a load and its value are displayed for each load (Fig. 16.3.1-5).

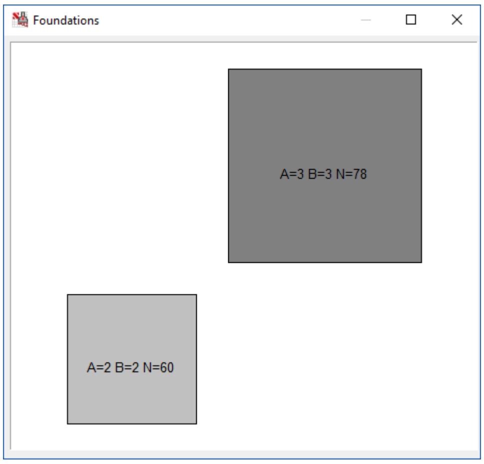


Figure 16.3.1-4. *The* **Foundations** *control window*

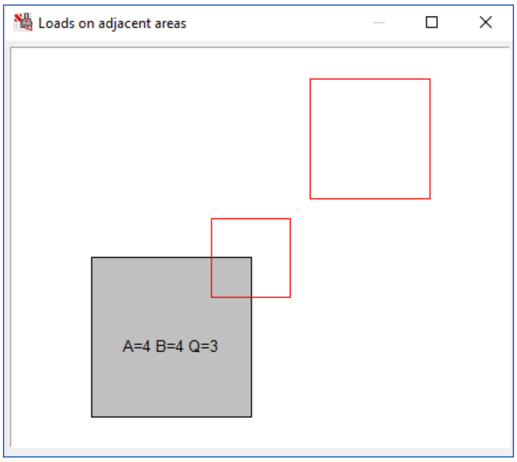


Figure 16.3.1-5. *The* **Loads on Adjacent Areas** *control window*

Results of Analysis

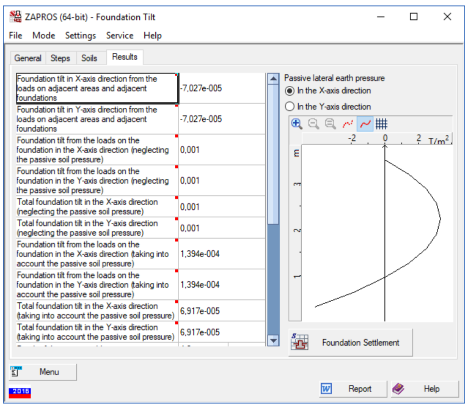


Figure 16.3.1-6. *The* **Results** *tab of the* **Foundation Tilt** *dialog box*

The analysis is performed by clicking the **Calculate** button. Results of this calculation are displayed in the specified units of measurement in the tabular form on the **Results** tab (Fig. 16.3.1-6) and include the following:

- foundation tilt in the X and Y axes directions from the loads on adjacent areas and adjacent foundations;
- foundation tilt in the X and Y axes directions from the loads on the foundation neglecting the passive soil pressure;
- total foundation tilt in the X and Y axes directions (caused by the working load on the adjacent areas, the adjacent foundations, and the loads on the current foundation) neglecting the passive soil pressure;
- foundation tilt in the X and Y axes directions from the loads on the current foundation, taking into account the passive soil pressure;

- total foundation tilt in the X and Y axes directions;
- depth of the compressible stratum;
- bending moments at the footing bottom level in the X and Y axes directions;
- values of the maximal edge pressure under the foundation base in the X and Y axes directions;
- values of the minimal edge pressure under the foundation base in the X and Y axes directions;
- maximal and minimal corner pressure under the foundation base;
- coefficient of non-uniform compression under the foundation base in the X and Y axes directions (formula (84) of the "Guide..." [7]);
- depth of the foundation rotation center in the X and Y axes directions;
- relative area of separation (for the case of uniaxial bending when calculating according to SP 22.13330).

Furthermore, if the analysis takes account of the passive lateral earth pressure, the **Results** tab will display diagrams of passive lateral earth pressure vs. depth in the X and Y axes directions. If the passive lateral earth pressure exceeds the design resistance of soil, a respective message indicating the axis will appear.

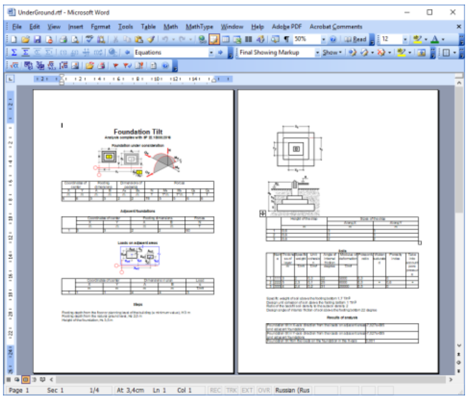


Figure 16.3.1-7. *Report generated in **MS Word***

A report is generated with the results of the analysis (the **Report** button); it contains tables with the initial data and the results of the analysis. If the analysis has taken account of the passive lateral earth pressure, the report will include a passive lateral earth pressure vs. depth diagram and a table of ordinates of the passive lateral earth pressure diagrams, in the X and Y directions, and the design resistance of soil in eleven cross-sections from top downward. The report will be opened automatically by an application associated with the format defined in the **ZAPROS** application settings (Fig. 16.3.1-7).

16.3.2 Settlement of Foundation

General Information

This mode enables you to perform a deformation analysis of pad and strip foundations rectangular in plan, and rigid slabs.

The mode calculates the values of an average settlement and subsidence of soil, checks that the pressure at the footing bottom level and that at the top of all soil layers corresponds to the design resistance of soil.

If the pressure at the footing bottom level exceeds the design value, the settlement is determined beyond the linear soil stress vs. strain relationship according to Section 2.226 of the "Guide to the design of foundations of structures (to SNiP 2.02.01-83)" [7].

Collapsible soils are classified into Type 1 and Type 2.

The deformation analysis of the subsoil takes account of the pressure from the adjacent foundations, vertical loads on the adjacent areas at the planning level, the presence of a basement, groundwater and confining beds.

It is assumed that the current and adjacent footing bottoms are located at the same level and the natural pressure at this level is the same, but the loads and sizes of the footing bottoms are different.

The analysis always uses the standard weighted mean value of the specific weight of the foundation, soil and floor above the footing bottom of 2 t/m^3.

The results include values of the deformations and a message indicating whether the requirements of the deformation analysis of the subsoil have been satisfied.

Algorigthm of the Analysis according to SNiP 2.02.01-83, KMK 2.02.01-98*

The application has been developed on the basis of Sections 2.40; 2.41; 2.48 and Annex 2 of SNiP 2.02.01-83*, KMK 2.02.01-98 and respective sections of the "Guide to the design of foundations of structures (to SNiP 2.02.01-83)" [7].

The application automatically chooses a subsoil design model – either a linearly deformable half-space or a layer of finite thickness.

It starts with an analysis that uses the linearly deformable half-space model and calculates the deformations and the depth of the compressible stratum. This model is replaced by the linearly deformable layer in two cases:

- when the compressible stratum contains a layer with $E > 10000$ tons/m^2 and its thickness satisfies Condition 32(6) of the "Guide to the design of foundations for structures";
- if both sides of the footing bottom exceed 10.0 m.

The design thickness of the linearly deformable layer is calculated (according to Sec. 2.220 of the "Guide ..."), and the moduli of deformation are analyzed within its limits. The linearly deformable layer model is adopted if within the design thickness of this layer the overall thickness of layers with the modulus of deformation $E < 1000$ tons/m^2 does not exceed 20%.

The design thickness of the linearly deformable layer is increased by the thickness of the layer with the modulus of deformation $E < 1000$ tons/m^2, if the latter layer is located below the bottom of the linearly deformable layer and its thickness does not exceed 5.0 m. If the thickness of this layer is greater, the analysis will be based on the linearly deformable half-space model.

If the footing depth from the planning level is greater than the footing depth from the ground level, then there is a fill soil layer. The pressure from the fill soil layer at any depth is taken as the weight of a column of this layer with the area of 1 square meter without decreasing with depth because the fill layer is assumed to cover a significant area. The settlement can be significantly affected by the presence of load on the adjacent areas if it is applied to a significant area. Stress caused by the dead weight of soil (natural pressure) at the footing bottom level is determined both in the case of grading by cutting (the planning level H has to be less than the ground level H_z) and in the case of grading by filling (the planning level H has to be greater than the ground level H_z).

In the analysis of settlement the depth of the compressible stratum is determined to the level at which the natural pressure is five times greater than the additional one. However, if the soil layer below that level has the modulus of deformation $E < 500$ tons/m^2, then this layer is included in the boundary of the compressible stratum. If this layer is very thick, the boundary of the compressible stratum is determined to the level at which the natural pressure is ten times greater than the additional one.

The precision of the determination of the compressible stratum depth is up to 1 mm, the defined lower layer is assumed to have a great thickness. The strength of soils at the top of all given layers, apart from the footing bottom level, is checked according to Sec. 2.48 of SNiP 2.02.01-83*, KMK 2.02.01-98. Subsoil subsidence is calculated within the given collapsible stratum. For the first type of subsidence the value of subsidence is determined only from loads on the subsoil and for all given soil layers; for the second type it is determined from the loads on the subsoil and from the dead weight of soil to the level at which the natural pressure is equal to the initial subsidence pressure, the given layer serves as a lower boundary.

When determining the subsidence factor the value of the initial subsidence pressure of the soil layers is used. For the second type of subsidence the subsidence factor is taken as 1.

For determination of the design subsoil resistance the values of φ_{II}, c_{II} and γ_{II} are taken as weighted mean for the soil layer with z thickness below the footing bottom: $z = b/2$ at $b \le 4{,}0$ m and $z=b/3$ at $b=4{,}0$ m.

Algorigthm of the Analysis according to SP 50-101-2004

The following sections have been used at the development of the software: Sec.5.5.7; 5.5.8; 5.5.10; 5.5.11; 5.5.25; 5.5.31; 5.5.32; 5.5.33; 5.5.35; 5.5.36; 5.5.37; 5.5.38; 5.5.39; 5.5.40; 5.5.41; 6.1.11; 6.1.13; 6.1.15; 6.1.17; 6.1.18.

Contrary to SNiP 2.02.01-83* the subsoil is in any case modelled as a linearly deformable half-space. Therefore for the foundations with the dimensions in plan $b>10{,}0$ m at the determination of the values of pressure the decrease of pressure due to the subtraction of the natural pressure is taken into account like for the foundations with small dimensions of the footing bottom. It is accepted that the dimensions of the excavation can be rather big.

SP 50-101-2004 (Sec. 5.5.41) assumes that the lower boundary of the compressible stratum is at the level where the natural pressure is five times greater than the additional one with the width of the foundation less or equal to 5 m (k = 0.2), or two times greater with the width greater than 20 m (k=0.5). When the width of the foundation is greater than 5 m and less than 20 m, the value of k is determined by interpolation.

The depth of the compressible stratum is assumed to be not less than b/2 at b≤ 10,0 m and (4+0,1b) at b>10,0 m.

According to Sec. 5.5.11 for the determination of the design subsoil resistance the values φ_{II}, c_{II} and γ_{II} are taken as weighted mean for the soil layer with z thickness below the footing bottom: z=b/2 at b <10,0 m and z= 4+0,1b at b≥10m.

The main difference between the analysis of settlement according to SP 22.13330 and DBN B.2.1-10-2009 and the methods described above lies in taking into account the pore pressure.

Data Preparation

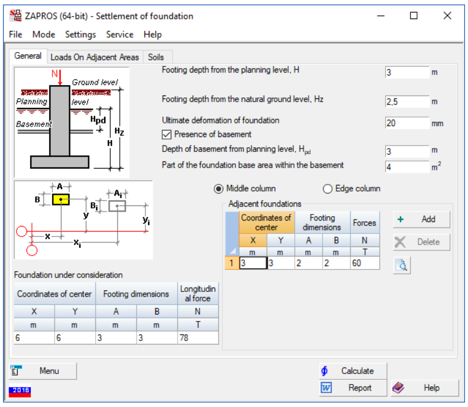


Figure 16.3.2-1. *The* **General** *tab of the* **Settlement of Foundation** *dialog box*

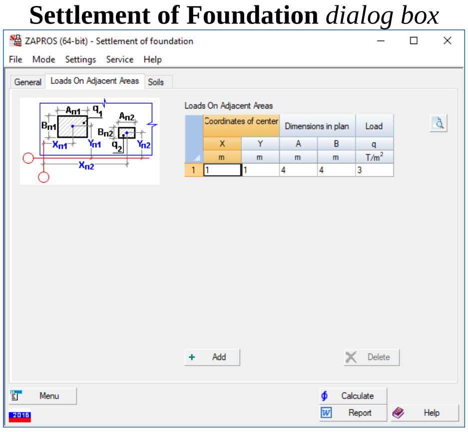


Figure 16.3.2-2. *The* **Loads on Adjacent Areas** *tab*

Initial data for the analysis is entered in the **Settlement of Foundation** multi-tab dialog box which contains the following tabs:

General (Fig. 16.3.2-1) — this tab is used to specify characteristics of the foundation under consideration and of the adjacent ones, together with normal forces, **N**, (applied at the top of the foundation) for these foundations. To check the information entered, use the **Preview** button.

Loads on Adjacent Areas — this tab is used to specify loads in the form of rectangular areas. To define each area, specify the coordinates of its center, sizes of the rectangle, and the value of the distributed load (Fig. 16.3.2-2).

Soils — this tab is used to specify design properties of soils for the deformation analysis (Fig. 16.3.2-3), and additional characteristics by slump (Fig. 16.3.2-4), the set of which depends on the subsidence type. In particular, for the type-1 subsidence the number of total pressures (from external loads and the dead weight of soil), **P**, can take values from two to five, and for the type-2 subsidence – from three to five. Moreover, the second type of subsidence requires that the first value of the relative subsidence of soil ($\boldsymbol{\varepsilon_1}$) should be one at the natural pressure.

If soil is water-saturated, you have to specify the specific weight of soil particles here, otherwise - the specific weight of soil. If the analysis is performed according to SP 22.13330.2016, the specific weight of soil should always be specified.

According to SP 22.13330 the groundwater pore pressure has to be taken into account. If the water-saturated soil layer satisfies the requirements of Sec. 5.6.40 of SP 22.13330, the user can check the checkbox in the "**Take into account pore pressure**" column. In this case the pore pressure at the layer border will be taken into account at the analysis of the vertical effective stress from the soil dead weight. When the calculations are performed according to SP 22.13330.2016, the pore pressure is always taken into account. The value of pore pressure is calculated according to Sec. B.1.2 SP 23.13330.2011.

When the calculations are performed according to SP 22.13330.2011, it is not recommended to check the "**Take into account pore pressure**" checkbox, because these codes require using the specific weight of saturated soils accounting for the buoyancy effect of water. And adding the pore pressure will double actual buoyancy effect of water.

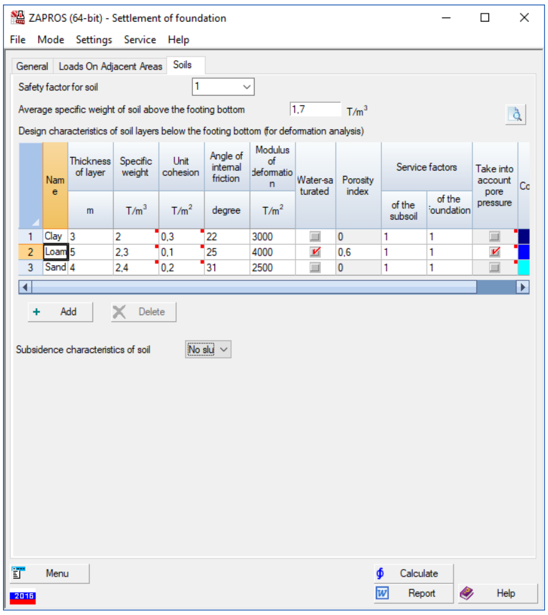


Figure 16.3.2-3. *The* **Soils** *tab without consideration of subsidence*

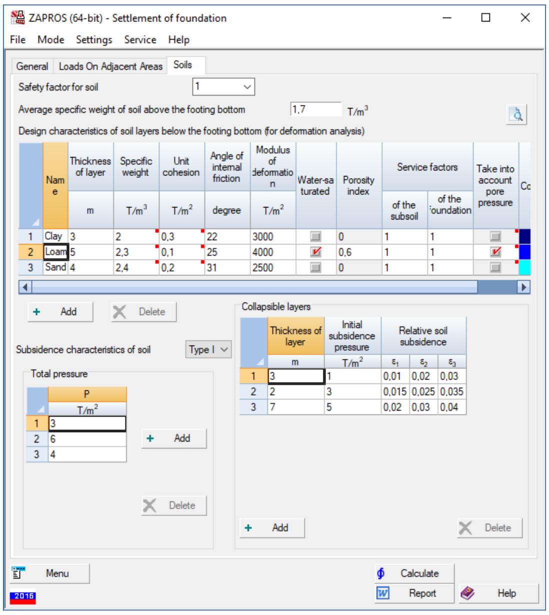


Figure 16.3.2-4. *The* **Soils** *tab with consideration of subsidence*

Information entered in the listed tabs can be checked by the **Preview** buttons similarly to the foundation tilt design mode.

Results of Analysis

The analysis is performed by clicking the **Calculate** button. Results of this calculation are displayed in the specified units of measurement in the tabular form on the **Results** tab (Fig. 16.3.2-5) and include the following values:

- design resistance of soil at the footing bottom level;
- average pressure under loads (including weight of the foundation, soil and floor) at the footing bottom level;
- foundation settlement;
- subsidence under load;
- subsidence due to the weight of soil;

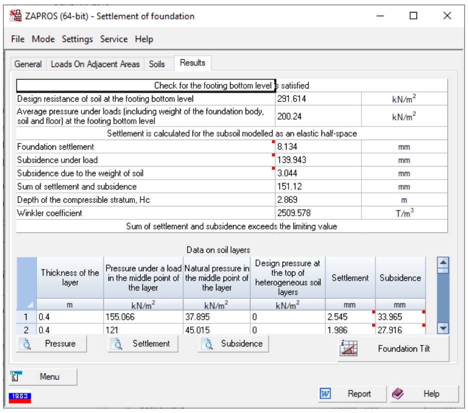


Figure 16.3.2-5. *The* **Results** *tab of the* **Settlement of Foundation** *dialog box*

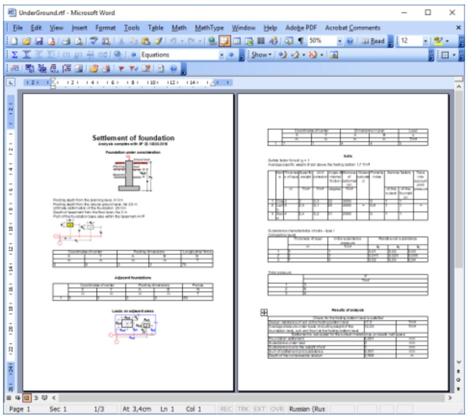


Figure 16.3.2-6. *Report generated in* ***MS Word***

- sum of settlement and subsidence;
- depth of the compressible stratum;
- Winkler coefficient.

Moreover, messages are generated which indicate the type of the subsoil design model used to determine the joint deformation of the subsoil and the structure — either a linearly deformable half-space or a linearly deformable layer, and characterize the obtained results according to various criteria, for example, “Check for the footing bottom level is satisfied (not satisfied)”, “Settlement exceeds the limiting value”, “Violated condition for design pressure of weak layer”, “Sum of settlement and subsidence exceeds the limiting value”.

Data on soil layers within the depth of the compressible stratum are additionally output (the maximum number of layers is 20). If the subsoil is modeled as an elastic half-space, the following information is output for each layer:

- thickness of the layer;
- pressure from the load in the middle point of the layer;
- natural pressure in the middle point of the layer;
- design pressure at the top level of heterogeneous soil layers;
- settlement;
- subsidence.

If the subsoil is modeled as a layer of finite thickness, the output consists of the following:

- thickness of the layer;
- pressure from the working load and adjacent foundations at the top level of the soil layer;
- design pressure at the top level of heterogeneous soil layers;
- settlement;
- subsidence.

A report is generated with the results of the analysis (the **Report** button) (Fig. 16.3.2-6).

16.3.3 Subsoil Parameters

General Information

This mode implements calculations similar to those described in Section 13 (Pasternak); the only difference is that this mode enables to select one more model – a layered half-space.

Suppose we have a horizontally layered soil deposit consisting of a finite number of layers, each layer being linearly deformable and having a constant thickness (h_i). Soil stiffness parameters in this case can be found using a technique proposed by M.I. Gorbunov-Posadov, V.Z. Vlasov, and P.L. Pasternak (see [1], [2], [4], [5]). The application suggests two modes for calculating the coefficients — by Pasternak model (Fig. 16.3.3-1) or by a layered half-space model [6] (Fig. 16.3.3-2). In both cases, the result of the calculation includes Winkler coefficient C_1 (compression) and Pasternak coefficient C_2 (shear).

The following relationships are used for the layered half-space model:

coefficient of the settlement decreasing for k-th soil layer $\gamma_k = \dfrac{4(1-2\nu_k)}{\sqrt{\pi A}(1-\nu_k)^2}$, where A is an actual support area of the structure;

constant $B_k = \begin{cases} 1 & at\ \ k=1 \\ B_{k-1}e^{-(\gamma_{k-1}h_{k-1})} & at\ \ k>1 \end{cases}$.

Then the subsoil parameters are:

$$C_1 = \sum_{k=1}^{n} \frac{E_k(1-\nu_k)\gamma_k B_k^{\ 2}}{2(1+\nu_k)(1-2\nu_k)}\left(1-e^{-2\gamma_k h_k}\right);$$

$$C_2 = \sum_{k=1}^{n} \frac{E_k B_k^2}{4(1+\nu_k)\gamma_k}\left(1-e^{-2\gamma_k h_k}\right).$$

Data Preparation and Analysis

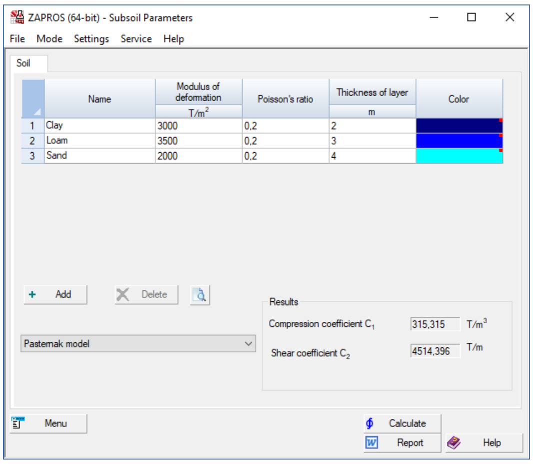


Figure 16.3.3-1. *The* **Subsoil Parameters** *dialog box* at the analysis by the Pasternak model

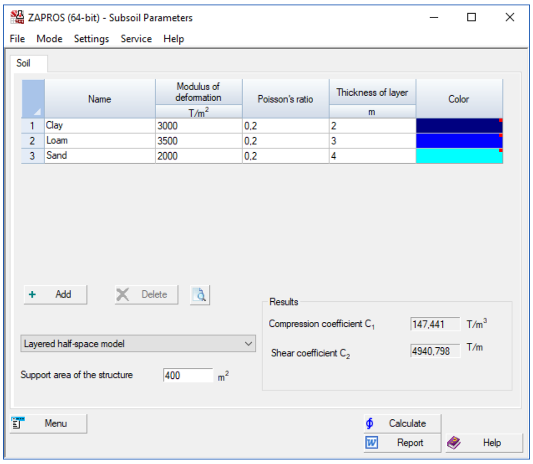


Figure 16.3.3-2. *The* **Subsoil Parameters** *dialog box* at the analysis by a layered half-space model

The model type should be selected in the respective drop-down list. Properties of the soil layers are entered into the table in order of increasing depth [11]. If the layered half-space model is used, then the support area of the structure should be specified in addition to the soil properties. Values of the subsoil parameters appear in the respective fields after clicking the **Calculate** button.

16.3.4 Ultimate Pressure in the Deformation Analysis

General Information

This mode is used to calculate the ultimate pressure under the foundation base (design resistance of soil) in a deformation analysis that models the subsoil as a linearly deformable half-space or a linearly deformable layer (Sec. 2.41 of SNiP 2.02.01-83*, KMK 2.02.01-98, Sec. 5.5.8 of SP 50-101-2004, Sec. 5.6.7 of SP 22.13330, Sec. E.4 of DBN B.2.1-10-2000).

[11] Subgrade compliance (and subsoil parameters) are expressed through the soil deformation modulus if the load on the subgrade acts for a long time and the inelastic soil settlement takes place. If it is subjected to short-term dynamic loads, the subgrade compliance is defined by the modulus of elasticity of the soil. The name of the respective column (the modulus of deformation) does not change in the program, and the user has to decide on the basis of the load type whether to use the modulus of elasticity or the modulus of deformation.

Data Preparation and Analysis

Initial data for the analysis is specified in the **Ultimate Pressure in the Deformation Analysis** dialog box (Fig. 16.3.4-1), which contains five groups of data.

The **Design characteristics of soil** group includes two radio buttons for selecting a method of obtaining the characteristics — either from code tables or from the results of testing.

The **Service factors** group contains drop-down lists for selecting values for factors γ_{c1} and γ_{c2}. A table that defines the values of these factors depending on the type of soil and the ratio of the structural dimensions is given in the respective information mode. These factors can be calculated in the **Service Factors** dialog box (Fig. 16.3.4-2) invoked by the button on the right from the drop-down lists.

The **Foundation**, **Basement**, and **Soil characteristics** groups are used to specify the respective data. An averaged design value of the specific weight of the soil pack below and above the footing bottom can be obtained with the **Soil Pack** "calculator" (Fig. 16.3.4-3), invoked by clicking the button on the left from the text field.

To define a soil pack, use the **Add** button to add a necessary number of rows to the table and then specify the thickness and specific weight of each soil layer. Click the **Apply** button; this will close the dialog and enter the specific weight value into the respective text field.

The analysis is performed by clicking the **Calculate** button.

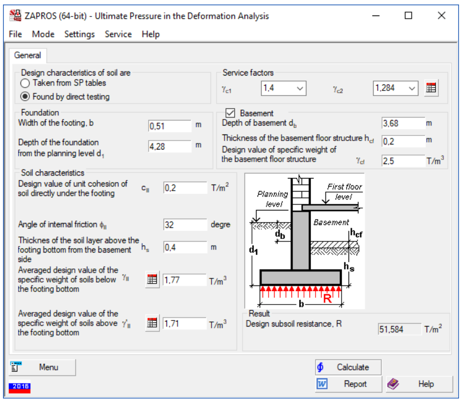


Figure 16.3.4-1. *The* **Ultimate Pressure in the Deformation Analysis** *dialog box*

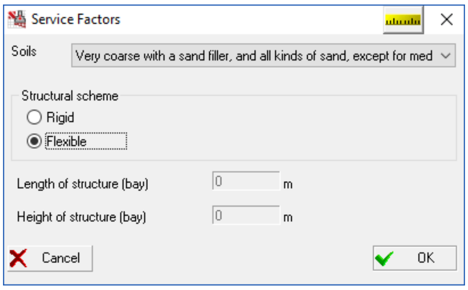


Figure 16.3.4-2. *The* **Service Factors** *dialog box*

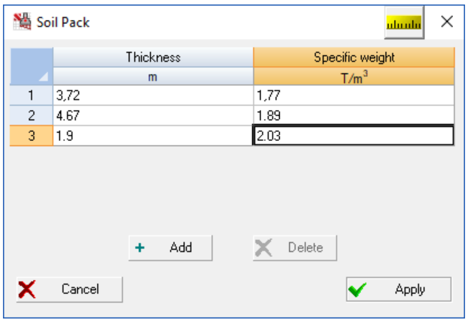


Figure 16.3.4-3. *The* **Soil Pack** *dialog box*

Limitations

The requirements of Sec. 2.41 of SNiP 2.02.01-83*, KMK 2.02.01-98 are not implemented in the analysis — it is assumed that the width of the basement does not exceed 20 m.

16.4 Piles

This group includes reference and design modes dealing with the design of piles.

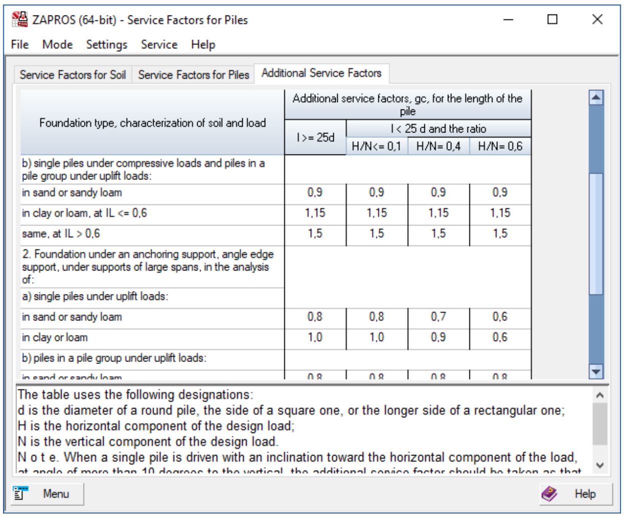


Figure 16.4-1. *The* **Service Factors for Piles** *dialog box*

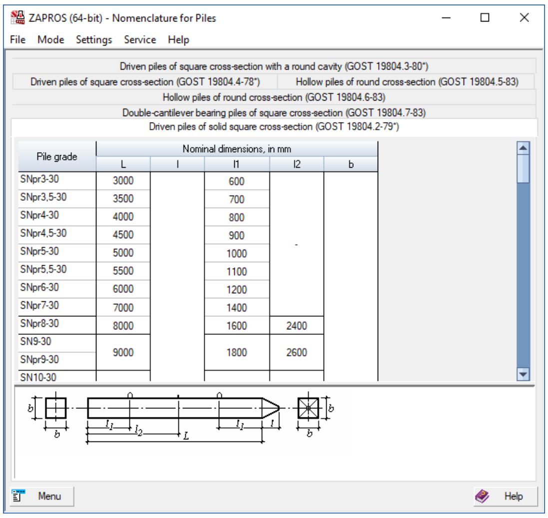


Figure 16.4-2. *The* **Nomenclature for Piles** *dialog box*

16.4.1 Service Factors for Piles

This mode provides information (Fig. 16.4-1) on service factors for soil (Table 3 of SNiP 2.02.03-85, KMK 2.02.03-98, Table 7.3 of SP 50-102-2003, Table 7.4 of SP 24.13330.2011), service factors for piles (Table 5 of SNiP 2.02.03-85, KMK 2.02.03-98, Table 7.5 of SP 50-102-2003, Table 7.6 of SP 24.13330.2011), and additional service factors (Table 19 of SNiP 2.02.03-85, KMK 2.02.03-98, Table 13.1 of SP 50-102-2003, Table 14.1 of SP 24.13330.2011). If the selected design code is DBN B.2.1-10-2009, the data on the service factors for piles is given according to the change № 1 to this DBN.

16.4.2 Nomenclature for Piles

This mode provides tables (Fig. 16.4-2) with grades and properties of solid and hollow piles of square and round cross-sections complying with GOST 19804.3-80*, 19804.4-78*, 19804.5-83*, 19804.6-83*, 19804.7-83*, 19804.2-79*.

16.4.3 Analysis of the Pile Load-Bearing Capacity

This mode enables you to determine the load-bearing capacity of end-bearing and friction piles subjected to vertical compressive load, F_d, and vertical uplift load, F_{du}. Driven, cast-in-place, drilled, and shell piles are considered according to the requirements of Section 4 of SNiP 2.02.03–85 (Section 7.2 of SP 50-102-2003 or SP 24.13330.2011, Section 8.1 of "Instruction Manual for Design and Engineering of Pile Foundations for Structures in the City of Moscow"), and requirements of Section 5 of "Guide to the design of pile foundations". When calculating the load-bearing capacity of piles, peculiarities of the design for seismic regions are taken into account (Section 11 of SNiP 2.02.03–85, Section 12 of SP 50-102-2003 or SP 24.13330.2011, and Section 12 of "Guide to the design of pile foundations"). If the selected design code is DBN B.2.1-10-2009, then the requirements of Sec. N.7 of Annex N of this DBN are taken into account.

The analysis takes into account that the pile may have a pedestal at its tip and that a soil plug may be left intact or fully removed from the shell pile when its hollow core is filled with concrete. The possibility of grading the adjacent area (by cutting, filling, or hydraulic filling) and the presence of excavation in the location of the pile installation are also taken into account. The service factors for the pile in soil, γ_c, under the tip of the pile, γ_{cr}, and on the side surface of the pile, γ_{cr}, are specified by the user (it is also possible to select their values from an available set).

Data Preparation

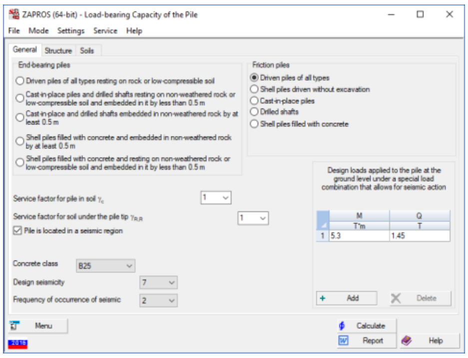


Figure 16.4.3-1. *The* **General** *tab of the* **Load-bearing Capacity of the Pile** *dialog box*

The **General** tab (Fig. 16.4.3-1) enables you to specify a pile type in the **End-bearing piles** or **Friction piles** groups. Depending on the pile type, the values of a safety factor for soil, γ_g, service factor for pile in soil, γ_c, and for soil under the pile tip, γ_{cR}, are selected from the drop-down lists.

If the construction site is in a seismic region and a friction pile is specified, you should check a respective checkbox and use the drop-down lists which will appear to select a concrete class, design seismicity of the site, and the frequency of occurrence of seismic actions, and then specify in the table the values of design loads (M and Q) applied to the pile at the ground level in a special combination of loads taking into account the seismic action. No additional data is required for an end-bearing pile, and it suffices to check the checkbox to allow for the seismic region.

It should be noted that the current edition of SNiP II-7-81* "Construction in seismic regions" does not provide any information on the frequency of occurrence of earthquakes, although Table 12.1 of the code contains a reference to such data. The data should, probably, be taken from the old edition of SNiP II-7-81*.

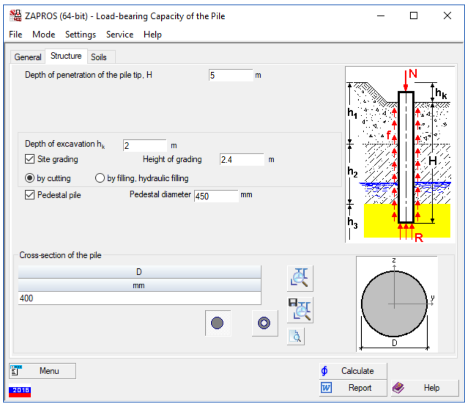


Figure 16.4.3-2. *The* **Structure** *tab*

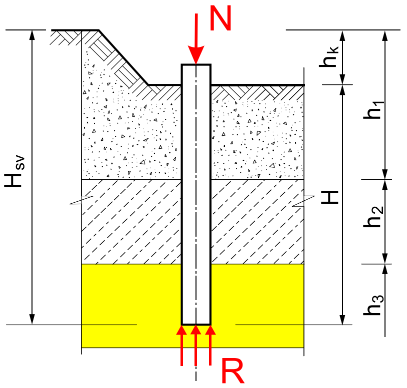
The **Structure** tab (Fig. 16.4.3-2) is used, depending on the selected pile type, to define the cross-section of the pile and its sizes and to specify some additional data. For various types of end-bearing piles, these may include the depth of pile embedment in rock or, if the pile is hollow, the height of the filled part of the pile hollow core.

The additional data for friction piles also includes the depth of penetration of the pile tip, the depth of excavation, parameters of the site grading; for a pedestal pile, the data includes the diameter of the pedestal and the type of connection between the pile cap and the pile (either hinged or rigid).

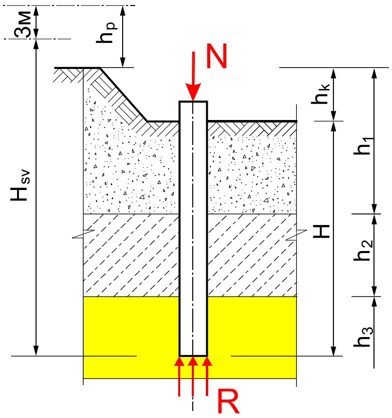
When specifying sizes of the pile cross-section, you can save them under a unique name in the database (use the button ), or to retrieve from the database (use the button ). The section can be checked by clicking the **Preview** button .

Глубина погружения нижнего конца сваи при отсутствии планировки территории (а также при планировке территории срезкой, подсыпкой, намывом до 3 м) отсчитывается от уровня природного рельефа (уровня земли), а при срезке, подсыпке, намыве от 3 до 10 м – от условной отметки, расположенной соответственно на 3 м выше уровня срезки или на 3 м ниже уровня подсыпки, намыва.

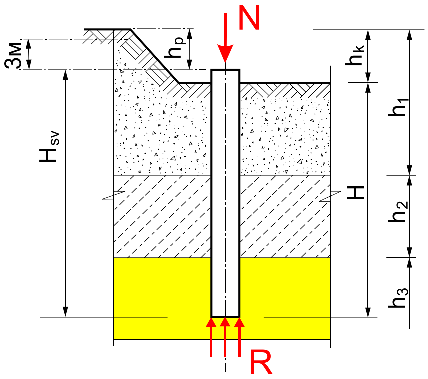
The depth of penetration of the pile tip when there is no grading (as well as when grading the area by cutting, filling, or hydraulic filling up to 3 m) is measured from the ground level, and in the case of cutting, filling, or hydraulic filling from 3 to 10 m – from a certain level 3 m above the level of cutting or 3 m below the level of filling, hydraulic filling respectively.



No grading and grading with h_p<3m



Grading by filling, or hydraulic filling (h_p>3m)



Grading by cutting (h_p>3m)

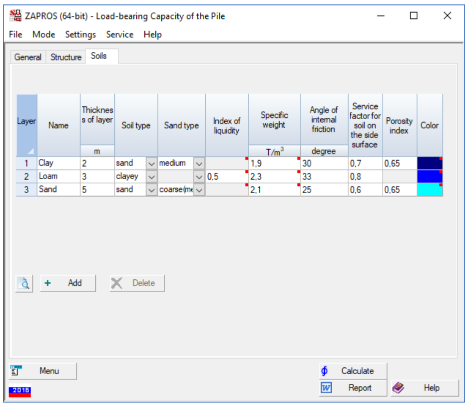


Figure 16.4.3-3. *The* **Soils** *tab*

Soil properties are specified in the table of the respective tab (Fig. 16.4.3-3). To enter the properties of a new soil layer (including the first one), click the **Add** button and a new row will be added to the table. Select a soil type from the list. If sand is selected, its type can be specified in the drop-down list of the **Sand type** column.

Index of liquidity should be specified for clayey soils. To delete a row or several consecutive rows, select them (place the mouse pointer over the number of layer, click and hold the left mouse button, and drag the pointer across the numbers of other layers to be deleted) and click the **Delete** button.

Since at the analysis of the load-bearing capacity SNiP requires information on whether a layer of sand is dense, the soil table for sand requests information on the porosity index. The conclusion on density is made on the basis of recommendations from Table 10 of Guide to SNiP [7].

If the friction pile is located in a seismic region, you have to specify additional information for sand soil: whether its layers are water-saturated or not (use the **Water-saturated** checkbox in the respective rows of the soil table).

The given soil pack can be checked by the **Preview** button .

Results of Analysis

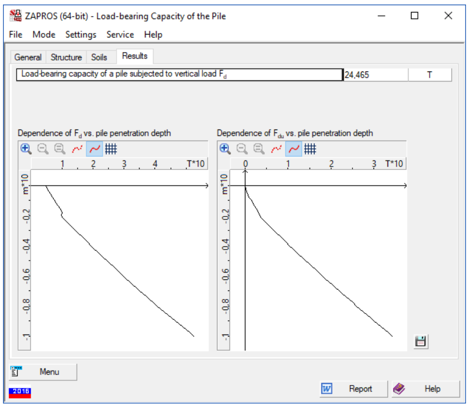


Figure 16.4.3-4. *The* **Results** *tab of the* **Load-bearing Capacity of the Pile** *dialog box*

The analysis is performed by clicking the **Calculate** button. Results of this calculation are displayed in the specified units of measurement in the tabular form on the **Results** tab (Fig. 16.4.3-4) and include the following:

a) for an end-bearing pile – load-bearing capacity of a pile subjected to vertical load, F_d;

b) for a friction pile – load-bearing capacity of a pile subjected to vertical load, F_d, and load-bearing capacity of a pile subjected to uplift load, F_{du}. Moreover, diagrams of the relationship of F_d and F_{du} vs. the depth of penetration of the pile are built for friction piles.

A report is generated with the results of the analysis (the **Report** button), see Section 2.5.

Limitations

1. Driven piles with their tips resting on loose sand or clayey soils with the index of liquidity I_L > 0,6 are not considered.
2. The current version does not consider pyramidal, trapezoid, rhombic, cylinder, or screw-down piles.
3. Notes 4–7 to Table 1 (Table 7.1 of SP 50-102-2003, Table 7.2 of SP 24.13330.2011), notes 3–4 to Table 2 (Table 7.2 of SP 50-102-2003, Table 7.3 of SP 24.13330.2011), and notes to Table 7 of SNiP 2.02.03–85 (Table 7.7 SP 50-102-2003, table N.3.3 of DBN B.2.1-10-2009) are not implemented.
4. Negative soil friction forces on the side surface of the piles are not taken into account.

Load-Bearing Capacity of a Pile Based on the Resistance of the Shaft Material

The check of the load-bearing capacity of a pile based on the resistance of the shaft material should comply with Sections 3.7 and 3.8 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.1.8, 7.1.9 of SP 50-102-2003 or SP 24.13330.2011). Only the first paragraph of Sec. 3.7 is taken into consideration, and the length of the pile segment from the free-standing pile cap base to the planning level is taken as zero (l_0=0).

The strength check of the pile material is performed in the **Resistance of R/C Cross-Sections** mode of **ARBAT**. **ZAPROS** sends to **ARBAT** the type of cross-section and the sizes of the pile together with the effective length and properties of heavy concrete taking into account a service factor that describes the effect of a particular piling method.

The following sequence of steps is recommended for this mode:

- initialize the **Load-Bearing Capacity of the Pile** mode in **ZAPROS**;
- enter the initial data and perform the analysis;
- click the **Save Data** button in the **Results** tab for the analysis of the load-bearing capacity based on the material in **ARBAT**;
- in the **Additional Data** dialog box specify a method of piling and click **OK**;
- in the dialog that opens, specify a directory and a filename (with the **.sav** extension) to save data necessary to check the load-bearing capacity;
- start **ARBAT** and initialize the **Resistance of R/C Cross-Sections** mode;
- select the **Open** item from the **File** menu and open the file previously generated and saved by **ZAPROS**;

- add missing data necessary for the check of the pile cross-section (class of longitudinal and transverse reinforcement; thickness of the concrete cover; arrangement, diameter and number of longitudinal rebars; diameter, spacing, and number of transverse rebars);
- perform the analysis of the interaction curves.

The check of the pile shaft material is performed as for a reinforced concrete element according to requirements of SNiP or SP regulating the analysis of reinforced concrete structures (see Sec. 3.6 of SNiP 2.02.03-85, KMK 2.02.03-98, Sec. 7.1.7 of SP 50-102-2003 or SP 24.13330.2011).

16.4.4 Analysis of Pile

This mode is used to analyze piles for a combined action of vertical and horizontal forces and moments in compliance with requirements of SNiP 2.02.03–85 (SP 50-102-2003 or SP 24.13330.2011). The analysis takes into account the possibility of the development of the first and second phase of the soil stress-strain state according to the recommended Annex 1 of SNiP 2.02.03–85, Annex E of SP 50-102-2003, Annex C of SP 24.13330.2011, and peculiarities of pile design for seismic regions. The mode calculates the load-bearing capacity of piles if there is the possibility of development of the second phase of the soil stress-strain state, the subsoil stability, and deformations of the piles including determination of the horizontal displacement of the pile head and its rotation angle. In the analysis according to Annex E of SP 50-102-2003 (Annex C of SP 24.13330.2011, Sec. 8.5.4.9 and Sec. N.8 of Annex N of DBN) only the first phase of the soil stress-strain state is considered (soil surrounding the pile is treated as an elastic, linearly deformable medium).

Only one loading plane is used in the analysis. The analysis takes into account the following: pile cap type (free-standing or ground-contacting), connection between the pile cap and the pile (hinged or rigid), arrangement of piles in a foundation with pile cap (single-row or multiple-row). Behavior of a pile in a pile group is not considered (Section 11 of Annex 1 of SNiP 2.02.03–85 and similar sections of SP).

There are no specific formulas for the analysis of pile deformations, values of moments and shear forces etc. in SP 24.13330.2011. Authors of SP 24.13330.2011 suggest analyzing a pile as a beam on elastic subgrade with a subsoil parameter the value of which should be determined according to the Annex C of SP 24.13330.2011. The problem on the behavior of a beam on elastic subgrade does not have a precise analytical solution, therefore an approximate solution given in Annex 1 of SNiP 2.02.03–85 is used in **ZAPROS**.

Data Preparation

The **General** tab (Fig. 16.4.4-1) enables you to specify a pile type in the **End-bearing piles** or **Friction piles** groups. The following data are specified for all types of piles:

- safety factor γ_k (the default value is 1.4);
- design loads applied to the pile at the ground level and load safety factor;
- fraction of the temporary part in the total moment in the foundation cross-section at the level of the pile tip.

Other required data includes the concrete class for the pile, arrangement of piles in a foundation with pile cap (single-row or multiple-row), pile cap type (free-standing or ground-contacting). For all types of piles except for driven ones, it is possible to take into account the embedment of the pile tip.

If the construction site is in a seismic region, you should check a respective checkbox and specify in the table the values of design loads applied to the pile at the ground level in a special combination of loads taking into account the seismic action.

Special checkbox allow to take into account change No. 2 to SP 24.13330.2011.

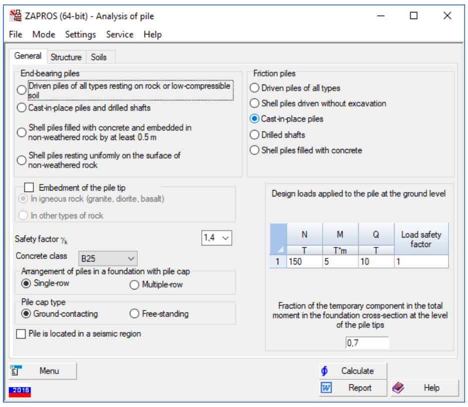


Figure 16.4.4-1. *The* **General** *tab of the* **Analysis of Pile** *dialog box*

The **Structure** tab (Fig. 16.4.4-2) is used to assign, depending on the selected pile type, the cross-section and its sizes (the driven piles can have a rectangular, Tee, I-beam, square with a round hollow core, circular or ring section; other types of piles can have either an annular or a round section), to specify the type of connection between the pile cap and the pile (either hinged or rigid), and to enter the following additional data:

- depth of penetration of the pile tip;
- distance from the pile cap base to the ground;
- depth of excavation;
- ultimate bending moment resisted by the pile cross-section, allowing for the longitudinal forces (only if the piles are arranged in multiple rows in a foundation with pile cap).

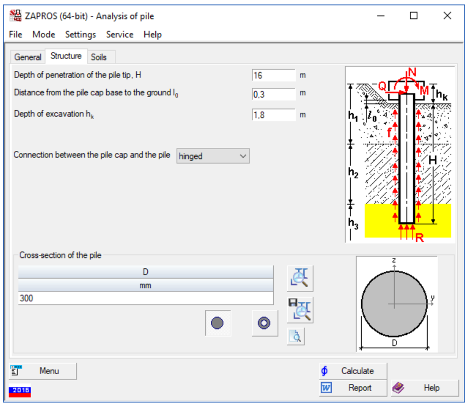


Рис. 16.4.4-2. *The* **Structure** *tab*

For sections with different geometric characteristics in different planes of loading, the analysis should be done separately for each plane, specifying the respective values of loads in the table on the **General** tab. The load plane is assigned using buttons of the respective group.

Soil properties are specified in a table on the respective tab in the same way as in the **Load-bearing Capacity of the Pile** mode.

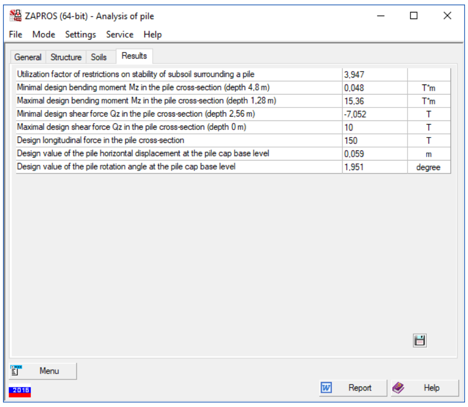


Figure 16.4.4-3. *The* **Results** *tab*

When specifying sizes of the pile cross-section, you can save them under a unique name in the database (use the button ), or to retrieve from the database (use the button ).

The section can be checked by clicking the **Preview** button .

Results of Analysis

The analysis is performed by clicking the **Calculate** button. Results of this calculation are displayed in the specified units of measurement on the **Results** tab (Fig. 16.4.4-3) and include the following:

- design moment in the rigid joint between the pile and the pile cap;
- utilization factor of restrictions on stability of subsoil;

- minimal design bending moment in the pile cross-section;
- maximal design bending moment in the pile cross-section;
- minimal design shear force in the pile cross-section;
- maximal design shear force in the pile cross-section;
- design longitudinal force in the pile cross-section;
- design value of the pile rotation angle at the pile cap base level;
- design value of the pile horizontal displacement at the pile cap base level;
- utilization factor of the pile load-bearing capacity;
- design value of the pile rotation angle at the ground level;
- design value of the pile horizontal displacement and the ground level.

There is a possibility to transfer the data to **ARBAT** for the analysis of the pile load-bearing capacity based on the material in the same way as in the **Load-bearing Capacity of the Pile** mode.

16.4.5 Settlement of the Pile

This mode enables you to determine the settlement of a single pile according to the recommended Annex 4 of SNiP 2.02.03–85 (Annex I of SP 50-102-2003, Sec. 7.4.2, 7.4.3 of SP 24.13330.2011). The mode considers both single piles with and without pedestals.

According to the analysis given in Section 9.2-9.4 of "Instruction Manual for Design and Engineering of Pile Foundations for Structures in the City of Moscow" [3], the settlement of a single pile is determined without taking into account the pedestal (the **Pedestal pile** checkbox must be unchecked).

Data Preparation

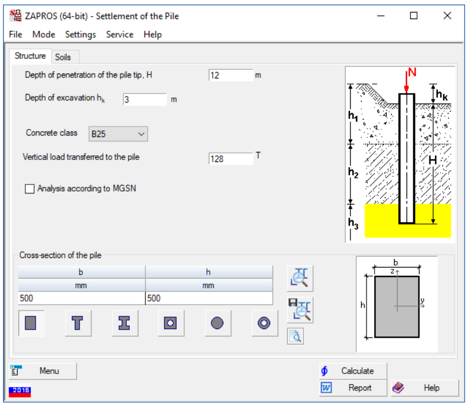


Figure 16.4.5-1. *The* **Structure** *tab of the* **Settlement of the Pile** *dialog box*

The **Structure** tab (Fig. 16.4.5-1) is used to assign a pile cross-section (rectangle, Tee, I-beam, square with a round hollow core, circular, or ring) and its sizes, and to enter the following additional data:

- depth of penetration of the pile tip;
- depth of excavation;
- concrete class;
- vertical load transferred to the pile.

A pile pedestal can be taken into account for piles of round and ring cross-section by checking the **Pedestal pile** checkbox. Dimensions of the pedestal are specified in the **Pedestal diameter** field.

When specifying sizes of the pile cross-section, you can save them under a unique name in the database (use the button ), or to retrieve from the database (use the button ). The section can be checked by clicking the **Preview** button .

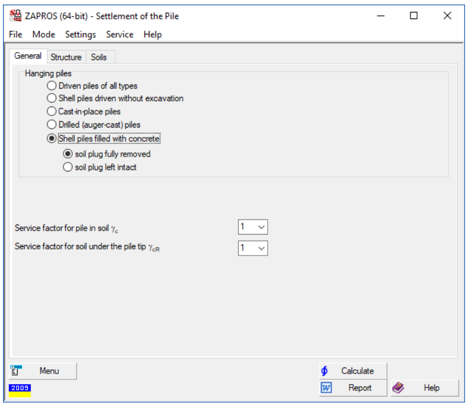


Figure 16.4.5-2. *The* **General** *tab of the* **Settlement of the Pile** *dialog box*

If the analysis is performed according to DBN, there is an additional **General** tab (Fig. 16.4.5-2), where the user can specify the pile type and service factors.

Soils are defined according to the same rules as in the **Load-bearing Capacity of the Pile** mode.

Results of Analysis

The analysis is performed by clicking the **Calculate** button. The result of this calculation (settlement of the pile) is displayed on the **Results** tab.

16.5 Pile Field Tests

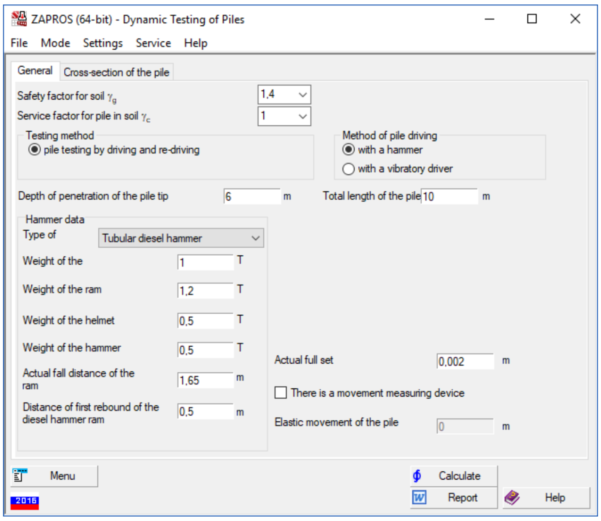
16.5.1 Dynamic Testing of Piles

This mode performs an analysis for determination of the load-bearing capacity of piles based on the results of their dynamic testing according to the requirements of Sec. 5.3, 5.4, 5.7 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.3, 7.3.4, 7.3.7 of SP 50-102-2003 or SP 24.13330.2011). If the selected design code is DBN B.2.1-10-2009, the analysis is performed according to the requirements of DSTU B B.2.1-27:2010.

Data Preparation

The application considers two methods of pile driving: either with a hammer (Fig. 16.5.1-1, *a*) or with a vibratory driver (Fig. 16.5.1-1, *b*) and two methods of pile testing — either by driving and re-driving or by checking the results of production driving.

a



b

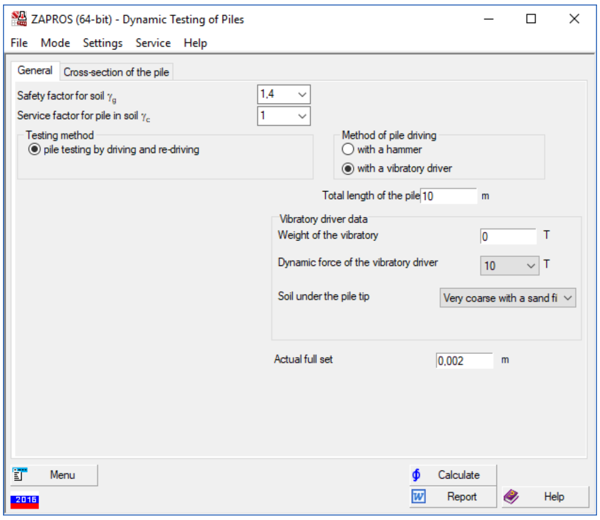


Figure 16.5.1-1. *The* **General** *tab of the* **Dynamic Testing of Piles** *dialog box*

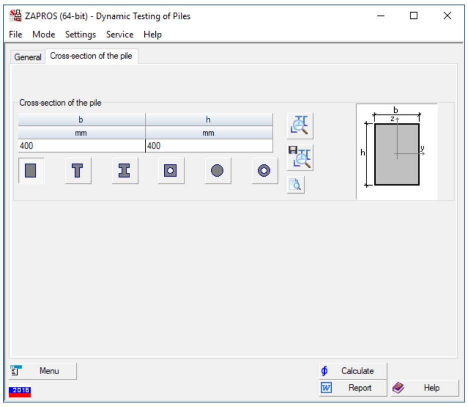


Figure 16.5.1-2. *The* **Cross-section of the Pile** *tab of the* **Dynamic Testing of Piles** *dialog box.*

Depending on the selected driving and testing method the application requests data necessary for the analysis. Moreover, the load-bearing capacity of piles can be also determined if there is data obtained by the movement measuring device.

The pile cross-section (rectangle, Tee, I-beam, square with a round hollow core, circle, or ring) and its sizes are assigned in the respective tab (Fig. 16.5.1-2).

Results of Analysis

The analysis is performed by clicking the **Calculate** button. The result of this calculation (load-bearing capacity of the pile) is displayed on the **Results** tab.

Limitations

Notes to Sec. 5.7 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.7 of SP 50-102-2003, SP 24.13330.2011 or notes to Sec. 5.2.2 of DSTU B.B.2.1-27:2010) are not taken into account in the analysis of the load-bearing capacity of the pile.

16.5.2 Testing with a Sample Pile

This mode enables you to determine the load-bearing capacity of a driven (production) friction pile subjected to compressive load based on the results of soil testing with a sample pile according to the requirements of Sec. 5.8, 5.9 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.8, 7.3.9 of SP 50-102-2003 or SP 24.13330.2011). If the selected design code is DBN B.2.1-10-2009, the analysis is performed according to the requirements of DSTU B B.2.1-27:2010.

A production pile is a pile commonly used in construction, of an appropriate material, size and design.

A sample pile is a standard compound steel pipe with a conical shoe at the bottom; in compliance with GOST 24942-81 and GOST 5686-94, it must have the outer diameter of 114 mm. Depending on the design of the connection between the conical shoe and the pile shaft (the pipe itself), the sample piles are classified into:

Type I – the pile shoe is rigidly attached to the pile shaft;

Type II – the pile shoe moves freely with respect to the pile shaft;

Type III – the pile shoe is connected to the pile shaft by a force transducer.

Depending on the number of tests with the sample pile and on the variability of the particular values of the pile ultimate resistance in the test locations, the safety factor for soil is varied. It is calculated by the application for the confidence level of 0.95 according to the requirements of GOST 20522-75.

Data Preparation

Data for the analysis should be prepared beginning with specifying a type of a sample pile and a number of soil tests. The **Particular value of ultimate resistance** table (Fig. 16.5.2-1) will display the respective number of rows.

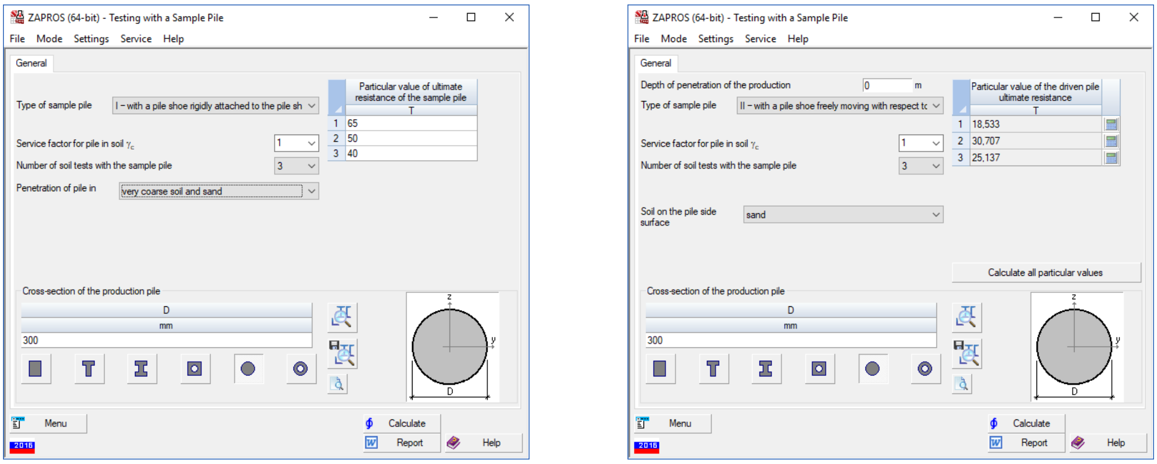


Figure 16.5.2-1. *The* **General** *tab of the* **Testing with a Sample Pile** *dialog box*

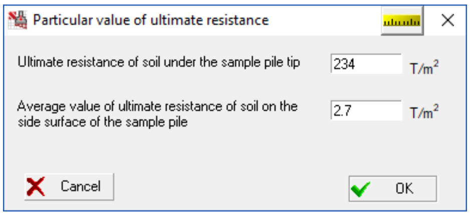


Figure 16.5.2-2. *The* **Particular Value of Ultimate Resistance** *dialog box*

When testing soil with the Type I sample pile, you need to directly specify a particular value of the sample pile ultimate resistance, $F_{u,sp}$, obtained from the results of static load testing according to Sec. 5.5 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.5 SP 50-102-2003 or SP 24.13330.2011), in order to determine the particular value of the ultimate resistance of the driven pile in the sample pile testing location, F_u.

The particular value of the ultimate resistance of the driven pile at the sample pile (Type II or III) testing location, F_u, is calculated depending on the ultimate soil resistance under the sample pile tip obtained by testing, average value of the ultimate resistance of soil on the side surface of the sample pile, depth of penetration of the production pile and on the dimensions of its cross-section. All these data should be specified for each test in the dialog box (Fig. 16.5.2-2) that can be invoked by clicking the button in the respective row of the table.

The particular value of the ultimate resistance of Type II and III sample piles depends for each test on the depth of penetration of the pile tip and on the cross-section of the production pile. Therefore, as soon as these parameters are modified, particular values calculated earlier and entered into the table become incorrect and the table gets cleared. To recalculate all particular values click the **Calculate all particular values** button.

Results of Analysis

The analysis is performed by clicking the **Calculate** button. The result of this calculation (load-bearing capacity of the pile) is displayed on the **Results** tab.

Limitations

Notes to Sec. 5.9b of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.9 b of SP 50-102-2003, SP 24.13330.2011 or notes to Sec. 6.2 of DSTU B.B.2.1-27:2010) are not taken into account in the analysis of the load-bearing capacity of the pile.

16.5.3 Testing with a Probe Pile

This mode enables you to determine the load-bearing capacity of a driven (production) friction pile subjected to compressive load based on the results of soil testing with a probe pile according to the requirements of Sec. 5.8, 5.10 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.8, 7.3.10 of SP 50-102-2003 or SP 24.13330.2011). If the selected design code is DBN B.2.1-10-2009, the analysis is performed according to the requirements of DSTU B B.2.1-27:2010.

A production pile is a pile commonly used in construction, of an appropriate material, size and design.

A probe pile is a standard compound steel pipe with a conical shoe and a friction sleeve; in compliance with GOST 5686-94, it must have the outer diameter of 127 mm.

Depending on the number of tests with a probe pile and on the variability of the particular values of the pile ultimate resistance in the probing locations, the safety factor for soil is varied. It is calculated by the application for the confidence level of 0.95 according to the requirements of GOST 20522-75.

Data Preparation

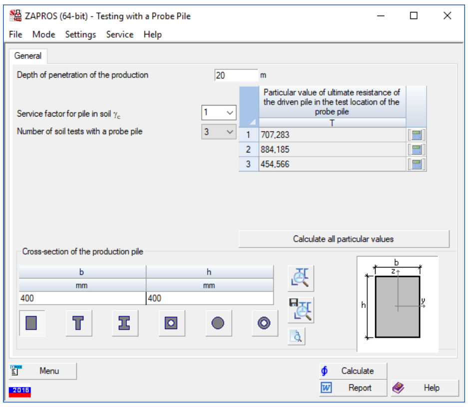


Figure 16.5.3-1. *The* **General** *tab of the* **Testing with a Probe Pile** *dialog box*

Data for the analysis should be prepared beginning with specifying the number of soil tests. The **Particular value of ultimate resistance** table (Fig. 16.5.3-1) will display the respective number of rows.

The particular value of the ultimate resistance of the driven pile at the probing location, F_u, is calculated depending on the ultimate soil resistance under the probe pile tip obtained by testing, average value of the ultimate resistance of i-th layer of soil on the side surface of the probe pile, dimensions of the production pile cross-section, and properties of soil on the side surface. All these data should be specified for each test in the dialog box (Fig. 16.5.3-2), that can be invoked by clicking the button in the respective row of the table.

As a rule, data obtained by a series of tests are rather close. To accelerate the input of data, the application allows to copy data from one test to another and then make necessary changes. To do it, click the **Copy Data** button and specify No. of the test to take the data from in the dialog box (Fig. 16.5.3-3) that opens.

Figure 16.5.3-2. *The* **Particular Value of Ultimate Resistance** *dialog box*

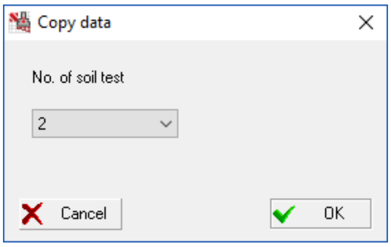


Figure 16.5.3-3. *The* **Copy Data** *dialog box*

The particular value of the ultimate resistance depends for each test on the depth of penetration of the pile tip and on the cross-section of the production pile. Therefore, as soon as these parameters are modified, particular values calculated earlier and entered into the table become incorrect and the table gets cleared. To recalculate all particular values, you can click the **Calculate all particular values** button.

Results of Analysis

The analysis is performed by clicking the **Calculate** button. The result of this calculation (load-bearing capacity of the pile) is displayed on the **Results** tab.

16.5.4 Cone Penetration Test

This mode enables you to determine the load-bearing capacity of a driven (production) friction pile subjected to compressive load based on the results of the cone penetration test according to the requirements of Sec. 5.8, 5.11 of SNiP 2.02.03-85, KMK 2.02.03-98 (Sec. 7.3.8, 7.3.11 of SP 50-102-2003 or Sec. 7.3.8, 7.3.10 of SP 24.13330.2011). If the selected design code is DBN B.2.1-10-2009, the analysis is performed according to the requirements of DSTU B B.2.1-27:2010.

The production pile is a pile commonly used in construction, of an appropriate material, size and design.

Depending on the design of the shoe, the penetrometers are classified into three types:

Type I – mantle cone penetrometer;

Type II – friction sleeve cone penetrometer;

Type III – friction sleeve cone penetrometer with a friction reducer.

According to GOST 20069-81 the Type I penetrometer rod should have the outer diameter of 35.7 mm, while the diameters of Types II and III penetrometer rods are based on the design considerations, but they should not exceed 55 mm.

Depending on the number of soundings and on the variability of the particular values of the pile ultimate resistance in the sounding points, the safety factor for soil is varied. It is calculated by the application for the confidence level of 0.95 according to the requirements of GOST 20522-75.

Data Preparation

Data for the analysis should be prepared beginning with specifying a penetrometer type and the number of soil tests. The **Particular value of ultimate resistance** table (Fig. 16.5.4-1) will display the respective number of rows.

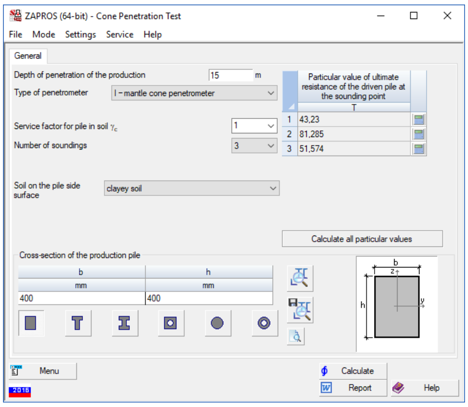


Figure 16.5.4-1. *The* **General** *tab of the* **Cone Penetration Test** *dialog box*

The particular value of the ultimate resistance of the driven pile at the sounding point, F_u, is calculated depending on the average value of the soil resistance under the cone penetrometer (within one diameter or the length of a bigger side above and four diameters or four lengths of the bigger side below the level of the current pile tip) obtained by testing, average value of the ultimate resistance of ***i***-th layer of soil on the penetrometer side surface, depth of penetration of the production pile, dimensions of the production pile cross-section, and properties of soil on the side surface. All these data should be specified for each test in the dialog box (Fig. 16.5.4-2), that can be invoked by clicking the button  in the respective row of the table.

As a rule, data obtained by a series of tests are rather close. To accelerate the input of data, the application allows to copy data from one test to another and then make necessary changes. To do it, click the **Copy Data** button and specify No. of the test to take the data from in the dialog box (Fig. 16.5.3-3) that opens.

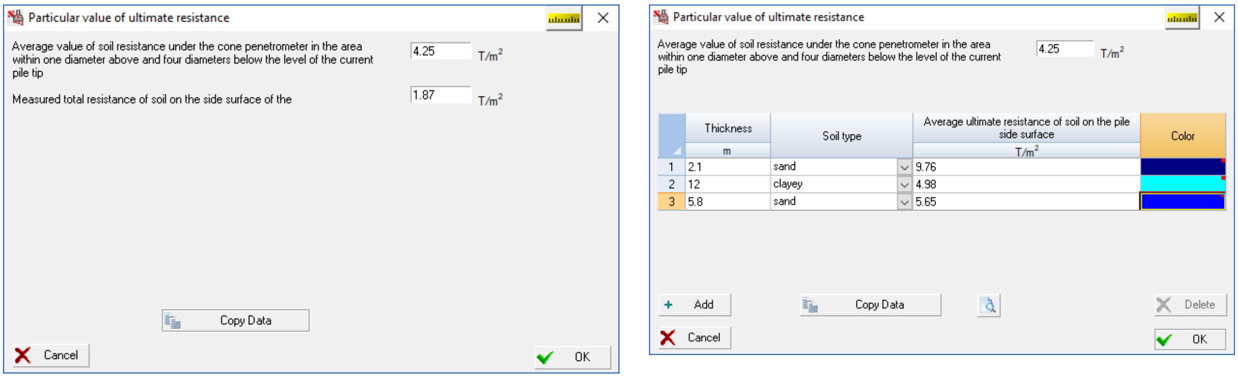


Figure 16.5.4-2. *The* **Particular Value of Ultimate Resistance** *dialog box*

The particular value of the ultimate resistance depends for each test on the depth of penetration of the pile tip and on the cross-section of the production pile. Therefore, as soon as these parameters are modified, particular values calculated earlier and entered into the table become incorrect and the table gets cleared. To recalculate all particular values, you can click the **Calculate all particular values** button.

Results of Analysis

The analysis is performed by clicking the **Calculate** button. The result of this calculation (load-bearing capacity of the pile) is displayed on the **Results** tab.

16.6 Design Codes Implemented by ZAPROS

Mode	Referenced to sections of codes and standards							
	SNiP 2.02.01-83*, KMK 2.02.01-98	**SP 50-101-2004**	**SP 22.13330.2011**	**SP 22.13330.2016**	**DBN B.2.1-10-2009**	**SNiP 2.02.03-85, KMK 2.02.03-98**	**SP 50-102-2003**	**SP 24.13330.2011**
Ultimate Soil Deformations	Annex 4	Annex F, table F.1	Annex E, table E.1	Annex D, table D.1	Annex I, table I.1	–	–	–
Design Soil Resistance	Annex 3, table 1–6	Annex E, table E.1–E.10	Annex C, table C.1–C.10	Annex B, table B.1–B.10	Annex F, table F.1–F.6	–	–	–
Soil Properties	Annex 1, table 1–3	Annex D, table D.1–D.7	Annex B, table B.1–B.8	Annex A, table A.1–A.8	Annex C, table C.1–C.3	–	–	–
Service Factors	table 3	table 5.2	table 5.4	table 5.4	Annex F.7, table F.7	–	–	–
Foundation Tilt	Sec. 9-11 of Annex 2	Sec. 5.5.43-5.5.45	Sec. 5.6.43-5.6.45	Sec. 5.6.43-5.6.45	Sec. E.13 of Annex E	–	–	–
Settlement of Foundation	Sec. 2.39-2.40, Sec. 1-8 of Annex 2	Sec. 5.5.31-5.5.41	Sec. 5.6.31-5.6.41	Sec. 5.6.31-5.6.41	Sec. E.1 – E.10 of Annex E	–	–	–
Subsoil Parameters	–	–	–		–	–	–	–
Ultimate Pressure in the Deformation Analysis	Sec. 2.41-2.47	Sec. 5.5.8-5.5.14	Sec. 5.6.7-5.6.13	Sec. 5.6.7-5.6.13	Sec. F.4 of Annex F	–		
Service Factors for Piles	–	–	–		Annex N, table N.2.3, N.3.1, N.4.1	table 3, 5, 19	table 7.3, 7.5, 13.1	table 7.4, 7.6, 14.1
Nomenclature for Piles	–	–	–		GOST 19804.4-78*, 19804.2-79*, 19804.5-83, 19804.6-83, 19804.7-83	GOST 19804.4-78*, 19804.2-79*, 19804.5-83, 19804.6-83, 19804.7-83	GOST 19804.4-78*, 19804.2-79*, 19804.5-83, 19804.6-83, 19804.7-83	GOST 19804.4-78*, 19804.2-79*, 19804.5-83, 19804.6-83, 19804.7-83

Mode	Referenced to sections of codes and standards							
	SNiP 2.02.01-83*, KMK 2.02.01-98	**SP 50-101-2004**	**SP 22.13330.2011**	**SP 22.13330.2016**	**DBN B.2.1-10-2009**	**SNiP 2.02.03-85, KMK 2.02.03-98**	**SP 50-102-2003**	**SP 24.13330.2011**
Load-bearing Capacity of the Pile	–	–	–		Sec. N.1, N.2, N.3, N.7	4.1, 4.2, 4.5, 4.6, 4.7, 4.8, 4.9	7.2.1, 7.2.2, 7.2.5, 7.2.6, 7.2.7, 7.2.8, 7.2.9	7.2.1, 7.2.2, 7.2.5, 7.2.6, 7.2.7, 7.2.8, 7.2.9
Analysis of Pile	–	–	–		Sec. 8.5.4.9, Sec. N.8.1 of Annex N	Sec. 1-15 Annex 1	Sec. E.1-E.8 Annex E	Sec. C.1-C.7 Annex C
Settlement of the Pile	–	–	–		Sec. P.1 of Annex P	Annex 4	Annex I	Annex E
Dynamic Testing of Piles	–	–	–		Sec. 5.2 DSTU B B.2.1-27:2010	Sec. 5.1-5.8	Sec. 7.3.1-7.3.8	Sec. 7.3.1-7.3.8
Testing with a Sample Pile	–	–	–		Sec. 6.1, 6.2 DSTU B B.2.1-27:2010	Sec. 5.1-5.6, 5.8, 5.9	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.9	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.9
Testing with a Probe Pile	–	–	–		Sec. 6.1, 6.3 DSTU B B.2.1-27:2010	Sec. 5.1-5.6, 5.8, 5.10	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.10	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.10
Cone Penetration Test	–	–	–		Sec. 6.1, 6.4 DSTU B B.2.1-27:2010	Sec. 5.1-5.6, 5.8, 5.11	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.11	Sec. 7.3.1-7.3.6, 7.3.8, 7.3.11

16.7 References

[1] V.Z.Vlasov, N.N.Leontiev, Beams, Plates, and Shells on Elastic Foundation, Moscow: Fizmatgis Publishers, 1960 — 491 p.

[2] M.I.Gorbunov-Posadov, Beams and plates on an elastic base. Mashstroyizdat Publishers, 1949.

[3] Instruction Manual for Design and Engineering of Pile Foundations for Structures in the City of Moscow. – Government of Moscow, 2001, 146 p.

[4] I.A.Mednikov, Coefficients of subgrade reaction of a linear multilayer base, Foundations, beds, and soil mechanics, 1967, – No.4.

[5] P.L.Pasternak, On a New Method of Analysis of an Elastic Foundation by Means of Two Constants. Moscow: Gosstroyizdat Publishers, 1954 — 56 p.

[6] V.G.Piskunov, Y.M.Fedorenko. A dynamic method for monitoring layered slabs on elastic subsoil. Architecture and Construction in Belarus, N5-6, 1994, p.10-22.
[7] Guide to the design of foundations of structures (to SNiP 2.02.01-83), Gersevanov Research Institute for Foundations and Underground Structures, USSR State Committee of Construction. Moscow, "Stroyizdat", 1986.

17. Appendix

17.1 Service Functions

As you work with the program, you may need to perform some relevant auxiliary calculations. The **Service** menu of all programs of the Office package contains items for invoking additional calculators: a standard one of MS Windows (if it has been installed together with the system), a formula calculator, a converter of units of measurement, as well as a special calculator for selecting discrete reinforcement based on the specified area (in **ARBAT**).

17.1.1 Formula Calculator

В разделе меню **Сервис** предусматривается возможность вызова как стандартного калькулятора среды MS Windows (если он установлен в системе), так и специального вычислителя (рис. 17.1.1-1), позволяющего выполнять расчеты по формулам.

The formula calculator is used to perform calculations by formulae that can be specified in a text field.

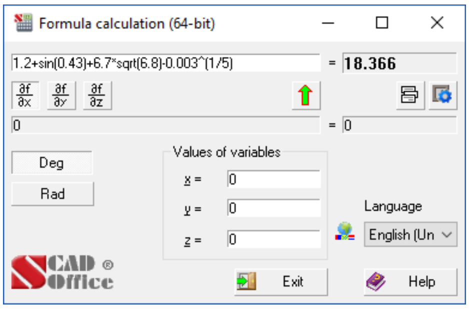


Figure 17.1.1-1. The dialog box of the calculator

The following rules should be observed when entering a formula:

- names of functions must be entered in lowercase Roman letters;
- the fractional and the integer parts of a number must be separated by a period or another separator assigned by the user during the customization of the operating system (see **Settings | Regional Settings | Number**);
- arithmetic operations must be specified with the symbols +, -, *, /, ^ (raising to a power); for example, 2.5*2.5*2.5 can also be written as 2.5^3.

The following mathematical functions can be used in the formulae:

floor — the greatest integer not greater than the argument;
tan — tangent;
sin — sine;
cos — cosine;
asin — arcsine;
acos — arccosine;
atan — arctangent;
exp — exponent;
ceil — the least integer greater than the argument;
tanh — hyperbolic tangent;
sinh — hyperbolic sine;
cosh — hyperbolic cosine;
ln — natural logarithm;
log — decimal logarithm;
abs — absolute value;
sqrt — square root.

Depending on the state of the **Degrees/Radians** switch buttons, arguments of the trigonometric functions (**sin**, **cos**, **tan**) and results of inverse trigonometric functions (**asin**, **acos**, **atan**) can be presented in degrees or radians, respectively.

Only parentheses are allowed for grouping arguments together; these can be nested as deeply as desired.

Example.

The following formula

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

must be written as follows:

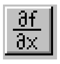
1.2+sin(0.43)+6.7*sqrt(6.8)–0.003^(1/5).

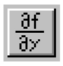
There is an additional option of using three independent variables **x**, **y**, **z** in formulas. Values for the variables will be specified in respective text fields. This makes it possible to perform a series of similar calculations with different parameters. For example, the following formula

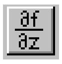
$$1,2+\sin(x)+6,7\sqrt{6,8}-\sqrt[5]{y}$$

must be written in this mode as follows:

1.2+sin(x)+6.7*sqrt(6.8)–y^(1/5).

Moreover, if you have entered a formula depending on the variables **x**, **y**, **z**, the field at the bottom will display a symbolic expression of its partial derivative with respect to one of the variables (**x**, **y**, or **z**) for which the respective marker 

, 

, or 

is enabled at the moment.

 button allows to save the current state of the text field with a formula and the respective values of variables, and to open data saved earlier. After pressing the button the drop-down menu appears. Selecting its first position (**Save**) will save the entered data. The other lines display the formulas entered earlier, which can be selected by the mouse-pointer.

The button  enables to invoke the **Settings** dialog box where you can specify the number of significant digits after the decimal point for the output of the results.

17.1.2 Converting Units of Measurement

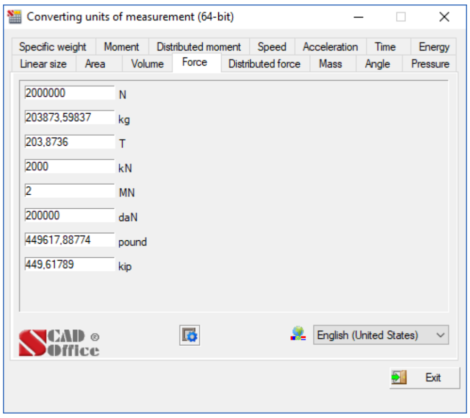


Figure 17.1.2-1. *The* **Converting Units of Measurement** *dialog box*

This piece of software converts data between different units of measurement (Fig. 17.1.2-1). In order to do this, select a tab of respective measures (**Length**, **Area** etc.).

The order of converting operations depends on whether the units of measurement are simple (i.e. length, area or time) or compound (i.e. pressure, speed or mass).

If you have to convert simple units of measurement, you just have to enter a number into one of the text fields. As a result you will receive values in all other units of measurement. If the units are compound, you have to select the original units of measurement in the drop-down lists of one line and the units you wish to convert to in the other one. Enter the number in the text field of the first line and you will receive the result in the second one.

You can change the language (Russian/English) with the help of the respective button with a flag.

17.1.3 Rebar Calculator

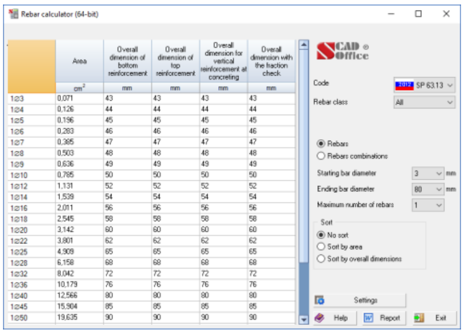


Figure 17.1.3-1. *The* **Rebar Calculator** *dialog box*

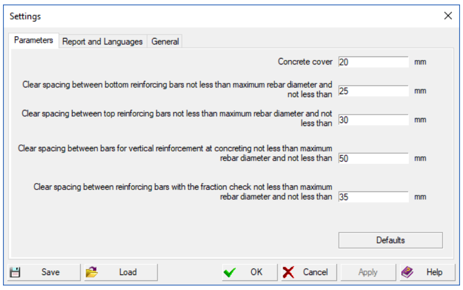


Figure 17.1.3-2. *The* **Parameters** *dialog box*

The **Rebar Calculator** (Fig. 17.1.3-1) enables you to obtain all possible combinations of rebars under the given limitations by the bar diameter and the number of rebars.

The following limitations can be specified:

- if the **Rebars** radio button is selected, there will be no combinations of rebars of different diameters included in the results. All the reinforcement options containing no more than a specified maximum number of rebars from the range assigned by the **Starting bar diameter** and **Ending bar diameter** lists will be generated;
- if the **Rebars combinations** radio button is selected, all the combinations of two rebars of different diameters will be found and presented. The bars will be selected from the range assigned by the **Starting bar diameter** and **Ending bar diameter** lists.

The lists of bar diameters are filled in compliance with the selected design code and the rebar class.

The table of results will contain a list of combinations with the following data provided for each combination: cross-sectional area of reinforcement, minimum overall dimensions of the top and bottom reinforcement, overall dimensions for vertical reinforcement and the overall dimensions with the fraction check.

All the dimensions are given taking into account the concrete cover, which is specified in the **Settings** dialog box, on the **Parameters** tab (Fig. 17.1.3-2). You can also change the minimum distance between rebars recommended by SNiP or return to standard values by clicking the **Defaults** button. The minimum dimensions are calculated in compliance with the limitations given in SNiP for the placement of reinforcement in cross-sections.

The list can be sorted by the area of reinforcement or by the overall dimensions by choosing the appropriate radio button.

17.2 List of Assortments of Rolled Profiles

17.2.1 Assortment of the Chelyabinsk Metallurgical Plant

Regular I-beam STO ASChM 20-93
Wide flange I-beam STO ASChM 20-93
Column I-beam STO ASChM 20-93
I-beam R 40-93
Special I-beam STO ASChM 20-93

17.2.2 GOST

Equal angle GOST 8509-93
Unequal angle GOST 8510-86*
Channel with parallel flanges GOST 8240-89
Channel GOST 5267.1-90
Channel with sloped inner flange surfaces GOST 8240-89
Column I-beam GOST 26020-83
I-beam with sloped inner flange surfaces GOST 8239-89
Additional series I-beam GOST 26020-83
Regular I-beam GOST 26020-83
Wide flange I-beam GOST 26020-83
Column T-bar TU 14-2-685-86
T-bar TU 14-2-685-86
Longitudinally electric welded Circular Hollow Sections GOST 10704-91
Circular Hollow Sections GOST 10704-91 (reduced list)
Bent channels with equal flanges GOST 8278-83 (steels C239-C245)
Bent channels with equal flanges GOST 8278-83 (steels C255-C275)
Square Hollow Sections TU 36-2287-80
Rectangular Hollow Sections TU 67-2287-80
Channel with parallel flanges GOST 8240-97
Channel with sloped inner flange surfaces GOST 8240-97
Economical channels with parallel flanges GOST 8240-97
Special channels GOST 8240-97
Light channels with parallel flanges GOST 8240-97
Special steel channel GOST 19425-74*
Special I-beam GOST 19425-74*
Seamless hot finished steel Circular Hollow Sections GOST 8732-78
Circular Hollow Sections GOST P 54157-2010
Square Hollow Sections GOST 12336-66
Square Hollow Sections GOST P 54157-2010
Rectangular Hollow Sections GOST 12336-66
Rectangular Hollow Sections GOST P 54157-2010
Square Steel Hollow Sections GOST 8639-68
Rectangular Steel Hollow Sections GOST 8645-68
Square Hollow Sections GOST 25577-83*
Rectangular Hollow Sections GOST 25577-83*

Square Hollow Sections GOST 30245-94
Rectangular Hollow Sections GOST 30245-94
Steel bent closed welded square section GOST 30245-2003
Steel bent closed welded rectangular section GOST 30245-2003
Square Steel Hollow Sections GOST 8639-82
Calibrated round steel GOST 7417-75
Round Hot-rolled Steel Bars GOST 2590-88
Square Hot-rolled Steel Bars GOST 2591-88
Steel bent closed welded square section GOST 30245-2012
Steel bent closed welded rectangular section GOST 30245-2012
Bent equal angle GOST 19771-93 (tab.1)
Bent equal angle GOST 19771-93 (tab.2)
Bent unequal angle GOST 19772-93 (tab.1)
Bent unequal angle GOST 19772-93 (tab.2)
Round Hot-rolled Steel Bars GOST 2590-2006
Square Hot-rolled Steel Bars GOST 2591-2006
Circular Hollow Sections TU 1381-103-05757848-2013
Wide flange I-beam TU 0925-016-00186269-2016
Wide flange I-beam TU 0925-036-00186269-2016
Column I-beam TU 0925-016-00186269-2016
Regular I-beam TU 0925-016-00186269-2016
Regular I-beam TU 0925-016-00186269-2016
Column I-beam TU 0925-036-00186269-2016
Regular I-beam TU 0925-036-00186269-2016
Pile I-beam TU 0925-036-00186269-2016
Narrow I-beam TU 0925-016-00186269-2016
Narrow flange I-beam TU 24107-016-00186269-2017
Regular I-beam TU 24107-016-00186269-2017
Medium flange I-beam TU 24107-016-00186269-2017
Wide flange I-beam TU 24107-016-00186269-2017
Column I-beam TU 24107-016-00186269-2017
Regular I-beam TU 24107-036-00186269-2017
Column I-beam TU 24107-036-00186269-2017
Wide flange I-beam TU 24107-036-00186269-2017
Pile I-beam TU 24107-036-00186269-2017
Additional series I-beam GOST R 57837-2017
Additional series column I-beam GOST R 57837-2017
Wide flange I-beam GOST R 57837-2017
Regular I-beam GOST R 57837-2017
Column I-beam GOST R 57837-2017
Pile I-beam GOST R 57837-2017
Circular Hollow Sections GOST 32931-2015
Square Hollow Sections GOST 32931-2015
Rectangular Hollow Sections GOST 32931-2015
Circular Hollow Welded Sections GOST R 58064-2018

Table B.1 of GOST P 54157-2010 (rectangular hollow sections) contains incorrect information on the hollow sections of size 400x300x25 and beyond (the moment of inertia I_x and section modulus W_x columns contain the same numbers).

Therefore, these hollow sections are not included in the **SCAD Office** database.

17.2.3 Reduced GOST

Equal angle GOST 8509-93
Unequal angle GOST 8510-86*
Channel with parallel flanges GOST 8240-89
Channels with sloped inner flange surfaces GOST 8240-89
Column I-beam GOST 26020-83
I-beam with sloped inner flange surfaces GOST 8239-89
Additional series I-beam GOST 26020-83
Regular I-beam GOST 26020-83
Wide flange I-beam GOST 26020-83
Column T-bar TU 14-2-685-86
T-bar TU 14-2-685-86
Longitudinally electric welded Circular Hollow Sections GOST 10704-91
Bent channel with equal flanges GOST 8278-83 (steels C239-C245)
Bent channels with equal flanges GOST 8278-83 (steels C255-C275)
Square Hollow Sections TU 36-2287-80
Rectangular Hollow Sections TU 67-2287-80

17.2.4 Old Russian Assortment

Equal angle OST 14-1926
Equal angle OST 14-1932
Unequal angle OST 15-1926
Unequal angle OST 15-1932
I-beam OST 16-1926
I-beam OST 16-1932
Channel with sloped inner flange surfaces OST 17-1926
Channel with sloped inner flange surfaces OST 17-1933
I-beams OST 10016-39
Channel OST 10017-39
Equal angle OST 10014-39
Unequal angle OST 10015-39
IPB Shapes DIN 1025
IPB1 Shapes DIN 1025
IPBv Shapes DIN 1025
IPE Shapes DIN 1025
IPEo Shapes DIN 1025
IPEv Shapes DIN 1025

17.2.5 ASTM

Equal Angles
Unequal Angles
H-Piles
Miscellaneous Shapes
American Standard Shapes
Wide Flange Shapes
Miscellaneous Tees
American Standard Tees
Wide Flange Tees
American Standard Channels
Miscellaneous Channels
Pipe
Extra Strong Pipe
Double-Extra Strong Pipe
Tube Steel (Square)
Tube Steel (Rectangular)

17.2.6 British Standard Sections

Universal Beams
Universal Columns
Universal Bearing Piles
Rectangular Hollow Sections
Circular Hollow Sections
Square Hollow Sections
Joists
Equal Angles
Unequal Angles
Channels
Structural Tees cut from UB's
Structural Tees cut from UC's
Circular Hollow Sections EN10219-2:2006(E)
Square Hollow Sections EN10219-2:2006 (E)
Rectangular Hollow Sections EN10219-2:2006 (E)

17.2.7 Overseas Shapes

ASTM W Shapes (Universal beams and columns)
IPE Shapes (European universal beams)
HE Shapes (European universal beams and columns)
Rectangular Hollow Sections
Circular Hollow Sections
Square Hollow Sections

17.2.8 Arbed

Equal Angles Euronorm 56-77
Unequal Angles Euronorm 57-78
European I-beams (IPE)
European standard beams (IPN)
European wide flange beams (HE)
European wide flange beams (HL)
Wide flange columns (HD)
Wide flange bearing piles (HP)
American wide flange beams (W)
British universal beams (UB)
British universal columns (UC)
Channels with parallel flanges
European standard channels

17.2.9 Welded profiles

Welded I-beam TU U 01412851.001-95
Welded I-beam TU U 01412851.001-95
Welded I-beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
Welded I-beam instead of STO ASChM 20-93 (TU 0925-001-97638531-2016)
Column I-beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
Welded I-beam instead of STO ASChM 20-93 (TU 0925-001-97638531-2016)
Wide flange I-beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
Wide flange I-beam instead of STO ASChM 20-93 (TU 0925-001-97638531-2016)

17.2.10 OTUA

Equal Angles NF A 45-009
Unequal Angles NF A 45-010
IPN Shapes NF A 45-209
PA Shapes NF A 45-205
IPE-A Shapes NF A 45-205
IPE Shapes NF A 45-205
IPE-R Shapes NF A 45-205
HEA-A Shapes NF A 45-201
HEA Shapes NF A 45-201
HEB Shapes NF A 45-201
HEM, HEC Shapes NF A 45-201
Structural Tees cut from PA NF A 45-205
Structural Tees cut from IPE-A NF A 45-205
Structural Tees cut from IPE NF A 45-205
Structural Tees cut from IPE-R NF A 45-205
Structural Tees cut from HEA-A NF A 45-211
Structural Tees cut from HEA NF A 45-201
Structural Tees cut from HEB NF A 45-201
Structural Tees cut from HEM, HEC NF A 45-201
Channels UPN NF A 45-202
Channels UPN-A NF A 45-202
Channels UAP NF A 45-255
Channels UAP-A NF A 45-255
Circular Hollow Sections
Rectangular Hollow Sections
Square Hollow Sections

17.2.11 DIN

Equal Angles DIN 1028
Unequal Angles DIN 1029
Beam DIN 1025
IP DIN 1025
IP DIN 1025 (9%)
Channels DIN 1026
Circular Hollow Sections DIN 2448
Rectangular Hollow Sections DIN 59410
Square Hollow Sections DIN 59410

17.2.12 Indian Assortment

Rolled steel equal angle
Unequal angle
Rolled steel beams ISHB
Rolled steel beams ISJB
Rolled steel beams ISWB
Rolled steel beams ISLB
Rolled steel beams ISMB
Rolled steel channels ISJC
Rolled steel channels ISLC
Rolled steel channels ISMC

17.2.13 Japanese Assortment

Equal Angles
Unequal Angles
Short Flange Shapes
Middle Flange Shapes
Wide Flange Shapes
Channel
Short Flange Tee
Middle Flange Tee
Wide Flange Tee
Circular Hollow Sections
Rectangular Hollow Sections
Square Hollow Sections

17.2.14 Assortment of Poland

Equal angle PN-EN 10056-1:2000
Equal angle PN-84/H-93401
I-beam PN-91/H-93419
Structural Tees cut from HEA PN-H-93452:2005
Wide flange I-beam PN-H-93452:2005
Structural Tees cut from HEC PN-H-93452:2005
Channels PN-86/H-93403
Square Hollow Sections PN-EN 10219-2:2000

17.2.15 Assortment of China

Equal angle GB/T 706-2008
Unequal angle GB/T 706-2008
Unequal angle type II GB/T 706-2008
I-beam with sloped inner flange surfaces GB/T 706-2008
Channel with sloped inner flange surfaces GB/T 706-2008
Circular Hollow Sections GB/T 6728-2002
Square Hollow Sections GB/T 6728-2002
Rectangular Hollow Sections GB/T 6728-2002

17.2.16 Cold-formed Profiles

Profile LP TU 1121-009-46216359-2010
Profile LC TU 1121-009-46216359-2010
Profile U TU 1122-002-10836231-2014
Profile C TU 1122-02-10836231-2014
Profile U TU 1120-001-37820873-2012
Profile C TU 1120-001-37820873-2012
Profile PC TU 1120-001-82861223-2009
Steel roll-formed C-shaped equal flange sections GOST 8282-83
Steel bent Z-shaped sections GOST 13229-78 tab.3
Steel bent Z-shaped sections GOST 13229-78 tab.4
Steel bent Z-shaped sections GOST 13229-78 tab.1
Steel bent Z-shaped sections GOST 13229-78 tab.2
Steel roll-formed unequal channels GOST 8281-80 tab.1
Steel roll-formed unequal channels GOST 8281-80 tab.2
Bent steel hat equal flange sections GOST 8283-93 tab.1
Bent steel hat equal flange sections GOST 8283-93 tab.2

17.2.17 Cold-formed zinc-coated steel profiles

Profile PC TU 1121-001-13830080
Profile PN TU 1121-001-13830080
Profile TC TU 1121-001-13830080
Profile TN TU 1121-001-13830080
Profile AC TU 1122-002-82866678-2013
Profile E TU 1121-001-99651760-2015 (steel 250/C255)
Profile E TU 1121-001-99651760-2015 (steel 350/C345)
Profile AE TU 1122-002-82866678-2013
Profile AZ TU 1122-002-82866678-2013 tab.2
Profile AZ TU 1122-002-82866678-2013 tab.3

17.2.18 Ukrainian cold-formed profiles

Profile C manufacture by Light House-Ukraine
Profile U manufactured by Light House-Ukraine
Profile C manufactured by Stalex
Profile Cw manufactured by Stalex
Profile U manufactured by Stalex
Steel bent Z-manufacture by Stalex
Steel bent Zw-manufacture by Stalex
Profile C manufacture by Pruszynski

Normative references

DBN V.2.6-14-97. Designs of buildings and structures. Roofs of buildings and structures: State building codes of Ukraine / Ukraine State Building.— Ukrarhbudinform, 1997.

DBN V.1.2-2:2006. Loads and actions, Ministry of Construction of Ukraine, Kiev, 2006 – 60 p.

DBN 362-92. Technical state assessment of steel structural members of industrial buildings and structures in service / Gosstroy of Ukraine.— K.: Ukrarhstroyinform, 1993.— 46 p.

DBN B.2.1-10-2009 "Bases and foundations of buildings", / Minregionbud of Ukraine. — Kyiv: 2009. — 104 p.

DBN B.2.6-163:2010 "THE STEEL CONSTRUCTIONS. Norms for design, fabrication and erection", / Minregionbud of Ukraine. — Kyiv: 2011. — 226 p.

DBN B.2.6-198:2014 "STEEL STRUCTURES. Design code ", / Minregion Ukraine. — Kyiv: 2014. — 199 p.

DSTU B V.1.2-3:2006. Deflections and displacements, Ministry of Construction of Ukraine, Kiev, 2006 – 10 p.

DSTU-N B B.2.6-211:2016. Design of Steel Structures. Structural Fire Design / Minregion Ukraine. – Kyiv: 2016. – 146 p.

DSTU B B.2.1-27:2010 "Ground and foundations structures. PILE. CALCULATION OF BEARING CAPACITY ON RESULTS OF FIELD TESTS".

EN 1993-1-1. Eurocode 3: Design of Steel Structures.— Part 1.1: General rules and rules for buildings.— Brussels (Belgium): CEN, 1992.— 344 p.

EN 1993-1-1/pr A2. Design of Steel Structures.— Part 1.1: General Rules and Rules for Buildings. Revised annex J: Joints in Building Frames.— Brussels (Belgium): CEN, 1994.— 70 p.

EN 1993-1-8:2005. Eurocode 3: Design of steel structures – Part 1-8: Design of joints. – Brussels (Belgium): CEN, 1992.

EN 10025-2:2004. Hot rolled products of structural steels – Part 2: Technical delivery conditions for non-alloy structural steels.

EN 10025-3:2004. Hot rolled products of structural steels – Part 3: Technical delivery conditions for normalized/normalized rolled weldable fine grain structural steels.

EN 10025-4:2004. Hot rolled products of structural steels – Part 4: Technical delivery conditions for thermomechanically rolled-weldable fine grain structural steels.

EN 10025-5:2004. Hot rolled products of structural steels – Part 5: Technical delivery conditions for structural steels with improved atmospheric corrosion resistance.

EN 10025-6:2004. Hot rolled products of structural steels – Part 6: Technical delivery conditions for flat products of high yield strength structural steels in the quenched and tempered condition.

EN 10210-1: 2006. Hot finished structural hollow sections of non-alloy and fine grain steels – Part 1: Technical delivery conditions.

EN 10219-1:2006. Cold formed structural hollow sections of non-alloy and fine grain steels – Part 1: Technical delivery conditions.

EN 1993-1-1:2005. Eurocode 3: Design of steel structures – Part 1-1: General structural rules.

EN 1090-1:2009. Execution of stell structures and aluminium structures – Part 1: Requirements for conformity assessment of structural components.

EN 1992-1-1:2004. Eurocode 2: Design of concrete structures – Part 1: General rules and rules for buildings.

EN ISO 898-1:1999-12. Mechanical properties of fasteners made of carbon steel and alloy steel – Part 1: Bolts, screws and studs.

EN 14399-3:2005-08. High-strength structural bolting assemblies for preloading – Part 3: System HR – Hexagon bolt and nut assemblies.

EN 14399-4:2005-08. High-strength structural bolting assemblies for preloading – Part 4: System HV – Hexagon bolt and nut assemblies.

EN 14399-5:2006. High-strength structural bolting assemblies for preloading – Part 5: Plain washers (includes AC: 2006).

EN 14399-6:2006. High-strength structural bolting assemblies for preloading – Part 5: Plain chamfered washers.

EN 15048-1:2007-07. Non-preloaded structural bolting assemblies – Part 1: General requirements.

EN 1993-1-5:2005. Eurocode 3: Design of steel structures – Part 1-5: Plated structural elements. – Brussels (Belgium): CEN, 2005.

Eurocode 3 – Design of steel structures, Part 1.1 General rules and rules for buildings, European Prestandard ENV 1993-1-1, CEN, Brussels, February 19.

GOST 26633-91. Heavy-weight and sand concretes. Specifications.

GOST 1759.4-87 Bolts, screws and studs. Mechanical properties and test methods. — Moscow, USSR State Standards Committee, 1987.

GOST 22356-77*. High strength bolts and nuts and washers. General specifications. – Moscow, USSR State Standard Committee.

GOST 20069-81. Soils. Field test method by static sounding.

GOST 5686-94. Soils. Field test methods by piles.

GOST 24942-81. Soils. Test methods by standard pile.

GOST 20522-75. Soils. Statistical rating characteristics.

GOST 8239-89. Hot-rolled steel flange beams. Rolling products. – Moscow, USSR State Standard Committee, 1989.

GOST 26020-83. Hot-rolled steel I-beam with parallel flange edges. Dimensions. – Moscow, "Stroyizdat", 1983.

GOST 7307-75*. Parts of wood and wood materials. Machining allowances.

GOST 16350-80. Climate of USSR. Zoning and statistical parameters of climatic factors for technical purposes.— Moscow, USSR State Standards Committee, 1981.

GOST 27751-88. Reliability for structures and bases. General principles (ST SEV 384-87). — Moscow, USSR State Standards Committee, 1989.

GOST 12.1.005-88 (1991). General sanitary requirements for working zone air.— Moscow, USSR State Standards Committee, 1988.

GOST 24454-80. Coniferous sawn timber. Sizes.

GOST 103-76*. Hot-rolled steel strip. – Moscow, USSR State Standard Committee.

GOST 27772-82. Rolled products for structural steel constructions. General specifications. – Moscow, USSR State Standard Committee, 1983.

GOST 27772-88*. Rolled products for structural steel constructions. General specifications. – Moscow, USSR State Standard Committee, 2001. – 76 c.

GOST 19903-74. Hot-rolled steel sheets. Dimensions. – Moscow, USSR State Standard Committee.

GOST 19904-90. Cold-rolled steel sheets. Dimensions. – Moscow, USSR State Standard Committee.

GOST 82-70*. Universal hot-rolled steel wide strips. – Moscow, USSR State Standard Committee.

GOST 23279-85. Welded reinforcing meshes for reinforced concrete structures and products. General specifications.

GOST 21.501-2011 (DSTU B A.2.4-7-95). System of design documents for construction. Rules for execution of the working documentation of architectural and construction solutions.

GOST 21.1101-2013 (DSTU B A.2.4-4-99). System of design documents for construction. Main requirements for design and working documents.

GOST 7473-94 (DSTU B B.2.7-96-2000). Ready-mixed concrete. Specifications.

GOST 5781-82. Hot-rolled steel for reinforcement of ferroconcrete structures. Specifications. II.— Moscow, USSR State Standard Committee, 1983.

GOST 14098-91. Welded joints of reinforcement and inserts for reinforced concrete structures. Types, constructions and dimensions.

GOST 8240-89. Hot-rolled steel channels. Assortment. — Moscow, USSR State Standards Committee, 1989.

GOST 8509-86. Hot-rolled steel equal-leg angles. Dimensions. – Moscow, USSR State Standard Committee, 1986.

GOST 8510-86*. Hot-rolled steel unequal-leg angles. Dimensions. — Moscow, USSR State Standards Committee, 1986.

GOST 8509-93. Hot-rolled steel equal-leg angles. Dimensions. — Moscow, USSR State Standards Committee, 1993.

SNiP II-25-80. Timber Structures/State Committee of Russia for Construction. Moscow, 2001, 30 p.

SNiP 52-01-2003. Concrete and reinforced concrete structures. Guidelines. Moscow, 2004, 24 p.

SNiP 2.03.01-84*. Concrete and reinforced concrete structures. / Ministry of Construction of Russia. Moscow, USSR State Comittee for Construction and Architecture Press, 1989, 80 p.

SNiP II-21-75. Concrete and reinforced concrete structures. Moscow, "Stroyizdat", 1976, 89 p.

SNiP II-22-81. Masonry and reinforced masonry structures, Russian Federation, Moscow, State Committee for Construction and Architecture, 2001. — 40 p.

SNiP 3.08.01-85. Mechanization of the construction process. Tower crane rail tracks.— M.: USSR State Committee for Construction, 1985.

SNiP 2.01.07-85. “Loads and actions” – M.: Central Institute of Standard Design, USSR State Committee for Construction, 1986.– 36 p.

SNiP 2.01.07-85*. Loads and actions. Building rules and regulations / State Committee of Russia for Construction, Moscow, 2001. — 44 p.

SNiP 2.02.01-83*. Foundations of structures / State Committee of Russia for Construction. Moscow, "Stroyizdat", 1984.

SNiP 2.03.13-88. Floors: Building rules and regulations / USSR State Committee for Construction.— Moscow, "Stroyizdat", 1988.— 27 p.

SNiP 2.02.03-85. Pile foundations / State Committee of Russia for Construction. Moscow, 2002.

SNiP 53-01-96. Steel structures: Design / V.Kucherenko Centr. Res. Inst. for Structural Constructions, M.P. Melnikov Central RDIsteelconstruction, UkrRDIsteelconstruction, Energosetproekt, and V.V. Kuibyshev Moscow Institute of Civil Engineering. — M., 1991.

SNiP II-23-81*. Steel structures / Ministry of Construction, Russia.— Moscow, 1996. — 96 p.

SNiP 2.01.01-82. Building climatology and geophysics. — Moscow, 1983. — 136 p.

SNiP II-3-79*. Building heat engineering. — Moscow, 1982. — 40 p.

SNiP II-7-81*. Construction in seismic regions. Building regulations / Ministry of Construction of Russia. Moscow, 1996. 52 p.

SP 52-101-2003. Concrete and reinforced concrete structures with no prestressing of reinforcement. Moscow, 2003, 126 p.

SP 53-102-2004. General rules for steel structure design, M.: Federal State Unitary Enterprise “Center for Design Products in the Construction Industry”,– 2005. – 131 p.

SP 50-101-2004. Design and construction of soil bases and foundations for buildings and structures, Moscow, –2005. – 130 p.

SP 50-102-2003. Design and construction of pile foundations, Moscow, 2005. – 81 p.

SP 20.13330.2011 "SNiP 2.01.07-85* "Loads and actions " / Ministry of Construction, Housing and Utilities of the Russian Federation. – M.: 2011.— 80 p.

SP 64.13330.2011 "SNiP 2.01.07-85* "Timber structures" / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011.— 87 p.

SP 22.13330.2011 "SNiP 2.02.01-83* "Soil bases of buildings and structures" / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011.— 162 p.

SP 24.13330.2011 "SNiP 2.02.03-85 "Pile foundations" / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011.— 86 p.

SP 16.13330.2011 "SNiP II-23-81* "Steel Structures" / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011.— 172 p.

SP 15.13330.2012 "SNiP II-22-81. Masonry and reinforced masonry structures” / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011. — 74 p.

SP 63.13330.2012 "SNiP 52-01-2003. Concrete and reinforced concrete structures. Guidelines" / Ministry of Construction, Housing and Utilities of the Russian Federation. — M.: 2011.— 156 p.

SP 16.13330.2017 "SNiP II-23-81* "Steel Structures" / Ministry of Construction, Housing and Utilities of the Russian Federation. – M.: 2017. – 148 p.

SP 294.1325800.2017 Steel Structures. Design Rules/ Ministry of Construction, Housing and Utilities of the Russian Federation. – M.: 2017.

SP 20.13330.2016 "SNiP 2.01.07-85* "Loads and actions ", Ministry of Construction, Housing and Utilities of the Russian Federation. – Moscow, 2017.

STR 2.05.08:2005. Plieninių konstrukcijų projektavimas. Pagrindinės nuostatos i skyrius. Bendrosios nuostatos.

TSN 102-00. Reinforced concrete structures with reinforcement of classes A500C and A400C: Local building regulations for Moscow.