ESPRI 2016 Manual

Engineering Assistance Package

(88 reference and calculation modules)

Kiev (Ukraine) 2017

Contents

General notes ... 6
Mathematics for engineer ... 7
Mathematics for engineer ... 7
Areas and volumes ... 8
Diagram multiplication ... 9
Linear algebra ... 11
Polynomial roots ... 15
Function interpolation ... 17
Advanced calculator ESPRI ... 18
Sections ... 22
Sections ... 22
Parametric sections ... 22
Parametric thin-walled sections ... 23
Separate sections as single unit ... 27
Moment of inertia in torsion ... 33
Static/Dynamic analysis, stability ... 37
Static and dynamic analyses, stability ... 37
Continuous beam ... 38
Truss ... 40
Parametric plane frame ... 43
Arbitrary plane frame ... 47
Arbitrary plane frame ... 47
Step 1. Generate model geometry ... 49
Step 2. Define boundary conditions (restraints) and hinges ... 50
Step 3. Define cross-sections for bars ... 51
Step 4. Define loads on design model ... 52
Options to edit and visualize the model ... 56
Output data ... 57
Sign conventions for displacements and forces ... 58
Rectangular slab on elastic foundation ... 58
Rectangular slab ... 62
Wall-beam ... 66
Shell on rectangular plan ... 70
Shell on circular plan ... 74
Mode shapes and frequencies of natural vibrations in cantilever ... 79
Stability factors and buckling modes in cantilever ... 81
Mode shapes and frequencies of natural vibrations in continuous beam ... 83
Cable and string ... 86
Cable and string ... 86
Cable and string. Input data ... 89
Cable and string. Output data ... 90
Cable and string. Notation ... 91
Influence lines in continuous beam ... 92
Steel Structures ... 95

Steel structures....95
Steel table....95
Analysis of steel elements....98
Principal and equivalent stresses in steel structures....105
Effective lengths of steel structure elements....107
Parametric joints of steel structures....110
Analysis of welds....115
Bolted connections....125
Cold-formed shapes....134
Reinforced concrete (RC) structures....140
Reinforced concrete (RC) structures....140
Properties of concrete....141
Table of reinforcement....142
Anchorage of reinforcement by DSTU 3760-07....143
Sections of RC elements....145
Reinforced concrete shell....147
Reinforced concrete wall-beam....148
Reinforced concrete slab....151
Principal and equivalent stresses in reinforced concrete structures....153
Strengthening with composite materials....155
Strength of reinforced concrete butt joint in shear....158
Concrete sections with fibre-reinforced polymer (FRP) bars....160
Concrete pipe sections....163
Composite steel and concrete columns....167
Column strengthened with composed materials....170
Composite steel and concrete slabs....173
Masonry and masonry reinforcing....178
Masonry and masonry reinforcing....178
Design compression strength in masonry....178
Brick pier....179
Masonry in local compression....183
Brick pier in tension....186
Brick pier by DBN V.2.6-162:2010....188
Timber structures....206
Timber structures....206
Sections of solid timber....206
Sections of glued timber....208
Sections of composite timber....210
Foundations and beddings....213
Foundations and beddings....213
Moduli of subgrade reaction C1, C2....214
Single pile....224
Pile under combined action of loads....228
Settlement of equivalent footing....231
Principal and equivalent stresses in soil....233
Slope stability....236
Stability of multi-layer slope....239

Bearing capacity of piles by field test results 243
Combined piled-raft foundation 244
Loads and actions 247
Loads and actions 247
Load factors 247
Dead weight of multi-layer coating 249
Snow loads 252
Wind loads 254
Ice loads 256
Climatic thermal loads 258
Hazardous energy combinations of forces (EnergyCF) 261
Resonance check for wind turbulence 265
Deflections 268
Deflections 268
Analysis of inelastic deflections 268
Ellipsoid 272
Ellipsoid 272
Ellipsoid. Bearing capacity of RC elements 272
Sheet piling 277
Sheet piling (chapter) 277
Sheet piling (module) 278
Diaphragm 285
Diaphragm 285
Strength of RC diaphragm in earthquake loads 285
Punching shear 295
Punching shear 295
Punching shear for arbitrary contour 296
Punching shear for rectangular contour 299
Punching shear analysis (Eurocode) 304
Punching shear analysis (Belarus) 309
Punching shear for circular contour (Eurocode) 313
Toster 319
TOSTER (Chapter) 319
TOSTER - General notes 319
Input data 321
Options to edit the model 336
Options to visualize the model 336
Circular arc 337
Output data 339
Sign convention 350
Prestressing 351
Prestressing (Chapter) 351
Prestressing 351
Analysis of RC support sections (posts) 356
Soil 359

Soil (Chapter)....359
Soil....359
Input data....360
Visualization of soil model....365
Analysis parameters....366
Output data....367
Calculation of subgrade moduli C1, C2 (summary)....373
Bibliography....376
Index....378

General notes

ESPRI 2016 (Engineering Assistance Package) is a set of reference and calculation programs for the day-to-day needs of civil and structural engineers.

ESPRI software contains programs (modules) that enable you to carry out computer-aided analyses for a wide range of tasks in design, engineering and research fields of construction.

Since all modules of the package are very user-friendly and calculation procedure requires minimum amount of time, ESPRI may be successfully applied for different purposes, such as: generating design model of the structure, evaluation of analysis results, expert appraisal of projects and technical supervision during erection of the structure.

ESPRI may be essential in many situations that require evaluation of real structures behaviour during erection, reconstruction, design procedure and supervision for their maintenance.

ESPRI helps the user in the routine work and provides the user with support to come up with an optimal solution. Modules of ESPRI program are significant supplement to the more powerful software for civil and structural engineering – LIRA-SAPR and MONOMAKH-SAPR.

The software package contains 85 reference and calculation programs (modules) that are organized in the following chapters:

Mathematics for engineer
Sections
Static/Dynamic analysis, stability
Steel structures
Reinforced concrete (RC) structures
Masonry and masonry reinforcing
Timber structures
Foundations and beddings
Loads and actions
Deflections
Ellipsoid
Sheet piling
Diaphragm
Punching shear
TOSTER
Prestressing
Soil

Mathematics for engineer

Mathematics for engineer

This chapter contains the following modules:
1. Areas and volumes
2. Diagram multiplication
3. Linear algebra:
 – inverting a matrix,
 – determining eigenvalues and eigenvectors;
 – solving a set of linear equations;
 – multiplying matrix by matrix (vector),
 – computing the determinant of a matrix.
4. Polynomial roots
5. Function interpolation
6. Advanced calculator ESPRI
– Calculator;
– Conversion of measurement units;
– Computing definite integral.

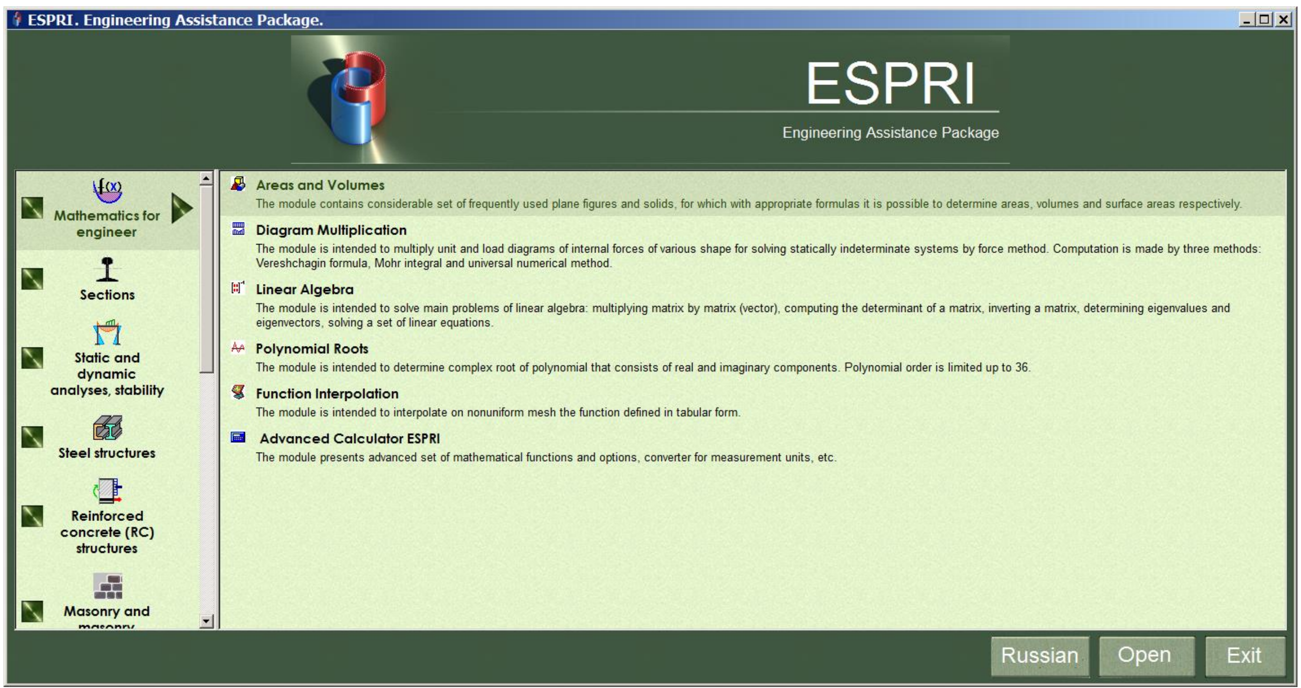


Figure 1.1

Areas and volumes

The module contains considerable set of frequently used plane figures and solids, for which with appropriate formulas it is possible to determine areas, volumes and surface areas respectively.

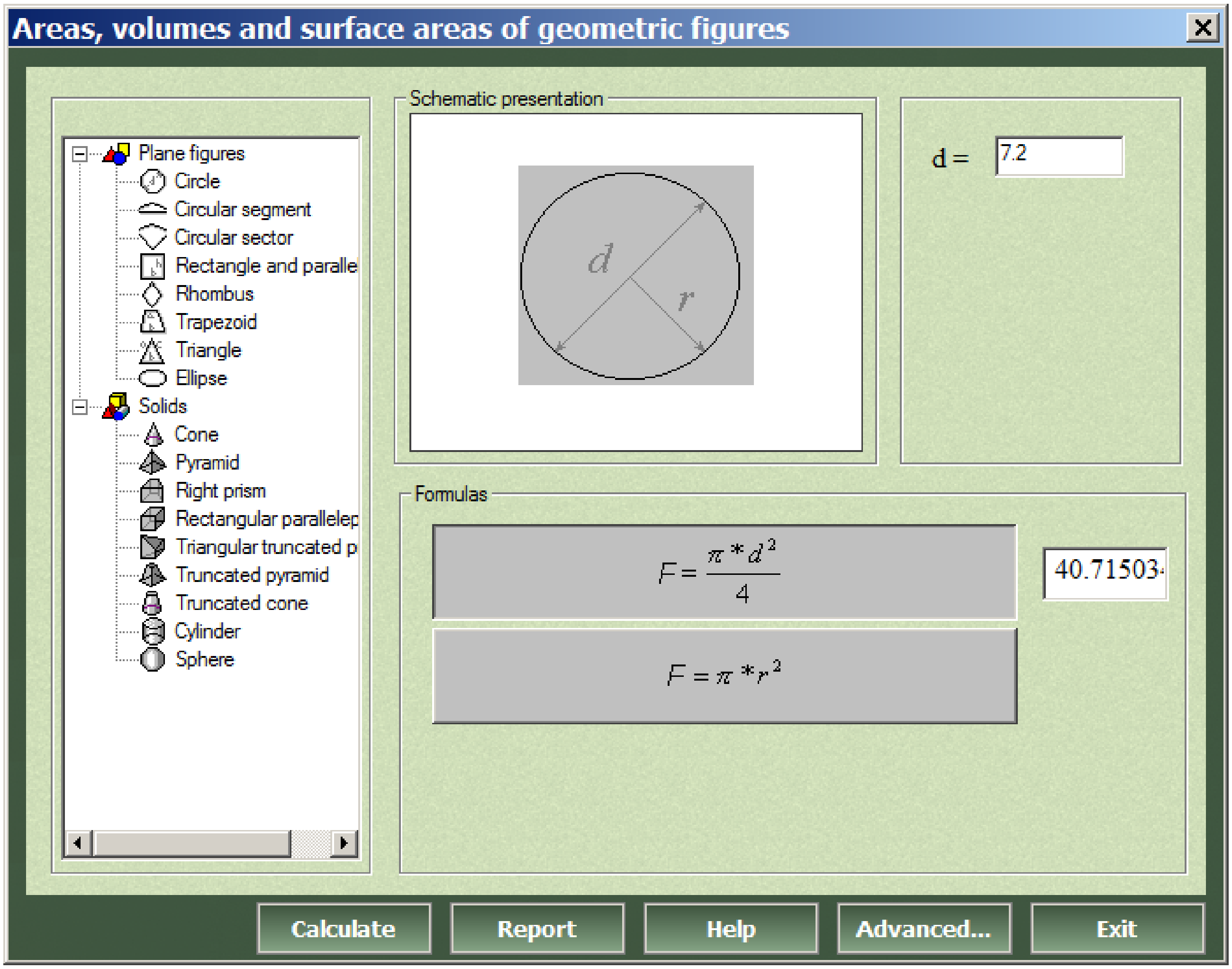


Figure 1.2

To define input data, follow these steps:

– In the **Areas, volumes and surface areas of geometric figures** dialog box, click appropriate plane figure or solid.

– Define input data in the appropriate boxes.

– Select formula for calculation.

– Click **Calculate**.

Input data for plane figures:

d – diameter of circle;

r – radius of circle;

α – opening angle of circular segment or sector;

a – height of rhombus, side of trapezoid or triangle, semi-axis of ellipse;

b – side of rectangle, parallelogram, trapezoid or triangle, height of rhombus, semi-axis of ellipse;
h – height of rectangle, parallelogram, trapezoid or triangle.

Output data for plane figures:
F – area.

Input data for solids:
r – base radius of cone, top base radius of truncated cone, radius of cylinder, radius of sphere;
g – bottom base radius of truncated cone;
d – diameter of sphere;
a, b, c – sides of rectangular parallelepiped or truncated triangular prism;
h – height of cone, pyramid, right prism, truncated pyramid, truncated cone, cylinder;
F – base area of pyramid, right prism, truncated triangular prism;
F1, F2 – top and bottom base area of truncated pyramid respectively.

Output data for solids:
V – volume;
S – surface area of solid.

Diagram multiplication

The module is intended to multiply unit and load diagrams of internal forces of various shape for solving statically indeterminate systems by force method. Computation is made by three methods: Vereshchagin formula, Mohr integral and universal numerical method.

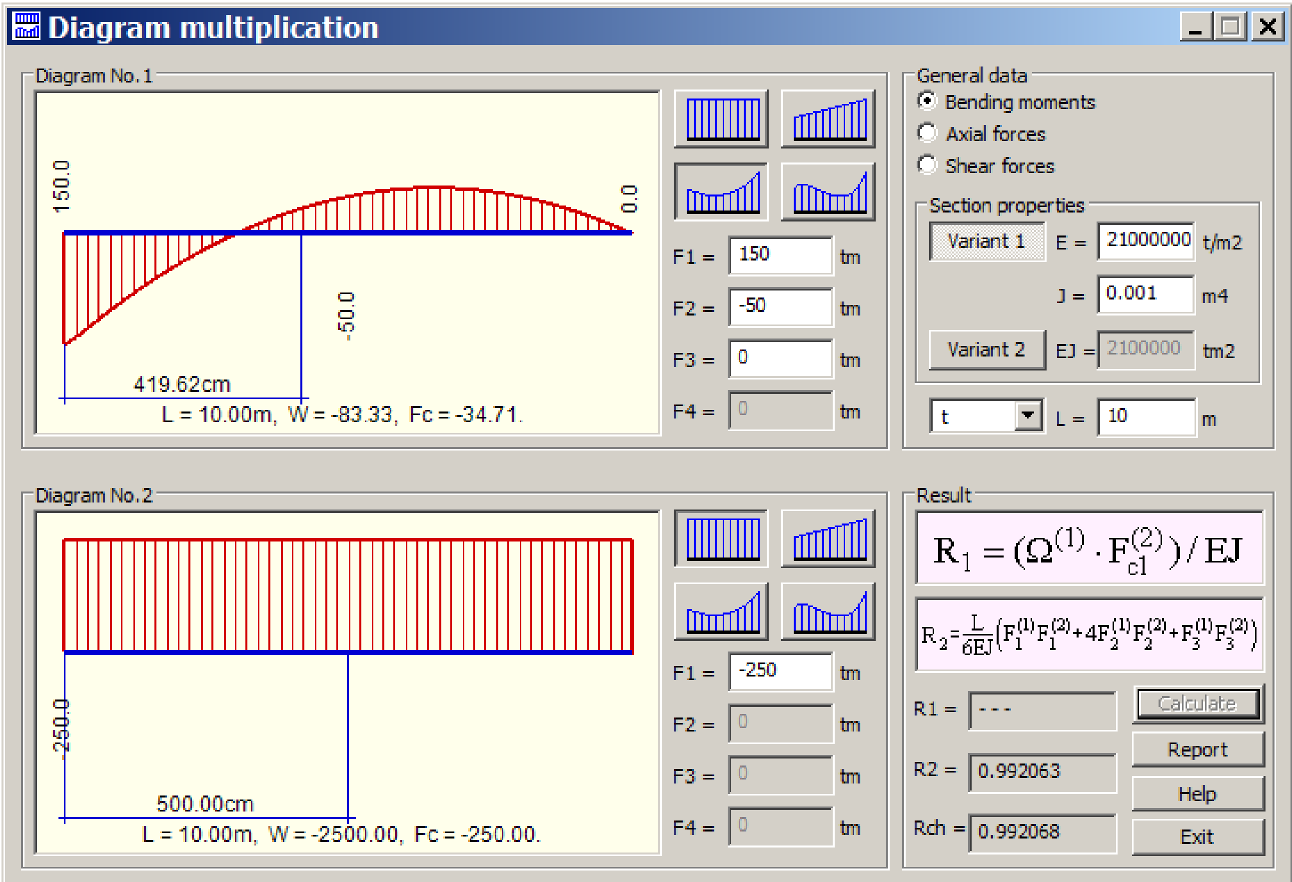


Figure 1.3

To define input data, follow these steps:

– Select appropriate diagram No.1 and diagram No.2 with the pointer.

– Define ordinates for characteristic points of diagrams F_1, F_2, F_3, F_4 .

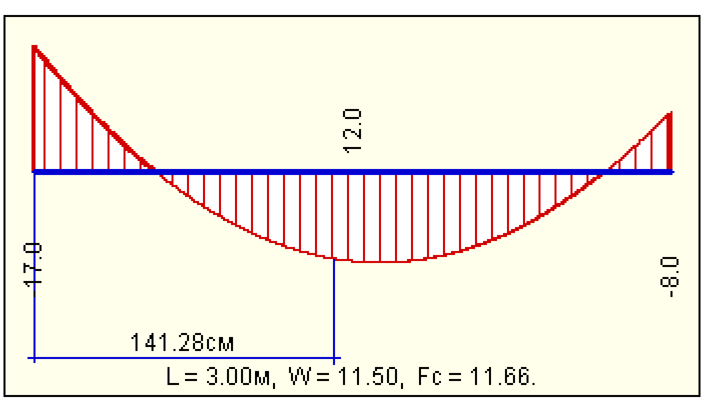
– Define type of internal forces for calculation – **bending moments**, **axial forces**, **shear forces**.

– Define section properties for **Variant 1** or **Variant 2**: E, G, I, A, EI, EA, GA – modulus of elasticity, shear modulus, geometric properties of section, stiffness properties of section;

– In the appropriate list box, select measurement units – **t** or **kN**;

– Define length of the line **L** (m);

– Click **Calculate**.



Notation for the scheme above:

L – length of line (m),
W – area of diagram (with account of sign),
Fc – value of ordinate at gravity centre of the diagram,
141.28 – location of gravity centre (cm).

Calculation of Mohr integral

General formula for multiplication of diagrams for 3D bar systems:

$$\Delta pi = \sum \int_{s} \left[\frac{M_x^P M_x^i}{GJ_x} + \frac{M_y^P M_x^i}{GJ_y} + \frac{M_z^P M_z^i}{GJ_z} + \frac{N^P N^i}{EA} + K_y \frac{Q_y^P Q_y^i}{GA} + K_z \frac{Q_z^P Q_z^i}{GA} \right] ds.$$

R1 – Calculation by Vereshchagin formula.
R2 – Calculation by Simpson-Kornoukhov formula.
Rch – For any kind of diagrams, the result is calculated by numerical methods.

Linear algebra

The module is intended to solve main problems of linear algebra:
– multiplying matrix by matrix (vector),
– computing the determinant of a matrix,
– solving a set of linear equations;
– inverting a matrix,
– determining eigenvalues and eigenvectors.

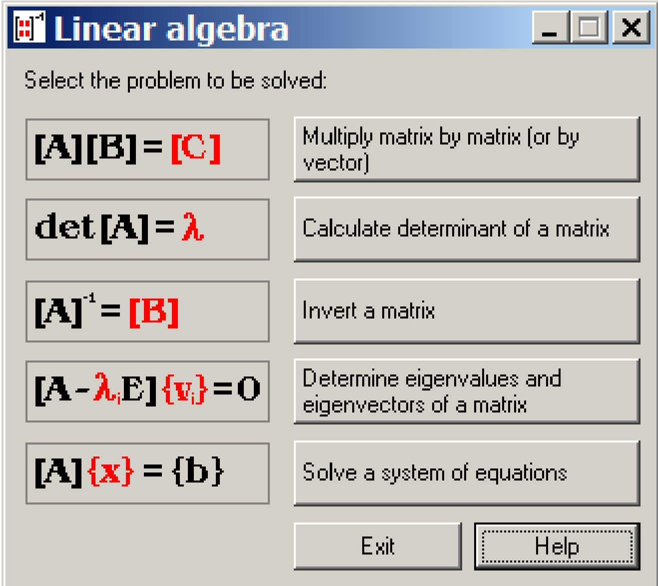
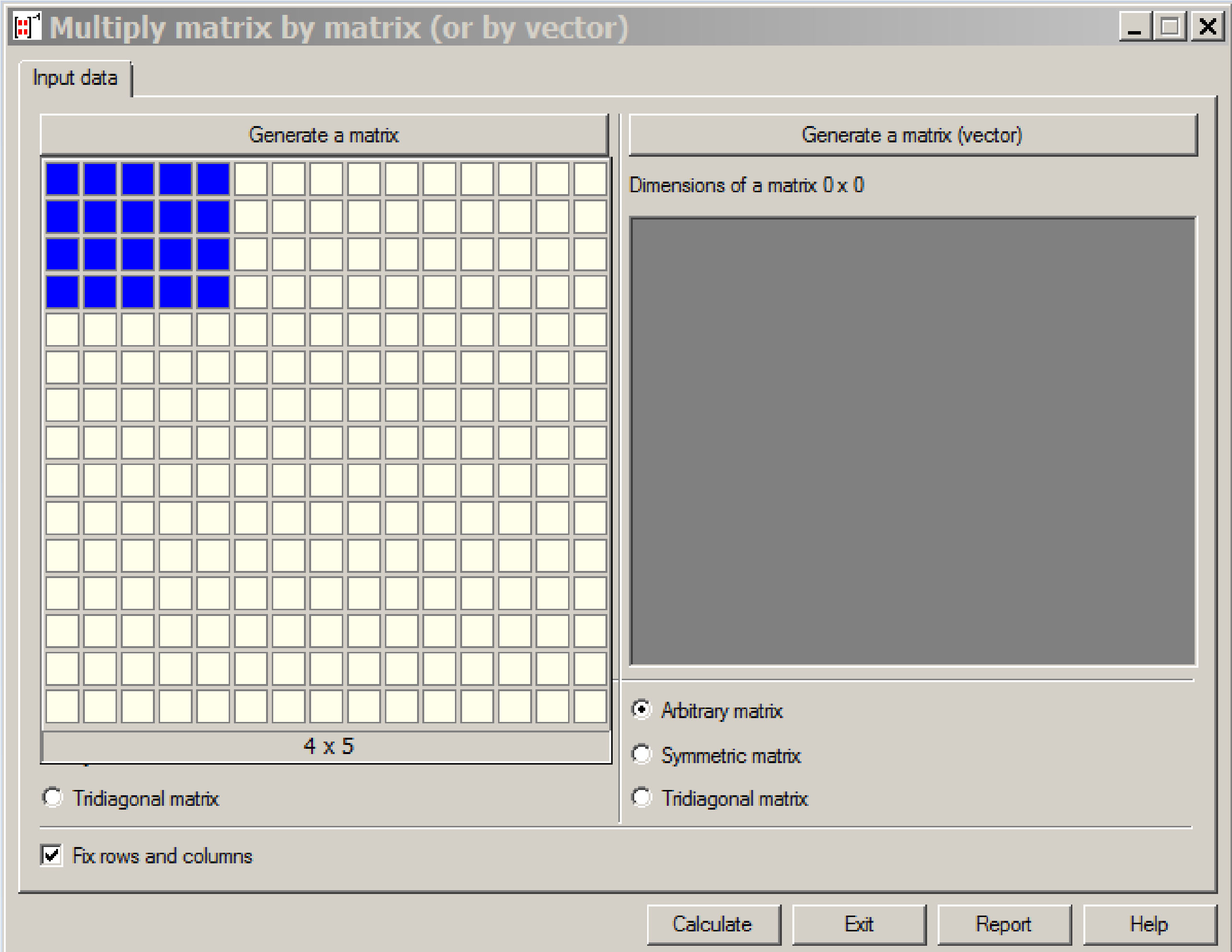
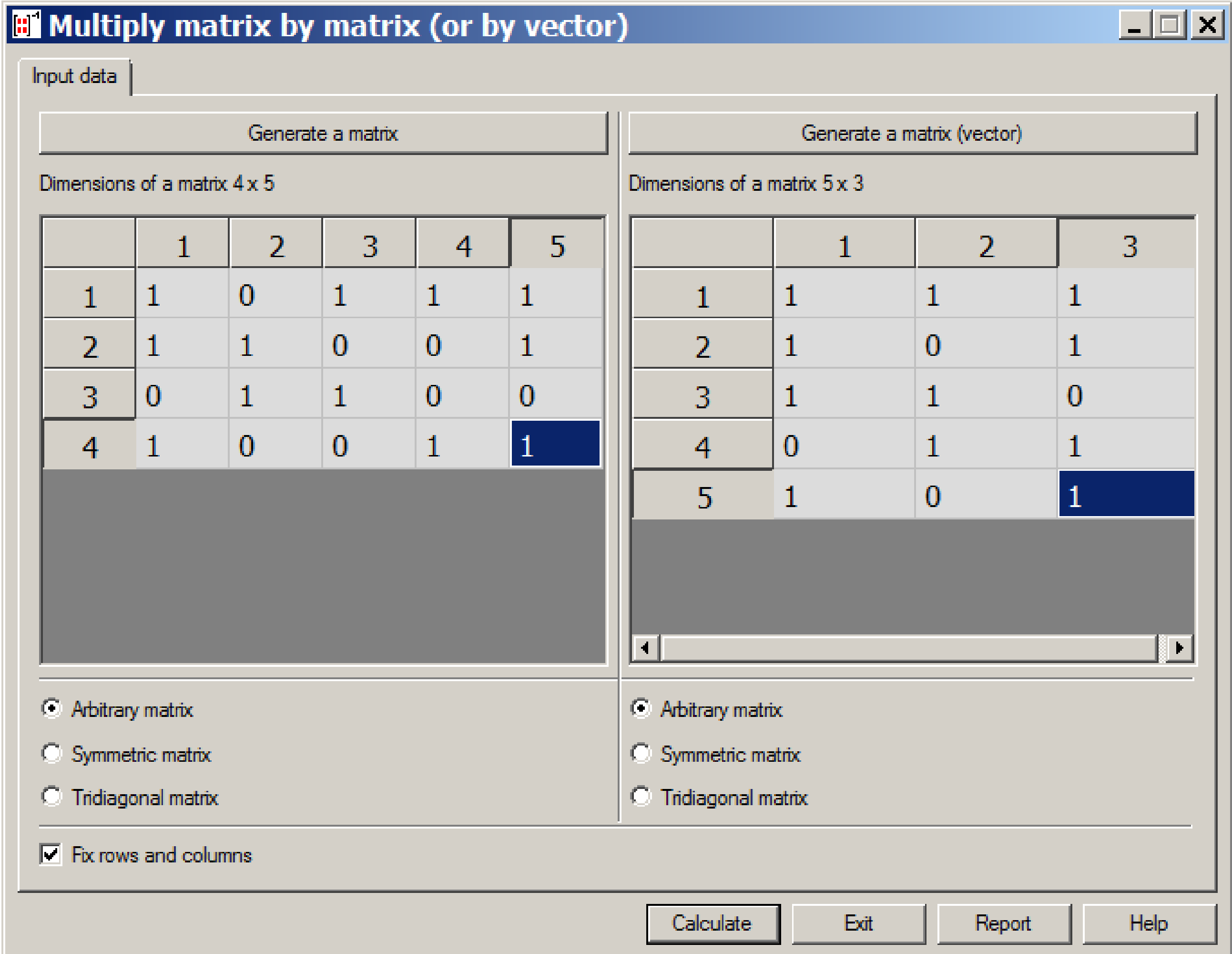


Figure 1.4

To define input data, follow these steps:
– Define necessary procedure with the pointer or use the TAB key to move through a dialog box.
– In another dialog box, define the matrix type with the option button:
  – Arbitrary matrix;
  – Symmetric matrix;
  – Tridiagonal matrix.
– Click **Generate a matrix (vector)**.
– Define dimension for input matrix by moving the pointer diagonally from the left upper corner (see Fig. 1.5).


Multiply matrix by matrix (or by vector)
Input data
Generate a matrix
Generate a matrix (vector)
Dimensions of a matrix 0 x 0
4 x 5
Arbitrary matrix
Symmetric matrix
Tridiagonal matrix
Tridiagonal matrix
Fix rows and columns
Calculate
Exit
Report
Help
Figure 1.5 a)


Multiply matrix by matrix (or by vector)
Input data
Generate a matrix
Generate a matrix (vector)
Dimensions of a matrix 4 x 5
Dimensions of a matrix 5 x 3
1 2 3 4 5
1 1 0 1 1 1
2 1 1 0 0 1
3 0 1 1 0 0
4 1 0 0 1 1
1 2 3
1 1 1 1
2 1 0 1
3 1 1 0
4 0 1 1
5 1 0 1
Arbitrary matrix
Symmetric matrix
Tridiagonal matrix
Arbitrary matrix
Symmetric matrix
Tridiagonal matrix
Fix rows and columns
Calculate
Exit
Report
Help
Figure 1.5 b)

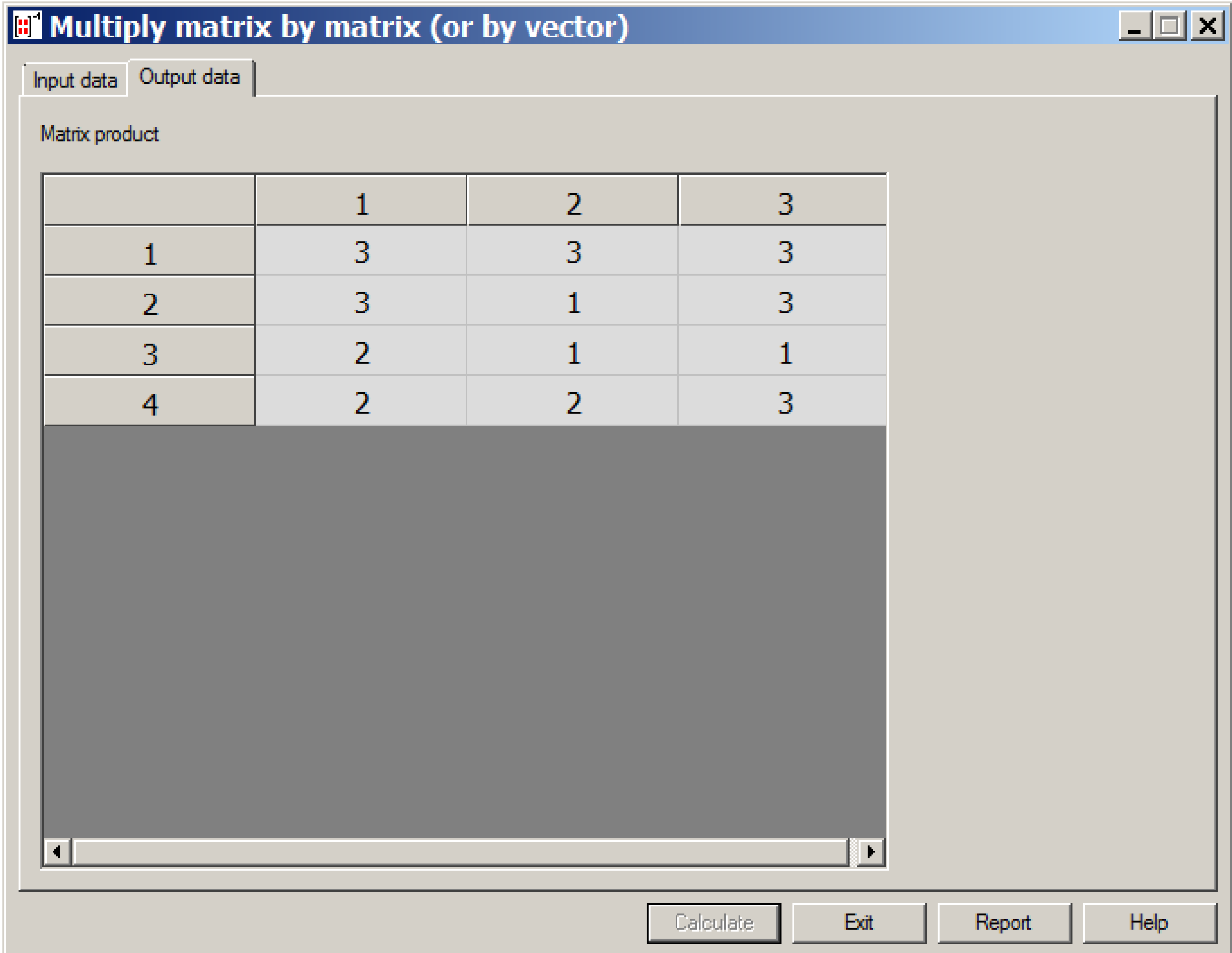


Figure 1.5 c)

– To define numerical values for matrix elements, just click appropriate cells in the table.
– Define the second matrix in the same way (see Fig. 1.5).
– Click **Calculate**.

On the **Output data** tab you will see results according to the procedure you selected.

Limitation to the order of problems is equal to 15.

Polynomial roots

The module is intended to determine complex root of polynomial that consists of real and imaginary components.

$$a_0 + a_1 x + a_2 x^2 + \ldots + a_{n-1} x^{n-1} + a_n x^n = 0.$$

Polynomial order **n** is limited up to 36.

Input data.

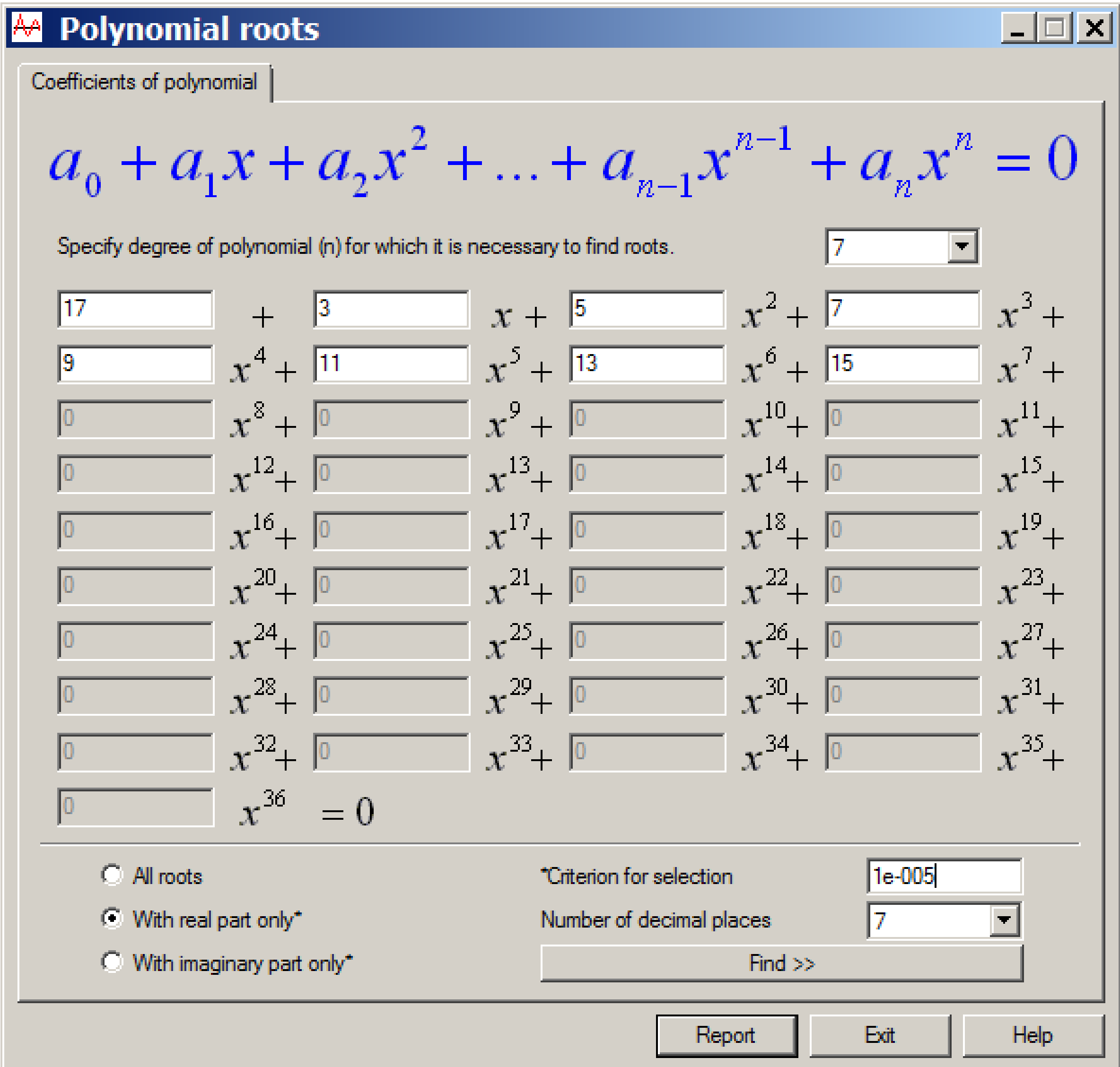


Figure 1.6

To define input data, follow these steps:
– Specify degree of polynomial in the appropriate box at the right part of the dialog box.
– Define values for coefficients.
– Specify one of the options:
  – all roots;
  – with real part only;
  – with imaginary part only.
– For the last two options, define **Criterion for selection**, that is, appropriate parts of the root that are less than this value will be considered as zero ones.
– Define the **Number of decimal places** for the output data.
– To calculate roots, click **Find**.

Output data.

Values for roots:
Re(Xi) – real part,
Im(Xi) – imaginary part.
CABS[P(X)] – result of root substitution in the polynomial (to evaluate solution accuracy).

Note. When finding roots of polynomial, an overflow may occur sometimes. Then the *1.#IND* symbols will appear in appropriate cells. It is not recommended to use such roots.

Function interpolation

This module is intended to interpolate on nonuniform mesh the function defined in tabular form and to calculate values of interpolation function from arbitrary defined arguments.

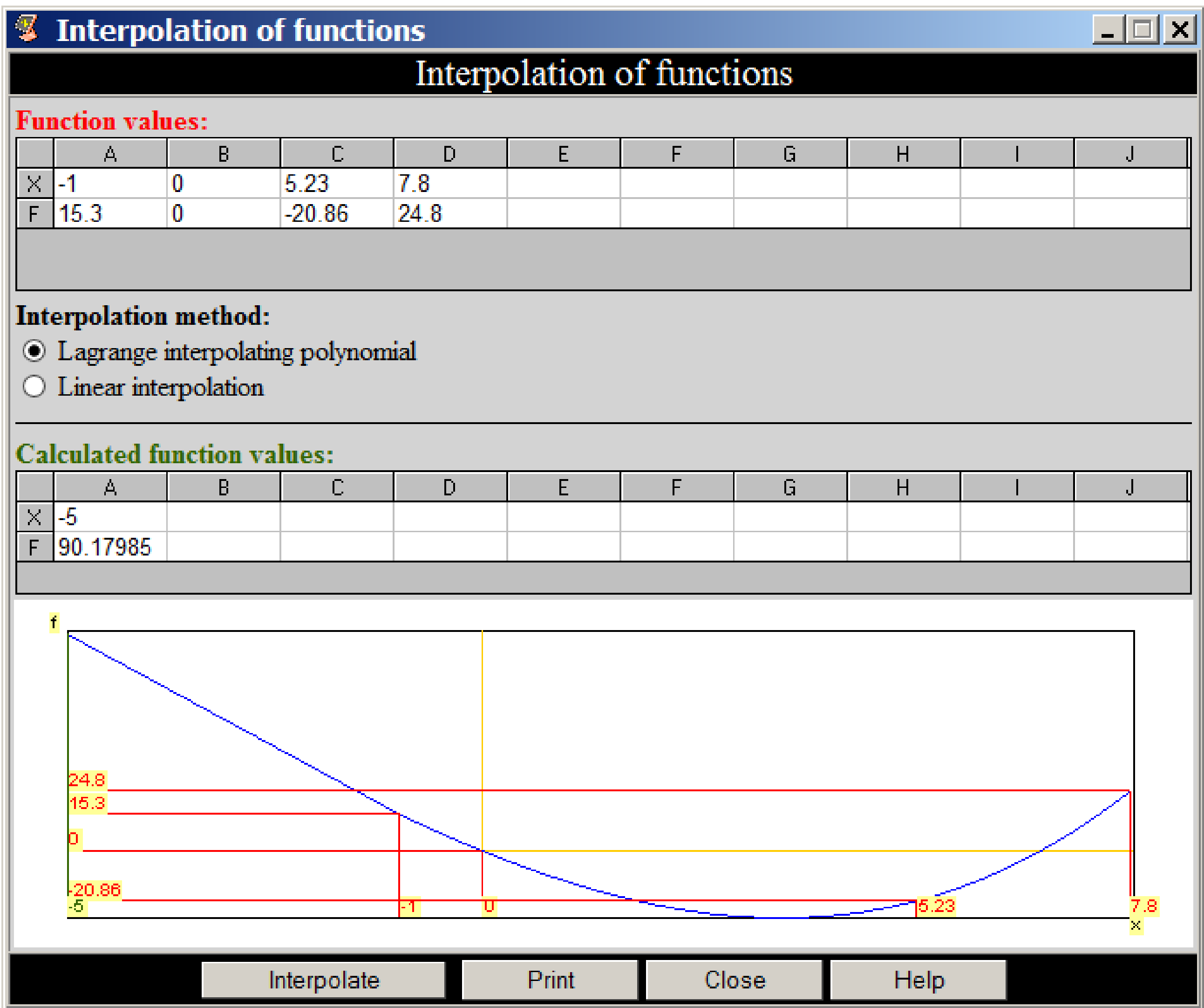


Figure 1.7

To define input data, follow these steps:
– In the **Interpolation of functions** table, define (in arbitrary order) ordinate values and corresponding function values.
– Define **Interpolation method** with appropriate option button:
  – **Linear interpolation** - interpolation only within defined values of **X**;
  – **Largange polynomial** - minor departure of input points from defined range of **X**.

Lagrange polynomial

$$L_n(x) = \sum_{i=0}^{n} F_i \left(\frac{(x-x_0)\ldots(x-x_{i-1})(x-x_{i+1})\ldots(x-x_n)}{(x_i-x_0)\ldots(x_i-x_{i-1})(x_i-x_{i+1})\ldots(x_i-x_n)} \right)$$

– In the **Calculated function values** table, input values of **X** for which it is necessary to calculate function values.
– Click **Interpolate**.

In the **Calculated function values** table, under appropriate **X** values you will see function values. The function will be sketched on the **f(x)** graph below.

Advanced calculator ESPRI

The module Advanced Calculator ESPRI presents advanced set of mathematical functions and options, such as:
– to calculate expressions defined by the user as formulas;
– to convert different units of measurement;
– to compute definite integral for arbitrary defined function within the range from *a* to *b*.

The dialog box contains three tabs: **Calculator**, **Convert units of measurement** and **Definite integral**.

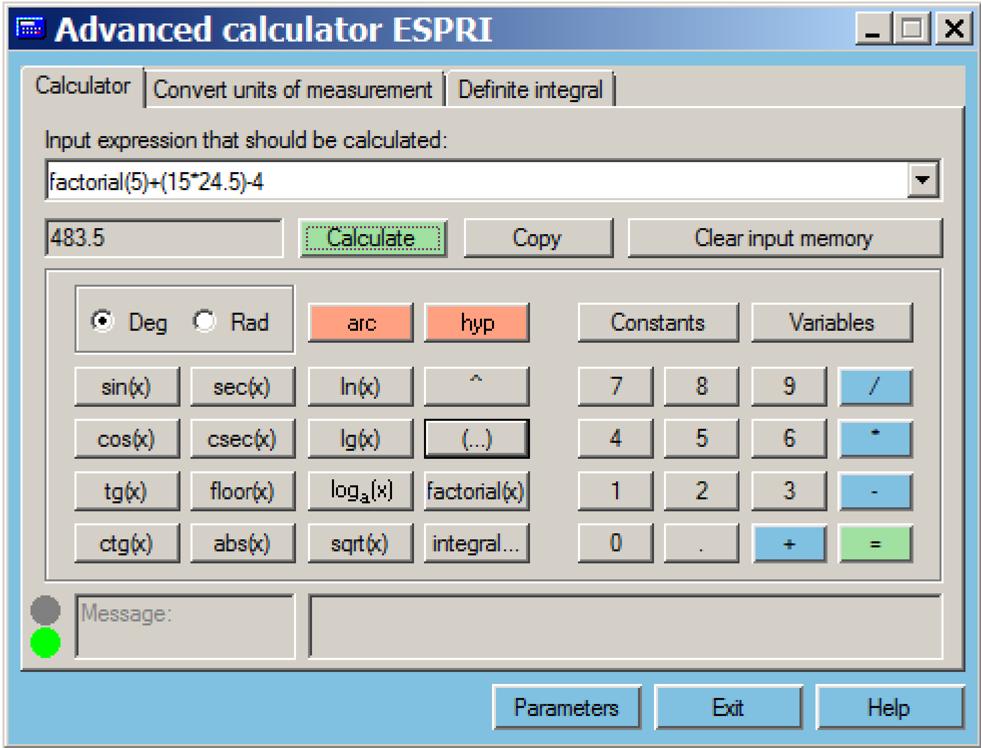


Figure 1.8 a)

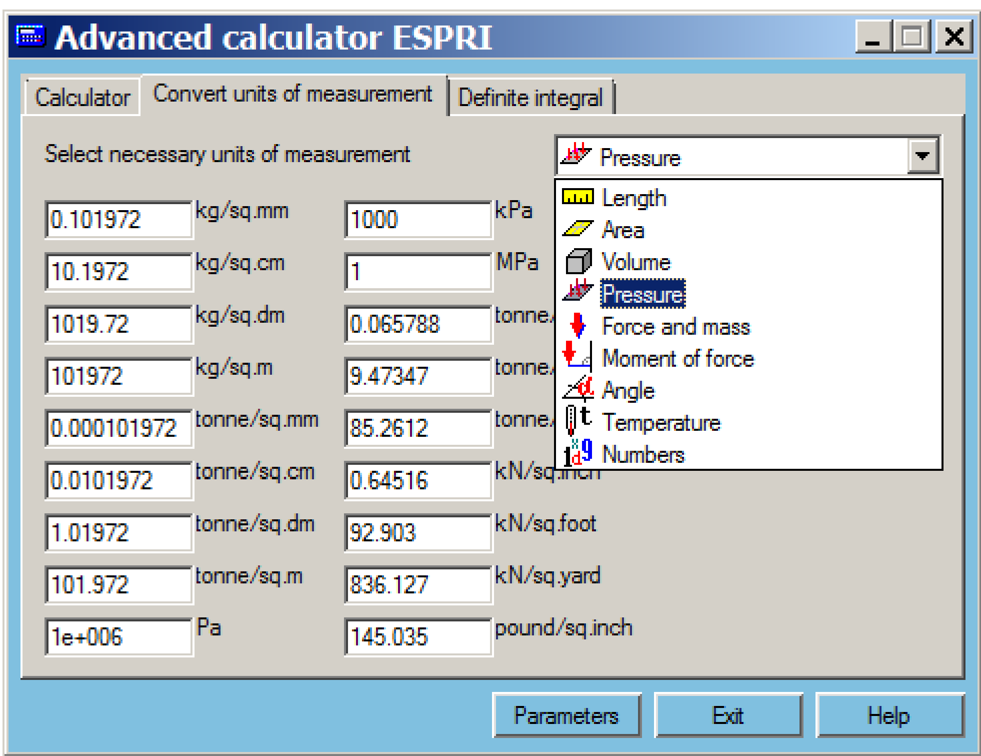


Figure 1.8 b)

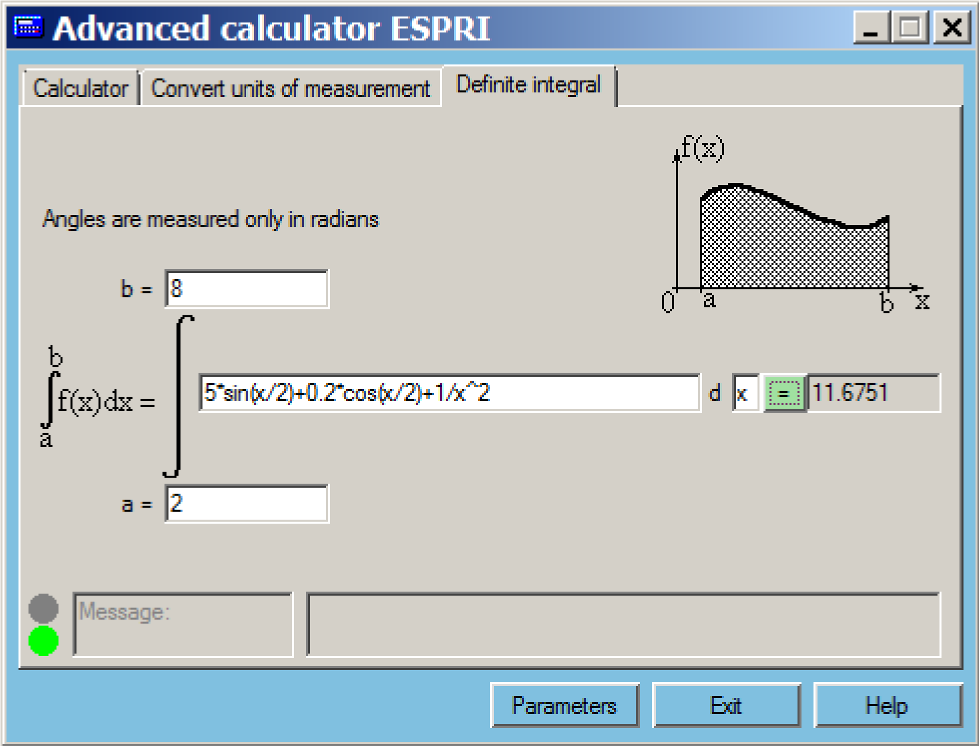


Figure 1.8 c)

To input expressions in the **Calculator** program, use appropriate buttons for algebraic and trigonometric functions or the keyboard. Complete guide on 'How to input formulas and functions' you will find in the Help system for the program. To calculate the value of expression, click **Calculate**.

When you click the **Constants** or **Variables** buttons, the dialog box is expanded to display the **Value** window with two appropriate tabs.
With the **Constants** button, it is possible to select a constant from the set and input its value to the line of expression beginning from the pointer location. To do this, select appropriate constant and click **Apply**.
With the **Variables** button, you could save and apply values for calculation results. To define value for a variable, you could simply type the value in the appropriate cell or (if it is result of calculation) copy it with the **Copy** button and paste into appropriate cell with SHIFT+INSERT or CTRL+V shortcut keys. To use variable in the line of expression, just indicate the name of variable.

On the **Convert units of measurement** tab you can convert different types of measurement units.
The program enables you to convert measurement units for the following categories:
– Length
– Area
– Volume
– Pressure
– Force and mass
– Moment of force
– Angle

– Temperature
– Number.

On the **Definite integral** tab, it is possible to compute definite integral (from function within the range from a to b) for the expression defined by the user. Instructions for the defined expression remain the same as for the formulas in **Calculator**. Any variable from the **Variables** dialog box may be considered as integration variable. The rest of variables (except the selected one) included into expression are considered as constants. To compute definite integral for the function, click [=] button.

In case of any error, you will see an error message in the lower part of **Calculator** tab.

In the **Parameters** dialog box you could select language for the program, define number of expressions that will be saved in **Calculator** (the most recent ones that were defined), define several parameters for calculator itself:

– to **start up with Windows**. When this option is selected, the icon will be displayed in notification area on the taskbar. To start the **Calculator** module, just double-click this icon. When you right-click the icon, it is possible to display context menu with options to present the dialog box with copyright information or to close the program (only **Calculator** module will be closed; if ESPRI software is active at the moment, it will remain active);

– to **allow multiple document interface**. If this check box is selected, you will be able to start one more copy of the **Calculator** module and vice versa.

Sections

Sections

With modules of this chapter you could compute stiffness parameters of solid, thin-walled sections and separate sections considered as single unit.

The chapter contains the following modules:
– Parametric sections;
– Parametric thin-walled sections;
– Separate sections as single unit;
– Moment of inertia in torsion.

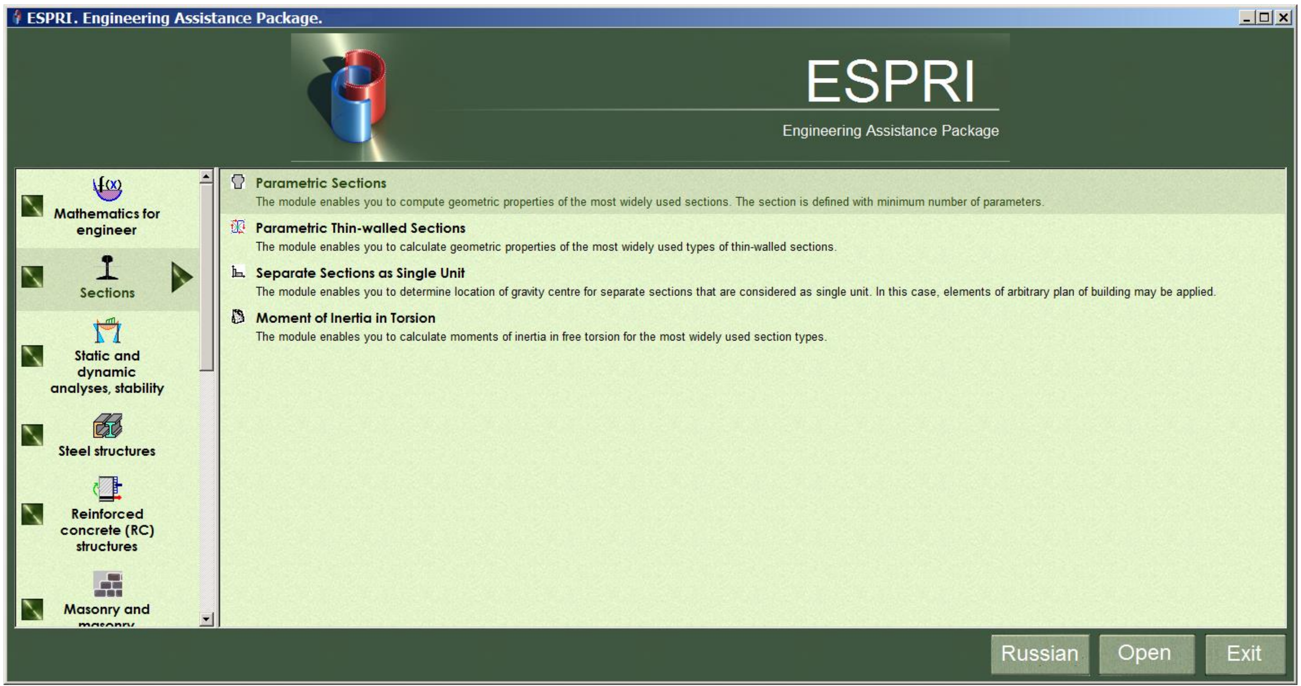


Figure 2.1

Parametric sections

The module enables you to compute geometric properties of the most widely used sections (22 types). The section is defined with minimum number of parameters.

Input data.

To define input data, follow these steps:
– click the section you would like to compute;
– select measurement units for input and output data;
– define necessary parameters of the section (see schematic presentation);

 www.liraland.com

– click **Calculate** (this button becomes available only when parameters that describe selected section are defined correctly).

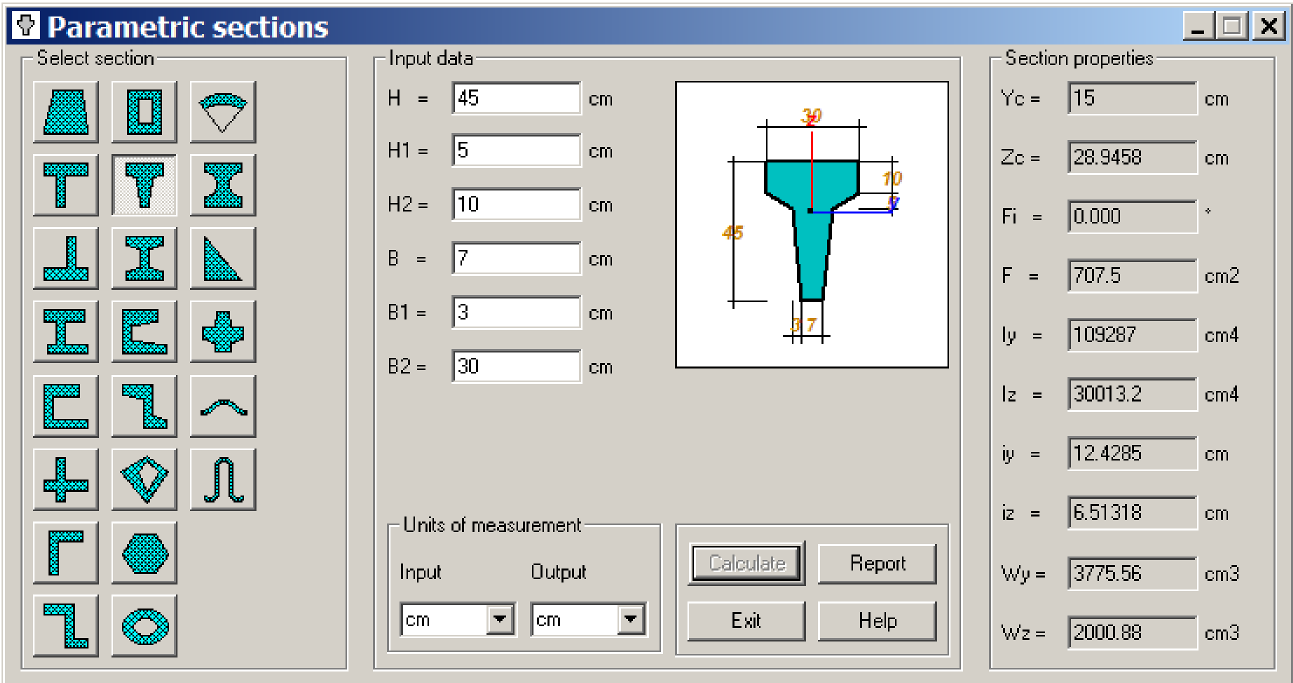


Figure 2.2

Output data.
Calculated geometric properties of the section:

Yc, Zc – coordinates of gravity centre in initial axes;
Fi – slope angle of principal axes of inertia;
F – area of section;
Jy, Jz – principal moments of inertia;
iy, iz – radii of inertia;
Wy, Wz – section moduli.

Parametric thin-walled sections

The module enables you to calculate geometric and sectorial stiffness properties of thin-walled sections.

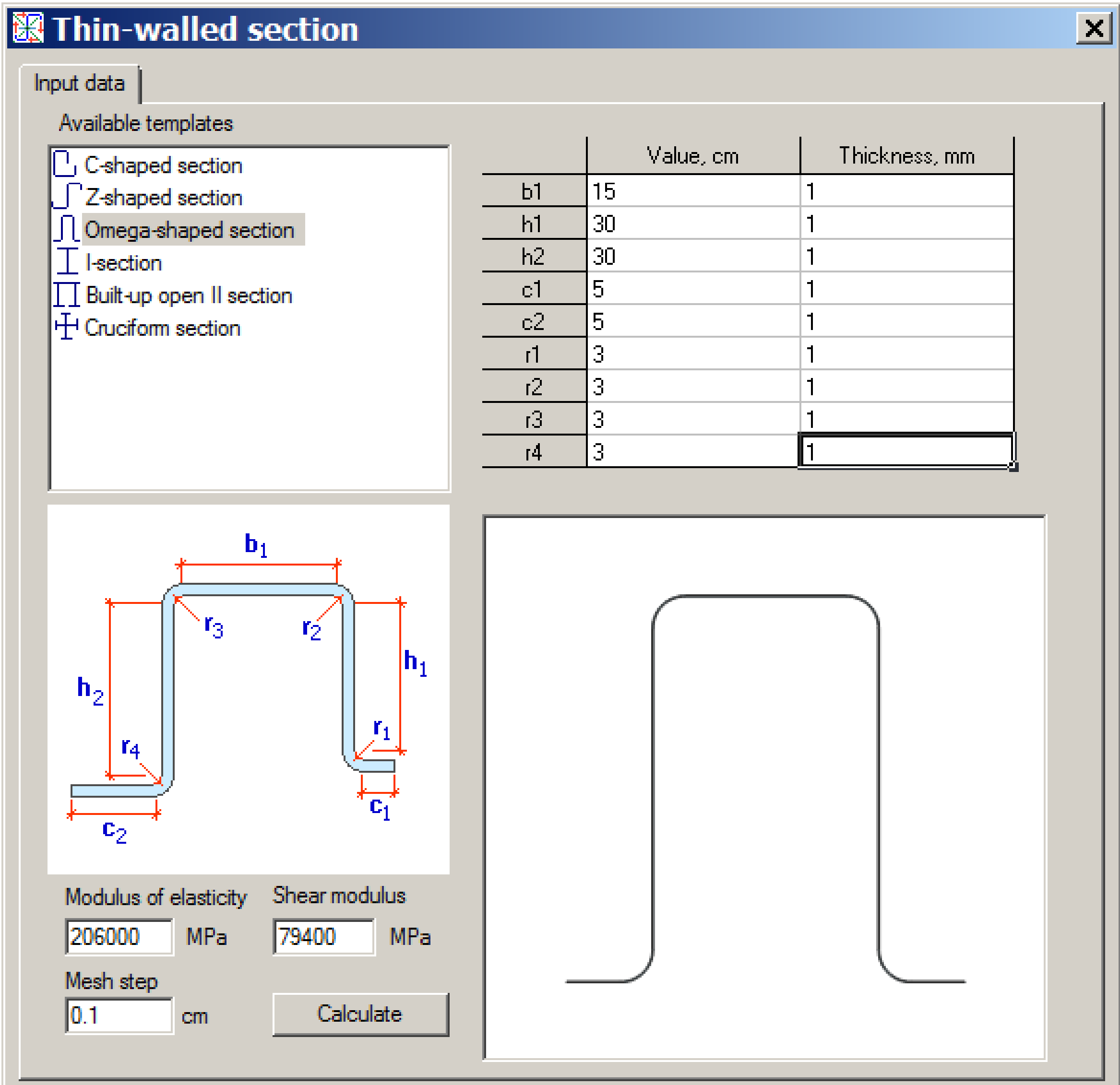


Figure 2.3

In this dialog box, on the **Input data** tab, select with the pointer the type of shape, define its dimensions and thickness according to schematic presentation and define modulus of elasticity, shear modulus and step for interior division of the section (for calculation).
Click **Calculate**. In the window below (see Fig. 2.4) you will see the stages of computation and its statistics.

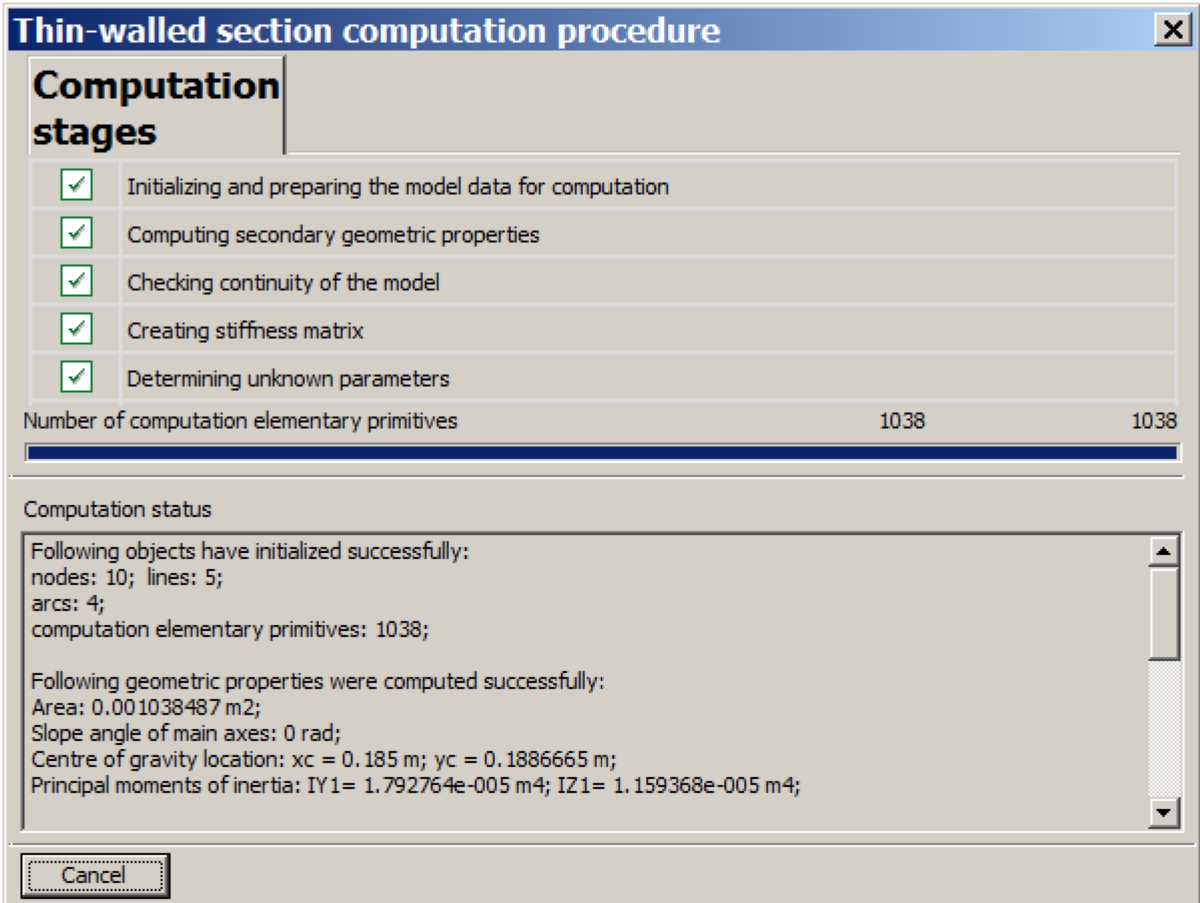
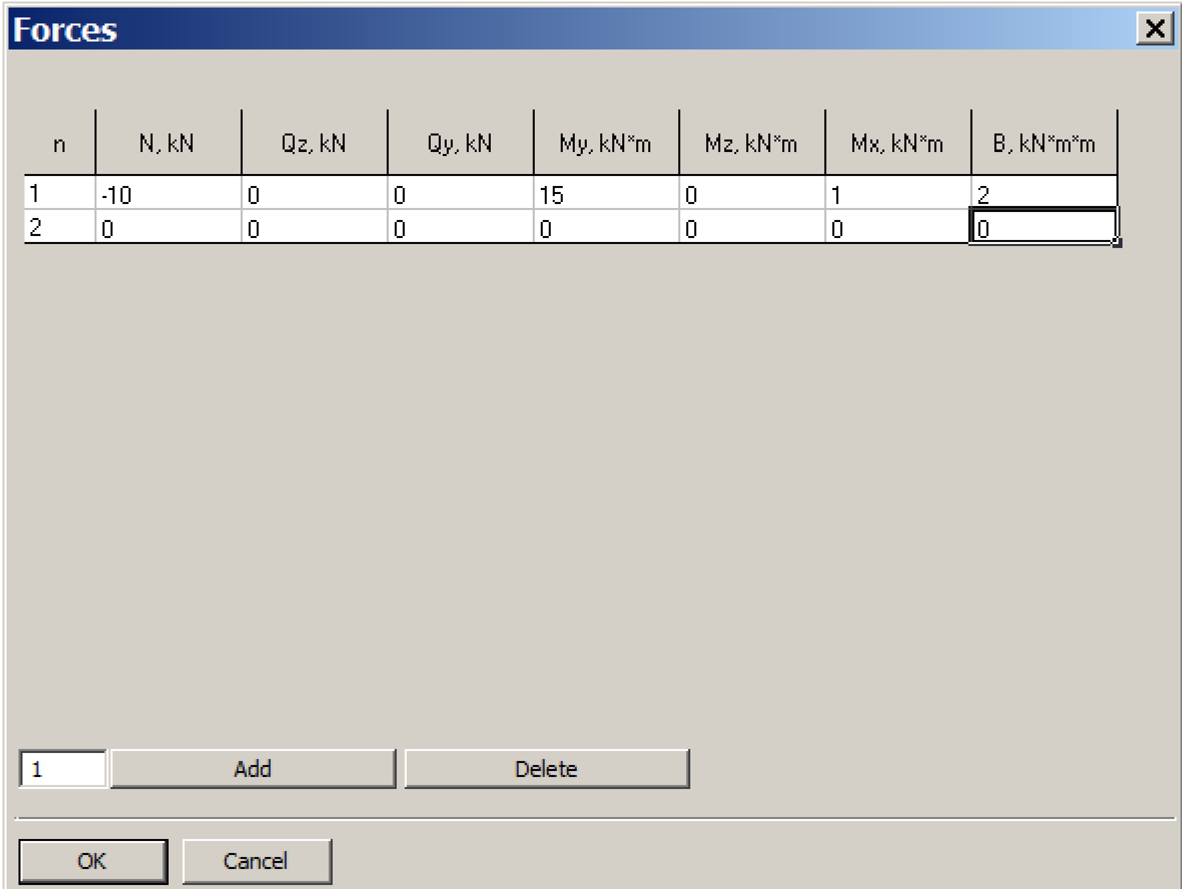


Figure 2.4

Click **Cancel** (see Fig. 2.4). The **Thin-walled section** window will be displayed automatically.

Output data.

After computation, you will obtain geometric and sectorial properties for the section. Diagrams of axes' coordinates, diagram of warping coordinate and stress diagrams (when forces at the section are defined) are also available. To display the diagrams, in the **Thin-walled section** window, in the drop-down list, select appropriate diagram.
To define forces for the thin-walled section, on the **Actions** menu, click **Modify forces** (button  on the toolbar).


Forces
n
N, kN
Qz, kN
Qy, kN
My, kN*m
Mz, kN*m
Mx, kN*m
B, kN*m*m
1
-10
0
0
15
0
1
2
2
0
0
0
0
0
0
0
1
Add
Delete
OK
Cancel
Figure 2.5

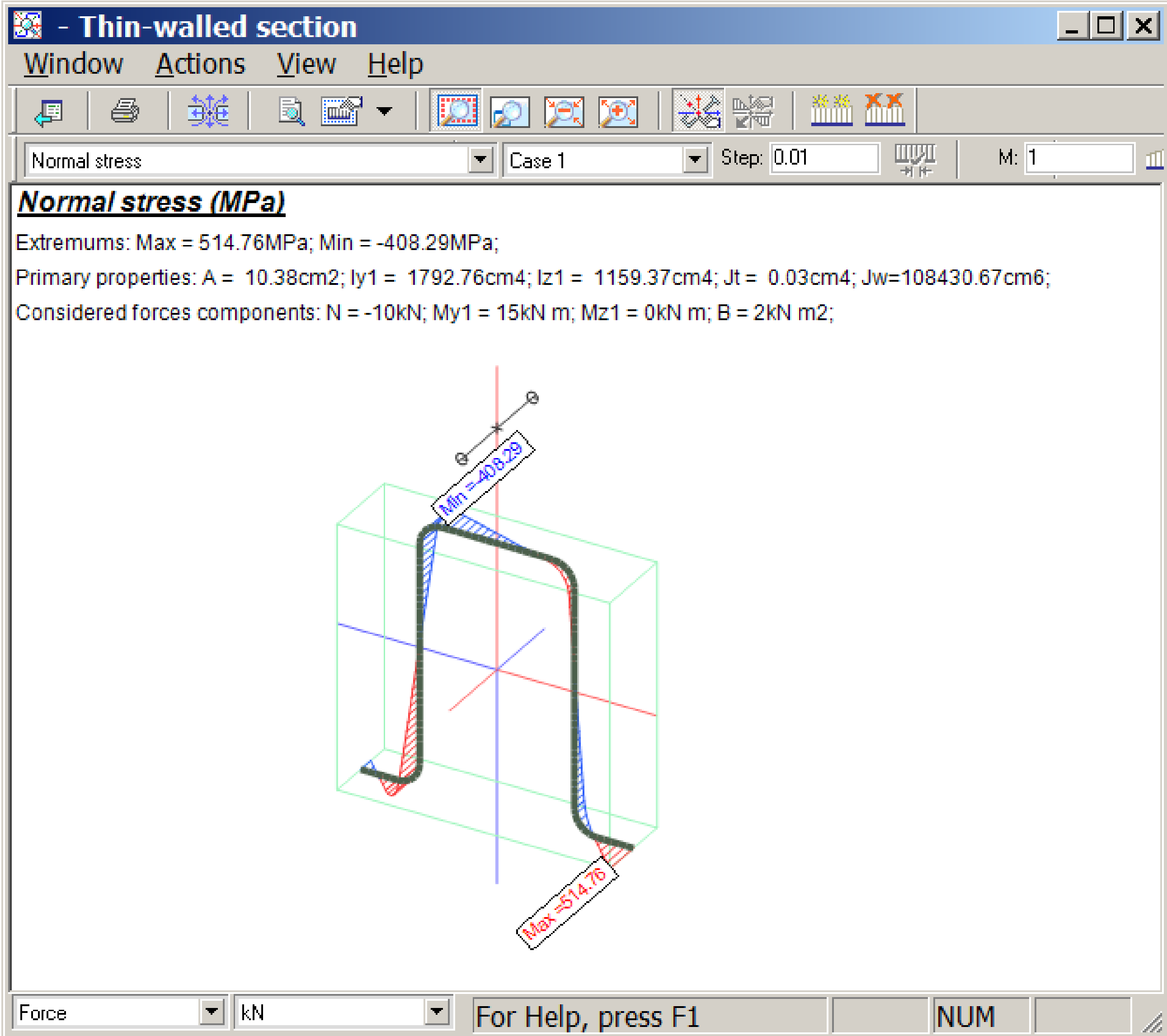
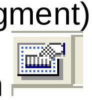


Figure 2.6

To paste the diagrams to report (completely or as a fragment), use the **Add current view to report** and **Select region and add to report** (button  on the toolbar) commands on the **Actions** menu.

To preview all output data for analysis of geometric properties, on the **Actions** menu, click **Show report** command (button  on the toolbar).

There are also additional tools with enhanced options to visualize objects.

Separate sections as single unit

The module enables you to determine location of rigidity centre for separate sections that are considered as single unit. In this case, elements of arbitrary plan of building may be applied.

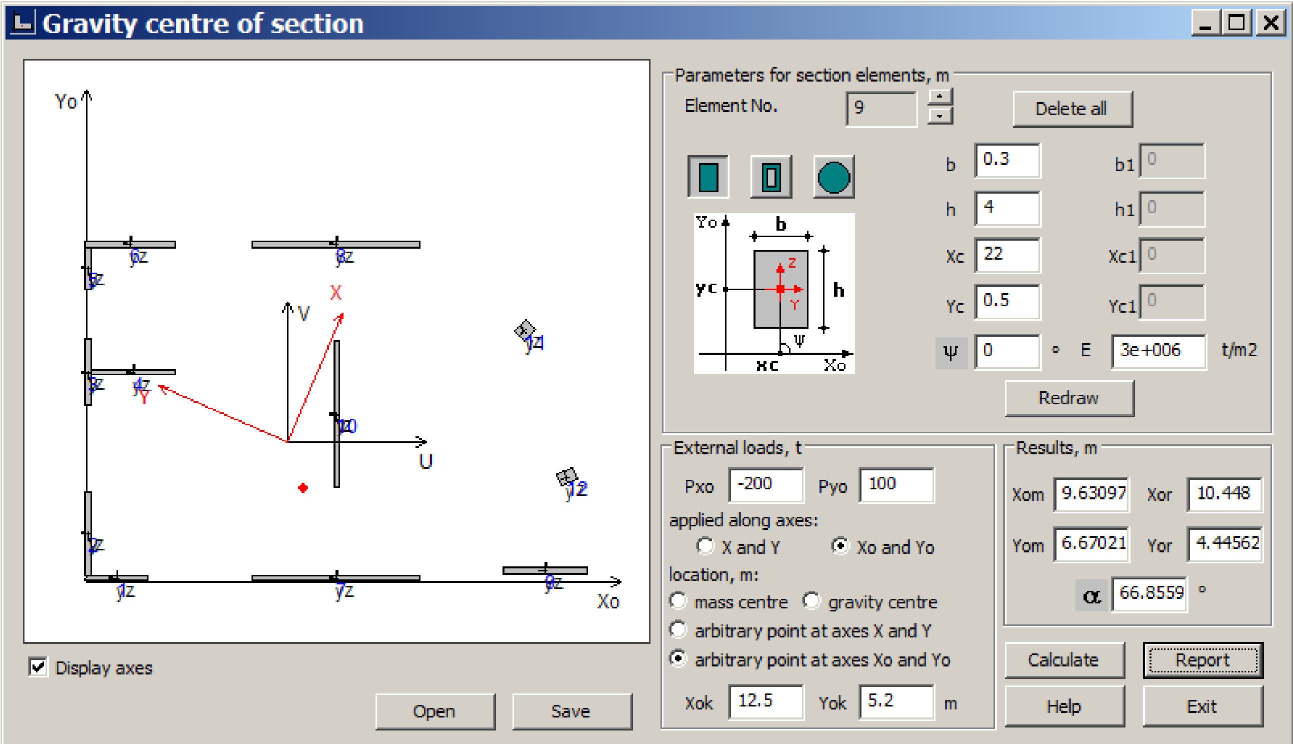


Figure 2.7

Plan of a building (section as a whole unit) consists of elements. Sections of columns (rectangular, circle and box) and walls (rectangular) are considered as elements of building plan.

Coordinates of the centre of rigidity (CR) are calculated based on the assumption that all elements of plan (section as a whole unit) have the same displacement in horizontal plane and the same rotation angle (linearly dependent on displacement) about vertical axis.

This module enables you to determine forces in elements of section from external loads. Loads may be applied at centre of mass (CM), at centre of rigidity (CR) and at the point with arbitrary coordinates.

External loads may be defined in initial or in principal axes. Location of load application may be also defined either in initial or in principal axes.

External loads Px and Py are considered positive if they coincide with direction of principal axes X and Y.

Positive twisting moment MZ caused by external loads is acting anti-clockwise if you look from the end of the Z-axis.

Forces are not calculated in the section that consists of the single element.

 www.liraland.com

Input data.

Every defined element is located at the first quarter of initial coordinate system Xo, Yo.

By default, local system of principal axes of inertia Y, Z is assigned to every element.

The serial number is assigned to every element.

In the appropriate boxes, define the following data:
– modulus of elasticity for material of an element E;
– dimensions of rectangular or box sections **b** and **h**, that are parallel to principal axes of inertia in an element Y,Z respectively; for ring section it is outside and inside diameters respectively;
– web thicknesses for box section **b1** and **h1**;
– distances Xc, Yc from gravity centre of an element to the axes Xo, Yo;
– slope angle Ψ between principal axes of inertia Z of an element and the Xo-axis.

In the appropriate boxes define external loads, coordinates of the point where they are applied and direction of load.

To display elements on schematic presentation area, click **Redraw**.

If the **Display axes** check box is selected, then for every element you will see location of its principal axes of inertia Y, Z.

To make calculation, click **Calculate**.

Output data.

Output data is presented in appropriate boxes as the following values:
– Xom, Yom coordinates of centre of mass (CM) (gravity centre) of a plan in initial axes Xo, Yo;
– Xor, Yor coordinates of centre of rigidity (CR) in the system of axes Xo, Yo;
– slope angle α of principal axes X, Y of a plan.

Location of rigidity centre on the plan is indicated with the red point.

If the **Display axes** check box is selected, then location of centre of mass (CM) is presented as well as principal axes X, Y.

To save the problem to a file, click **Save**.

All above-mentioned data as well as forces in elements of the section from external loads and intermediate calculations are presented in the **Report**.

Below you will se the fragment of the report (table of total forces in elements).

№	QYi	QZi	QXoi	QYoi
1	0.0180413	5.75392	5.75392	-0.0180413
2	0.0818408	-3.70471	0.081841	-3.70471
3	0.0801473	-1.89681	0.0801474	-1.89681
4	0.0021634	1.48421	1.48421	-0.0021634
5	0.0560477	-0.463089	0.0560477	-0.463089
6	0.0275237	25.3616	25.3616	-0.0275237
7	0.0981373	109.111	109.111	-0.0981373
8	0.0261699	43.5522	43.5522	-0.0261699
9	0.0728911	13.7872	13.7872	-0.0728911
10	0.333969	-93.503	0.333973	-93.503
11	0.198507	0.0692695	0.189347	-0.091385
12	0.16148	0.162715	0.208131	-0.0960898
Σ			200	-100

Notation system accepted for the report:

b – dimension of the i-the element along its own Y-axis;
h – dimension of the i-th element along its own Z-axis;
Xc, Yc – coordinates of the gravity centre of the i-th element in initial axes Xo, Yo;
F – area of the i-th element.

Xom, Yom – coordinates of gravity centre of a plan in initial axes Xo, Yo.

Jy – moment of inertia of the i-th element relative to its own principal Y-axis;
Jz– moment of inertia of the i-th element relative to its own principal Z-axis.
ψ – angle between axes Z and Xo for the i-th element; measured anti-clockwise from the Xo-axis to the Z-axis.

Axes U, V – central (not principal) axes that are parallel to the axes Xo, Yo respectively.

Ui = Xoi– Xom; Vi= Yoi – Yom – coordinates of gravity centre of the i-th element in U, V axes .

Jui – moment of inertia of the i-th element relative to the U-axis;
Jvi – moment of inertia of the i-th element relative to the V-axis;
Juvi – product of inertia of the i-th element in axes U, V.

ui, vi – coordinates of gravity centre of the i-th element in axes U, V.

$Ju = \Sigma(Jui + Fi^* ui^2)$;
$Jv = \Sigma(Jvi + Fi^* vi^2)$;
$Juv = \Sigma(Juvi + Fi * ui *vi)$.

Ju – total moment of inertia relative to the U-axis;

Jv – total moment of inertia relative to the Vaxis;
Juv – total product of inertia of the i-th element in axes U, V.

$K_1 = (Ju + Jv)/2;$
$K_2 = (Ju - Jv)/2;$
$K_3 = \sqrt{(K_2^2 + Juv^2)}.$
$Jx = Jmax = K_1 + K_3;$
$Jy = Jmin = K_1 - K_3.$

Jx, Jy – principal moments of inertia of a plan.

$\alpha = arctg((Juv / (Jmin – Ju))$.
α – rotation angle of principal central axes X, Y of a plan; measured anti-clockwise from the Xo-axis to the X-axis (angle between axes Xo and X).

Ai – moment of inertia of the i-th element relative to the principal X-axis;
Bi – moment of inertia of the i-th element relative to the principal Y-axis;
Di – product of inertia of the i-th element in principal axes.

Numerical parameters:
$A = \sum Ai;$ $B = \sum Bi;$ $D = \sum Di;$
$Ay = \sum(Ai * Yi);$ $Bx = \sum(Bi * Xi);$ $Dx = \sum(Di * Xi);$ $Dy = \sum(Di * Yi);$
$E = A*B – D^2;$ $E_1 = Dy – Bx;$ $E_2 = Dx – Ay.$

Coordinates of rigidity centre (CR) in principal axes X, Y:

$Xr = – (E_1*A+E_2*D)/E;$
$Yr = – (E_2*B+E_1*D)/E.$

Coordinates of rigidity centre (CR) in initial axes Xo, Yo:

$Xor = Xom + Xr * \cos(\alpha) – Yr * \sin(\alpha);$
$Yor = Yom + Xr * \sin(\alpha) + Yr * \cos(\alpha).$

Algorithm for calculation of forces in elements

1. Calculation of displacements and forces in elements of a plan from external loads Px and Py applied at the centre of mass (CM) along the direction (positive) of principal axes X and Y.

Displacements in principal axes:
$\Delta X = (- Px * B + Py * D) / E;$
$\Delta У = (Px * D - Py * A) / E.$

Displacements in initial axes:
$\Delta Xo = \Delta X * \cos(\alpha) - \Delta Y * \sin(\alpha);$
$\Delta Yo = \Delta X * \sin(\alpha) + \Delta Y * \cos(\alpha).$

Let's define: Si, Ci – sin(ψi-α) and cos(ψi-α) respectively.

Displacements in local axes of the i–th element:
ΔУi= ΔX * Si- ΔУ * Ci;
ΔZi = ΔX * Ci+ ΔУ * Si.

Forces in local axes of the i–th element:
Qyi(P) = Jzi* ΔУi;
Qzi(P) = Jyi* ΔZi.

Positive moment MZ acts ***anti-clockwise***.
Moment MZ = MZx +MZy , where MZx = Px * bи MZy= –Py * a.

More notations:

χ_i = Xi – a; γ_i= Уi– b.

$A_\beta = \Sigma(Ai * \gamma_i^{\ 2})$; $B_\beta = \Sigma(Bi * \chi_i^{\ 2})$; $D_\beta = \Sigma(Di * \chi_i * \gamma_i)$;

$Jt = A_\beta – 2*D_\beta + B_\beta$; β = Mz / Jt;

Displacements of the i–th element in local axes caused by moment MZ:

ΔYi (M) = β* (χ_i * Ci + γ_i * Si);
ΔZi (M) = β* (-χ_i * Si + γ_i * Ci).

Forces in local axes of the i–th element from moment MZ:

Qyi(M) = Jzi * ΔYi (M);
Qzi(M) = Jyi * ΔZi (M).

Total forces in local axes of the i–th element:

QYi = QYi(P) + QYi(MZ);
QZi = QZi(P) + QZi(MZ).

Forces in the i–th element in initial axes Xo, Yo:

QXoi = QYi * sin(Ψi) + QZi * cos(Ψi);
QУoi = -QYi * cos(Ψi) + QZi * sin(Ψi).

2. If external loads Pxo and Pyo applied at the centre of mass (CM) are oriented relative to initial axes Xo and Yo, then they are transformed into loads Px and Py (relative to the principal axes):

Px = Pxo * cos(α) + Pyo * sin(α);
Py = –Pxo * sin(α) + Pyo * cos(α).

Moment MZ = MZx +MZy , where MZx = Px * b and MZy= –Py * a.

If external loads Pxo and Pyo applied at the centre of rigidity (CR) are oriented relative to initial axes Xo and Yo, then they are transformed into loads Px and Py (relative to the principal axes):

Px=   Pxo * cos(α) + Pyo* sin(α);
Py= –Pxo* sin(α) + Pyo* cos(α).

In this case, there is no moment (MZ=0).

3. If external loads Pxk and Pyk are applied at arbitrary point K of a plan and oriented relative to the principal axes X and Y, then the moment will appear.

In this case, external moment MZ may be calculated by formula:

MZ = – Px*(Yк – Yr) + Py*(Xк – Xr).

4. If external loads Pxok and Pyok are applied at arbitrary point K with coordinates Xok, Yok in initial axes Xo and Yo, then coordinates of point K should be recalculated into principal axes X and Y, while loads are transformed into loads Px and Py (relative to principal axes):

Xк =   (Xoк – Xoм) * cos(α) + (Yoк –Yoм) * sin(α);
Yк = – (Xoк – Xoм) * sin(α) + (Yoк –Yoм) * cos(α).

Px=   Pxoк* cos(α) + Pyoк* sin(α);
Py= –Pxoк * sin(α) + Pyoк * cos(α).

In this case, external moment MZ may be calculated by formula:

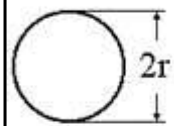
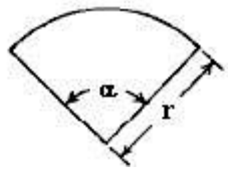
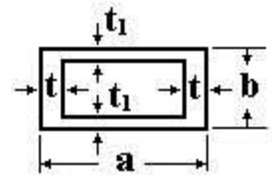
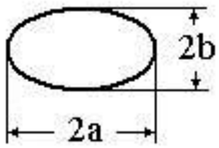
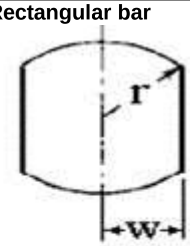
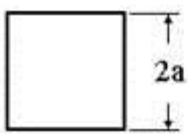
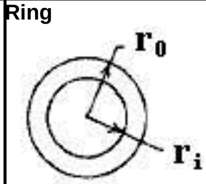
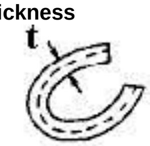
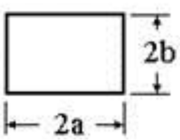
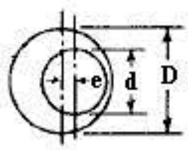
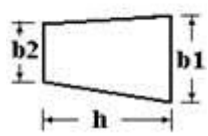
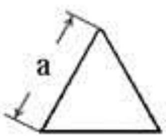
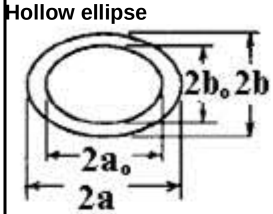
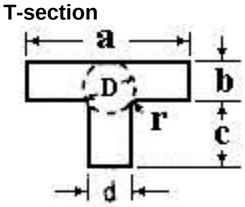
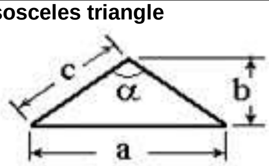
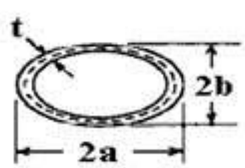
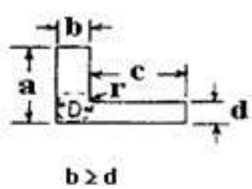
MZ = – Px*(Yк – Yr) + Py*(Xк – Xr).

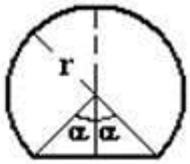
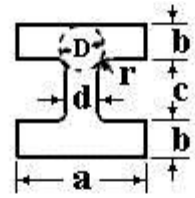
Forces are considered positive if they coincide with direction of appropriate axes.

Moment of inertia in torsion

The module enables you to calculate moments of inertia in free torsion for the most widely used section types. The calculation is made according to formulas from [1, 2]. Section moduli are also calculated for several types of sections.

Realized types of sections are presented in the table below.

No.	Shape of section	No.	Shape of section	No.	Shape of section
1	**Circle** 	8	**Sector** 	15	**Box** 
2	**Ellipse** 	9	**Rectangular bar**  **with 2 or 4 parallel cuts**	16	**Thin-walled open ring of uniform thickness** 
3	**Square** 	10	**Ring** 	17	**Arbitrary convex open section of uniform thickness** 
4	**Rectangle** 	11	**Off-centre ring** 	18	**Trapezium** 
5	**Equilateral triangle** 	12	**Hollow ellipse** 	19	**T-section** 
6	**Isosceles triangle** 	13	**Thin-walled hollow ellipse** 	20	**Angle section** 

7	Segment	14	Arbitrary convex hollow section of uniform thickness	21	I-section
					

Input data.

Define necessary dimensions for the section in appropriate boxes and click **Calculate**.

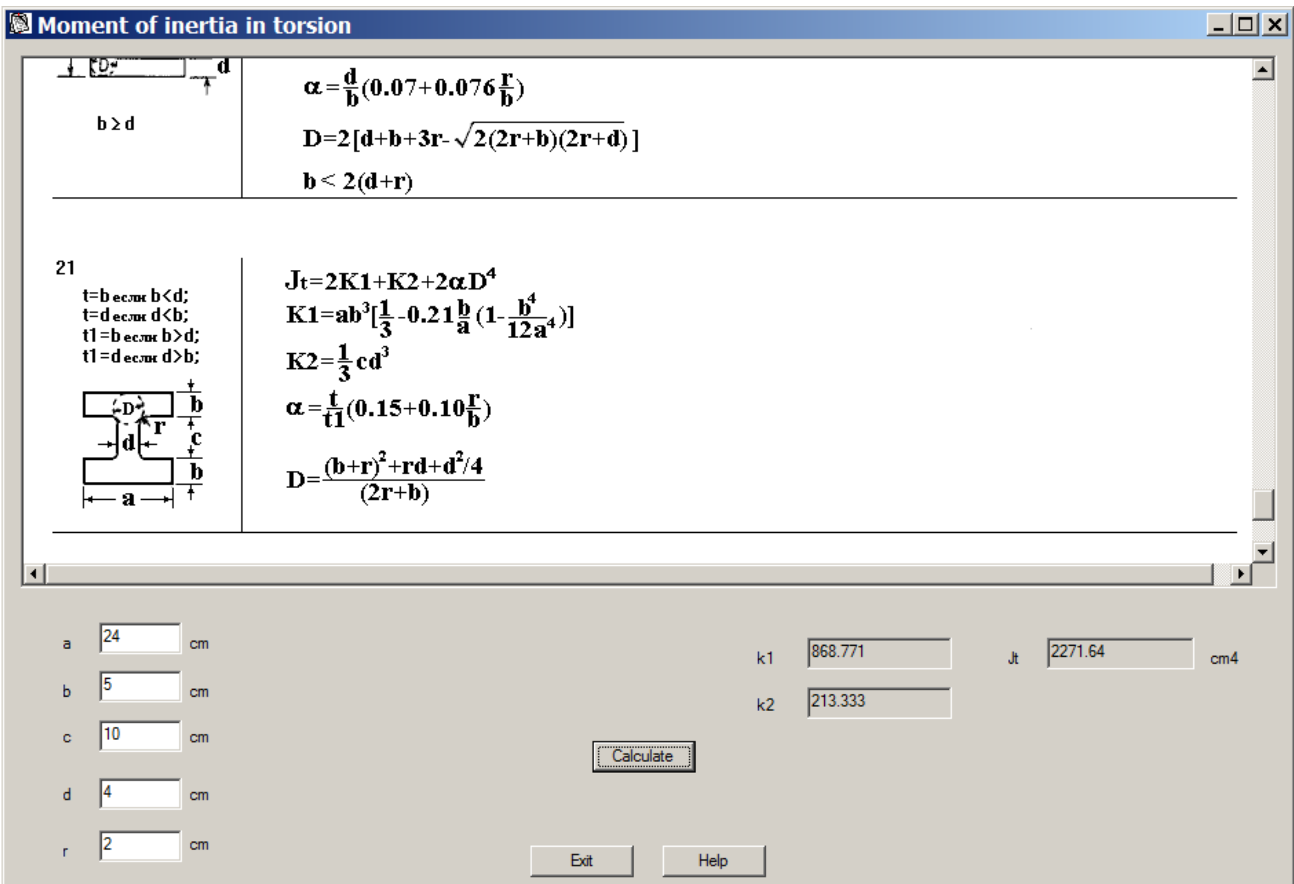


Figure 2.8

Output data.

In the appropriate boxes you will see calculated moments of inertia in torsion **Jt**, section moduli **Wt** and some additional parameters.

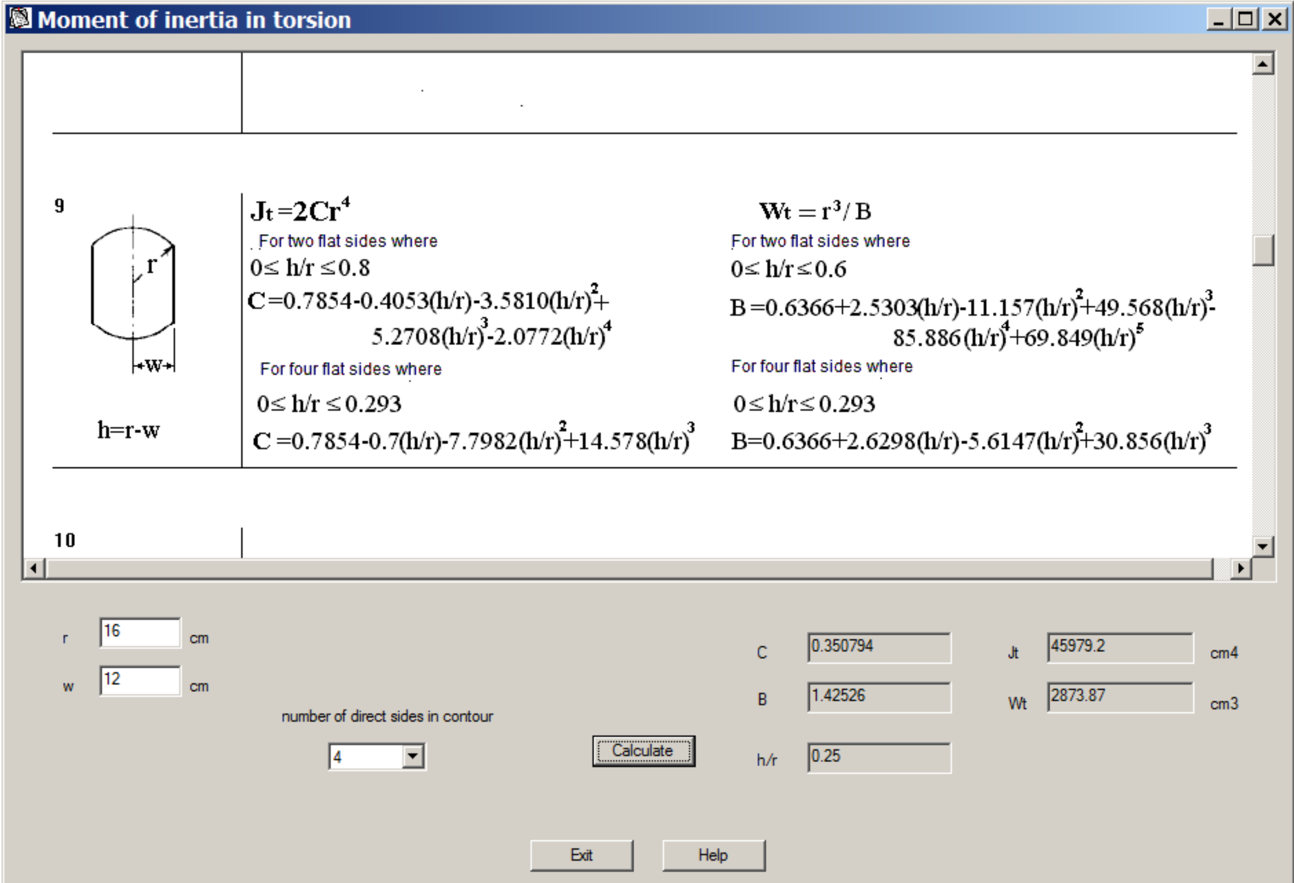

Moment of inertia in torsion
9
r
w
h=r-w
Jt =2Cr4
For two flat sides where
0≤ h/r ≤0.8
C=0.7854-0.4053(h/r)-3.5810(h/r)2+
5.2708(h/r)3-2.0772(h/r)4
For four flat sides where
0≤ h/r ≤0.293
C =0.7854-0.7(h/r)-7.7982(h/r)2+14.578(h/r)3
Wt = r3/ B
For two flat sides where
0≤ h/r≤0.6
B =0.6366+2.5303(h/r)-11.157(h/r)2+49.568(h/r)3-
85.886(h/r)4+69.849(h/r)5
For four flat sides where
0≤ h/r≤0.293
B=0.6366+2.6298(h/r)-5.6147(h/r)2+30.856(h/r)3
10
r
16
cm
w
12
cm
number of direct sides in contour
4
Calculate
C
0.350794
B
1.42526
h/r
0.25
Jt
45979.2
cm4
Wt
2873.87
cm3
Exit
Help

Figure 2.9

Static/Dynamic analysis, stability

Static and dynamic analyses, stability

This chapter contains modules for analysis of the most frequent problems in statics, dynamics and stability of structures.

Continuous beam
Influence lines in continuous beam
Truss
Parametric plane frame
Arbitrary plane frame
Rectangular slab on elastic foundation
Rectangular slab
Wall–beam
Shell on rectangular plan
Shell on circular plan
Mode shapes and frequencies of natural vibrations in cantilever
Stability factors and buckling modes in cantilever
Mode shapes and frequencies of natural vibrations in continuous beam
Cable and string

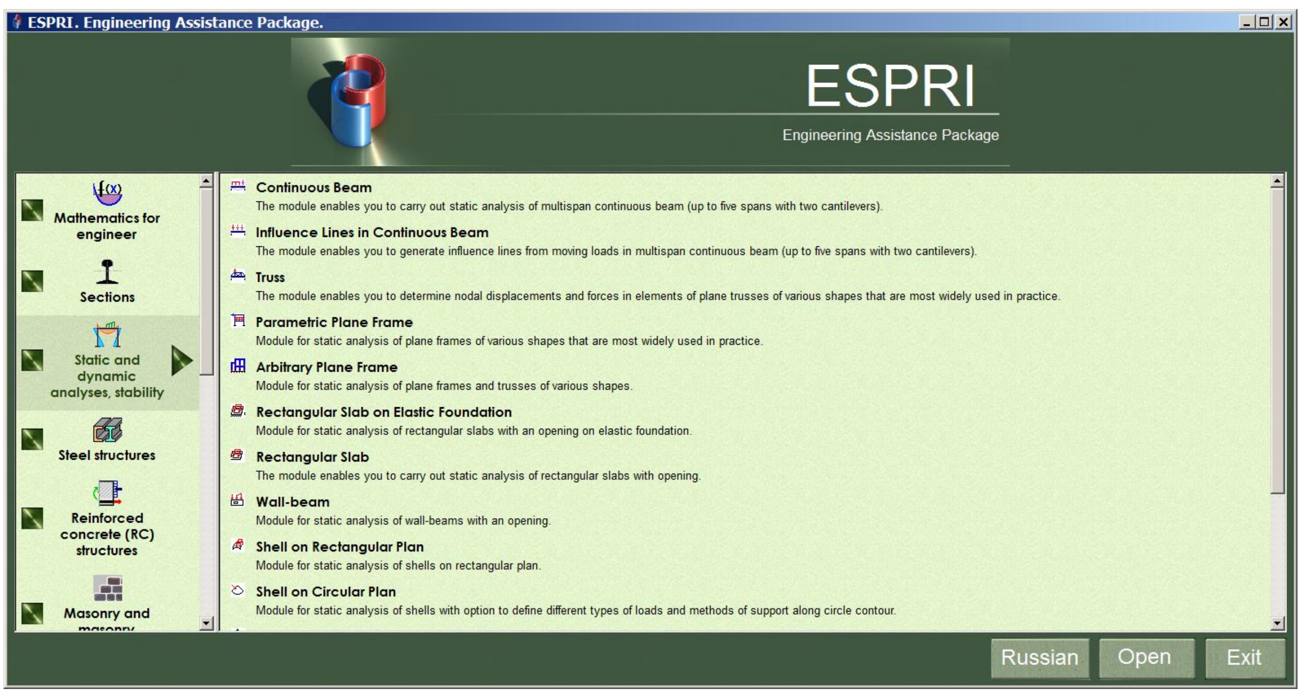
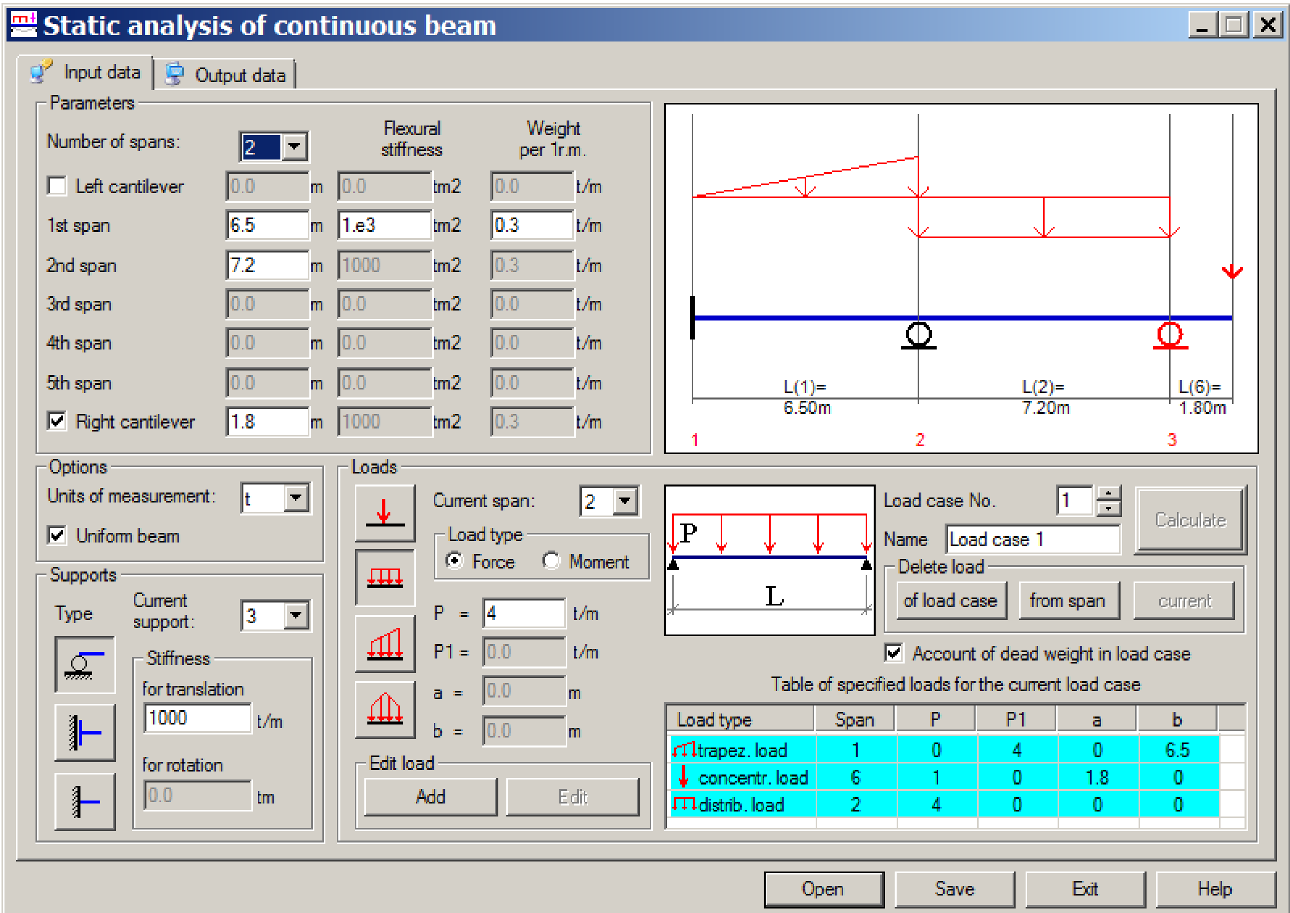


Figure 3.1

Continuous beam

The module enables you to carry out static analysis of multispan continuous beam (up to five spans with two cantilevers).

Input data.



Load type	Span	P	P1	a	b
trapez. load	1	0	4	0	6.5
concentr. load	6	1	0	1.8	0
distrib. load	2	4	0	0	0

Figure 3.2

– Select number of spans in the appropriate list box and, if necessary, define left and right cantilever (to do this, just select appropriate check boxes).
– Select measurement units for load and stiffness - **t** or **kN**.
– Define length of span (cantilever) in **m** and flexural stiffness (if all spans and cantilevers have the same flexural stiffness, select the **Uniform beam** check box).
– Select the support for which it is necessary to apply boundary conditions. You could select the support in the **Current support** list box or by clicking directly on schematic presentation.
– Select the type of boundary condition with the **Type** buttons (the type will be displayed on schematic presentation) and, if required, in the **Stiffness** area, define the stiffness values for translation and rotation.
– Select number for the current load case (up to 3 load cases).

– Select spans where the load should be applied to. You could select the span in the **Current span** list box or by clicking directly on schematic presentation (it will be displayed there).
– Define the type of load with one of the 4 buttons, then define the load type (**force** or **moment**).
– Input load value (**P, P1**), distance to the beginning/end of the load (**a, b**) (if required) and click **Add**.
– To modify parameters of load, select certain load with the pointer in the table of specified loads, modify necessary parameters and click **Edit**.
– To delete load, select certain load with the pointer in the table of specified loads. In the **Delete load** area, click **current**. To delete all loads from the current span, click **from span**.
– To add dead weight to the current load case, select the **Account of dead weight in load case** check box.
– When you define input data, click **Calculate**.
– If required, it is possible to save the problem.

Output data.

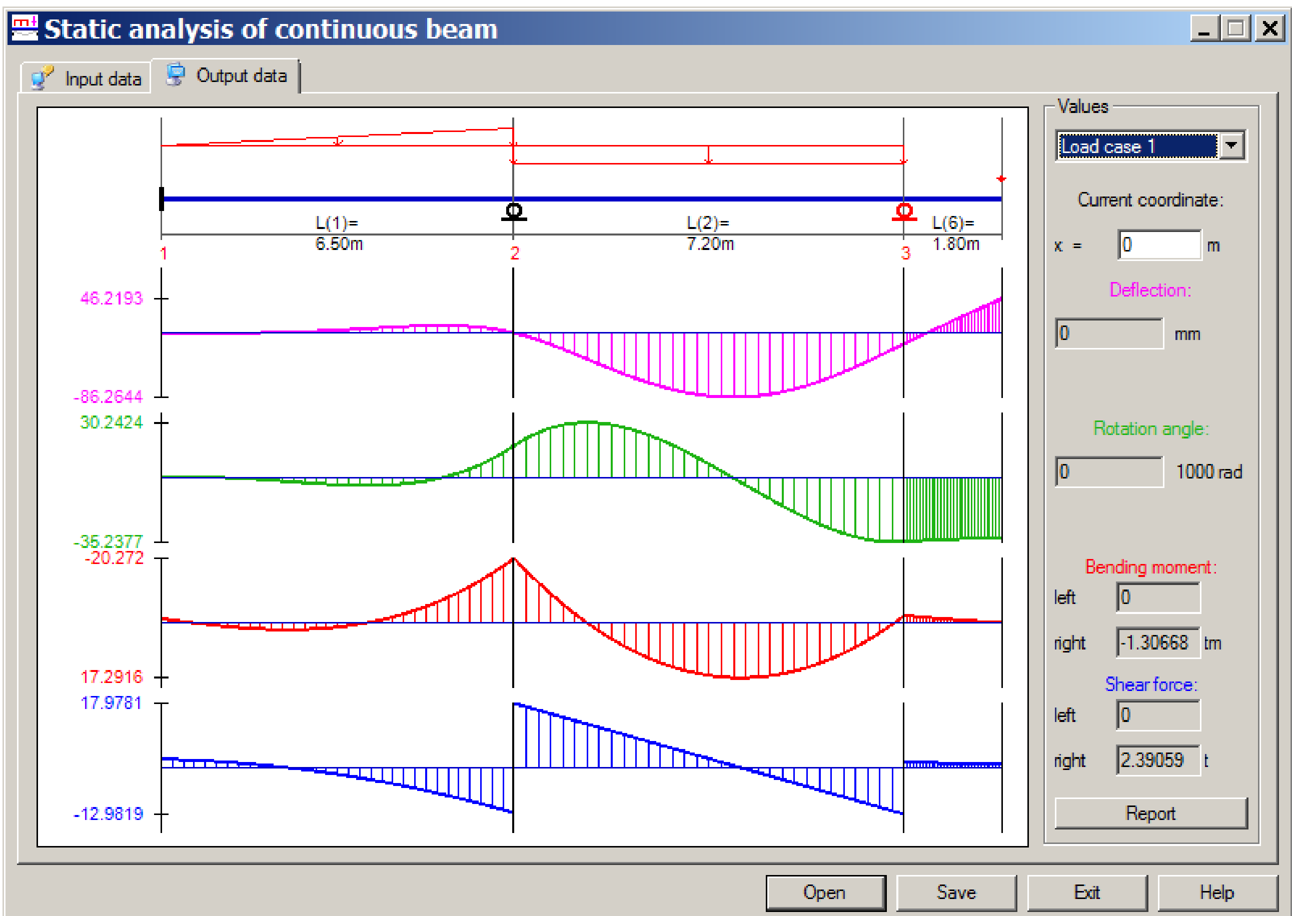


Figure 3.3

On the **Output data** tab you will see the diagrams of displacements, rotation angles, bending moments and shear forces. To find out ordinates of diagrams at any point of beam, drag the pointer across the diagram and view the ordinate at appropriate box.

Colour of the diagram corresponds to the colour of diagram name at the right part of the dialog box.

To present a report document in HTML format, click **Report**.

Truss

This module enables you to determine nodal displacements and forces in elements of plane trusses of various shapes that are most widely used in practice.

Input data.

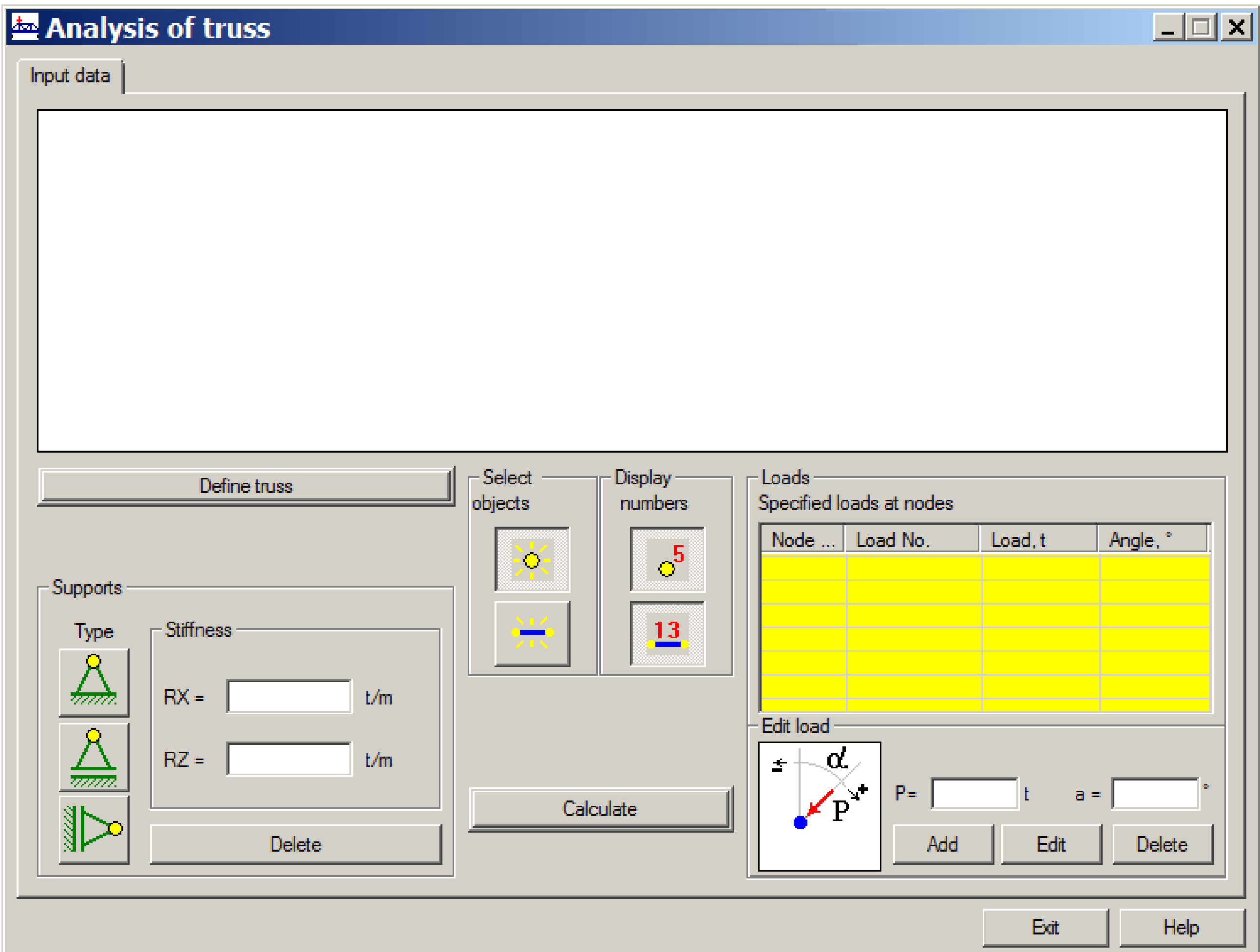


Figure 3.4

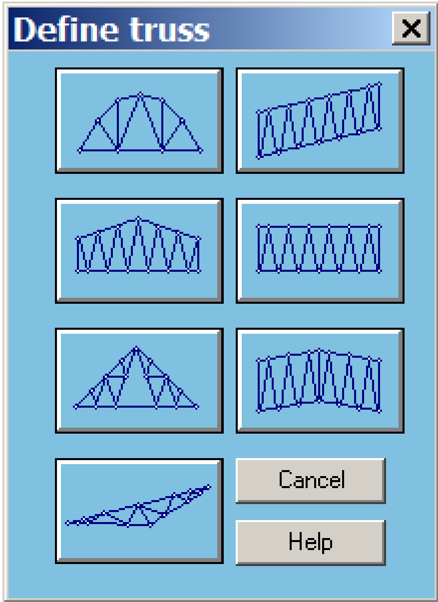


Figure 3.5

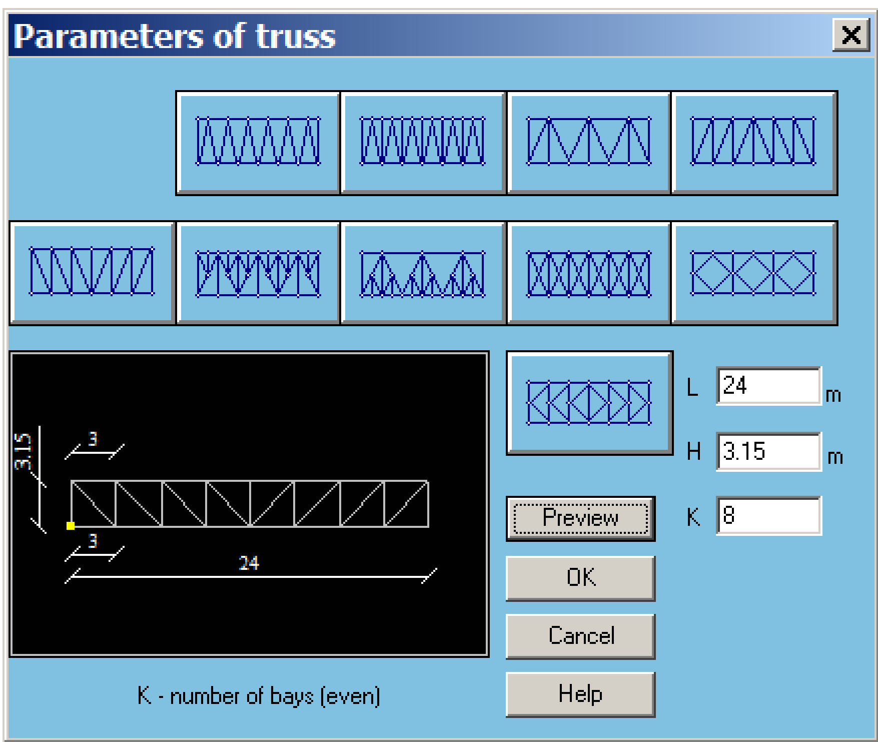


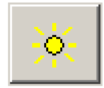
Figure 3.6

To define input data, follow these steps:

– Click the **Define truss** button.

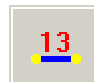
In the **Define truss** dialog box, select the truss by shape of the chord. To do this, press the button with the corresponding icon:

In the **Parameters of truss** dialog box, select the web of truss. Then define appropriate parameters (see the Tip at the bottom of the dialog box). To check whether parameters are defined correctly, click **Preview**. Then click **OK**.

– In the **Select objects** area, click the **Nodes** button  and select necessary nodes with the pointer on schematic presentation. In the **Supports** area, define boundary conditions (just click appropriate icon). If required, define stiffness of restraints.

– To delete restraints at any node, select this node with a pointer and click the **Delete** button in the **Supports** area.

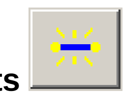
– To display numbers of nodes and elements on schematic presentation, in the **Display numbers** area, click **Nodes** 

or **Elements** 

.

– To define load at node, select appropriate node with the pointer, then in the **Edit load** area, define the load value (**P**) and angle of load application (**α**). Click **Add**.

– To modify or delete nodal load, click appropriate load in the **Specified loads at nodes** table. Modify values of (**P**) and (**α**), click **Edit** or click **Delete**.

– To assign stiffness to elements, in the **Select objects** area, click **Elements**  . Select appropriate elements either on schematic presentation or in the table of elements, define **EA** value and click **Apply**.

– Click **Calculate**.

Output data.

On the **Output data** tab you will see deformed shape, tables of forces in bars and nodal displacements. When you select node or element on schematic presentation, the row that corresponds to this node or element in appropriate table will be selected in the list.

To present a report document in HTML format, click **Report**.

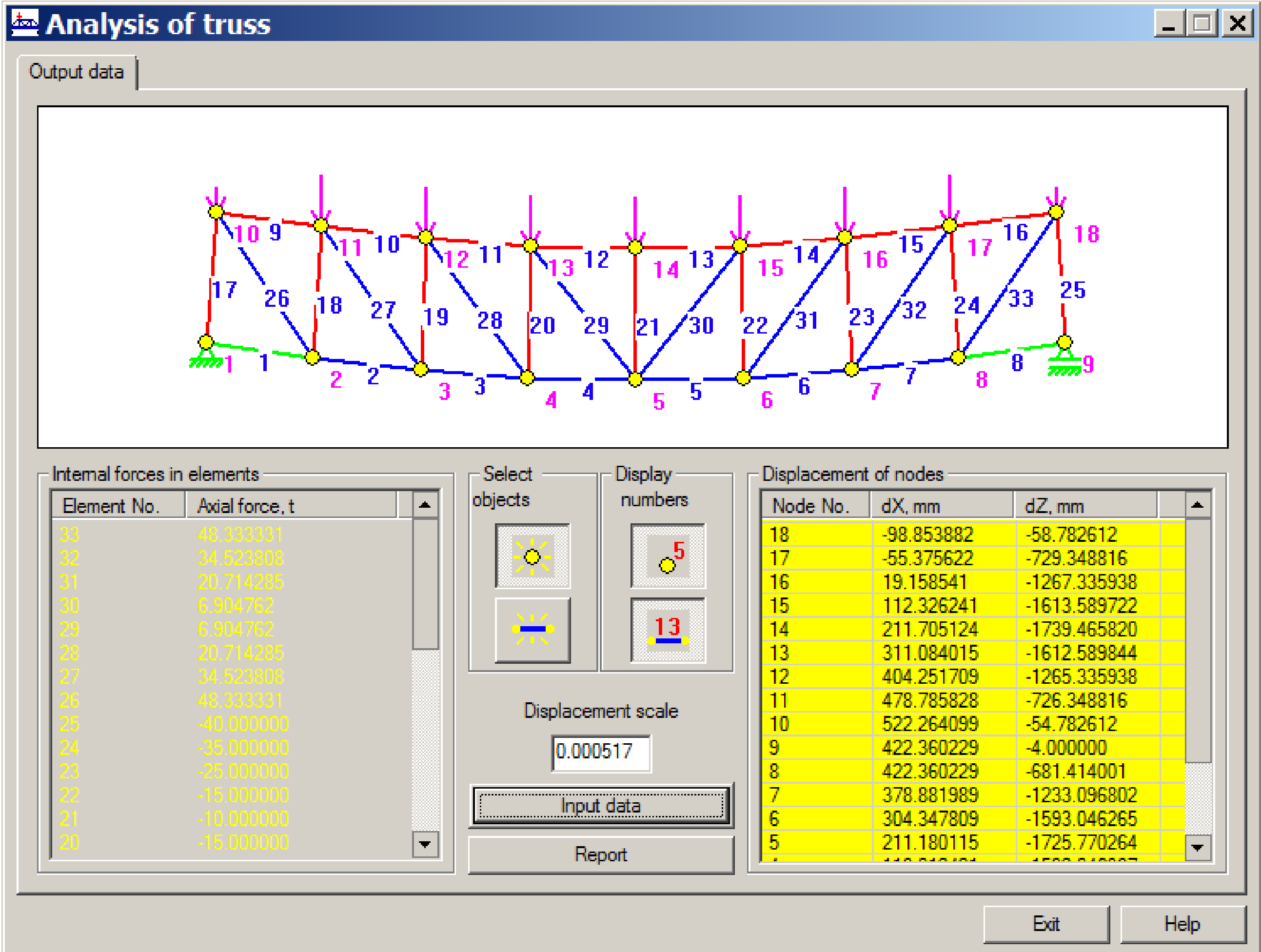


Figure 3.7

Parametric plane frame

The module enables you to carry out static analysis of plane frames of various shapes that are most widely used in practice.

Input data.

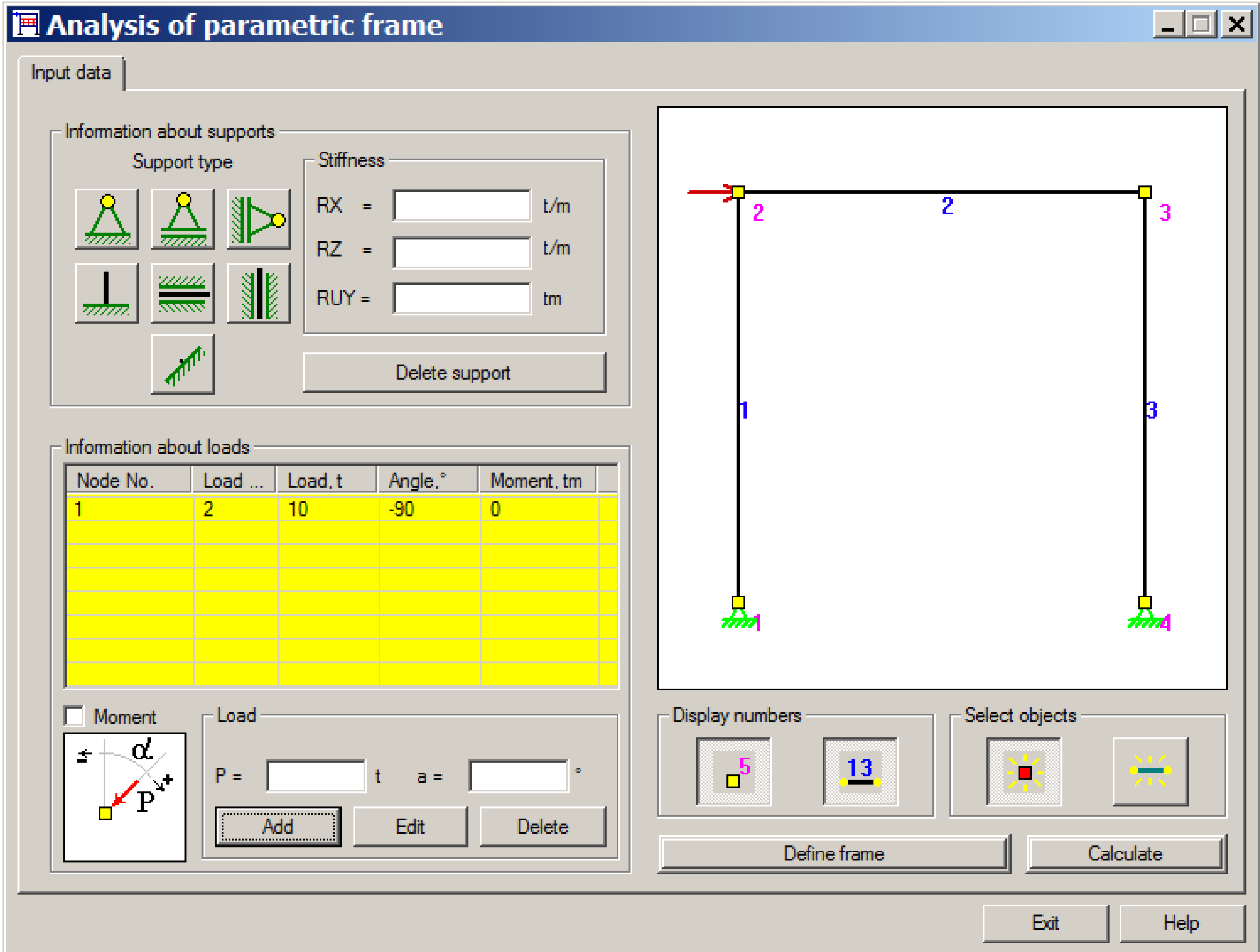

Analysis of parametric frame
Input data
Information about supports
Support type
Stiffness
RX =
t/m
RZ =
t/m
RUY =
tm
Delete support
Information about loads
Node No.
Load ...
Load, t
Angle, °
Moment, tm
1
2
10
-90
0
Moment
Load
P =
t
a =
°
Add
Edit
Delete
Display numbers
Select objects
Define frame
Calculate
Exit
Help

Figure 3.8

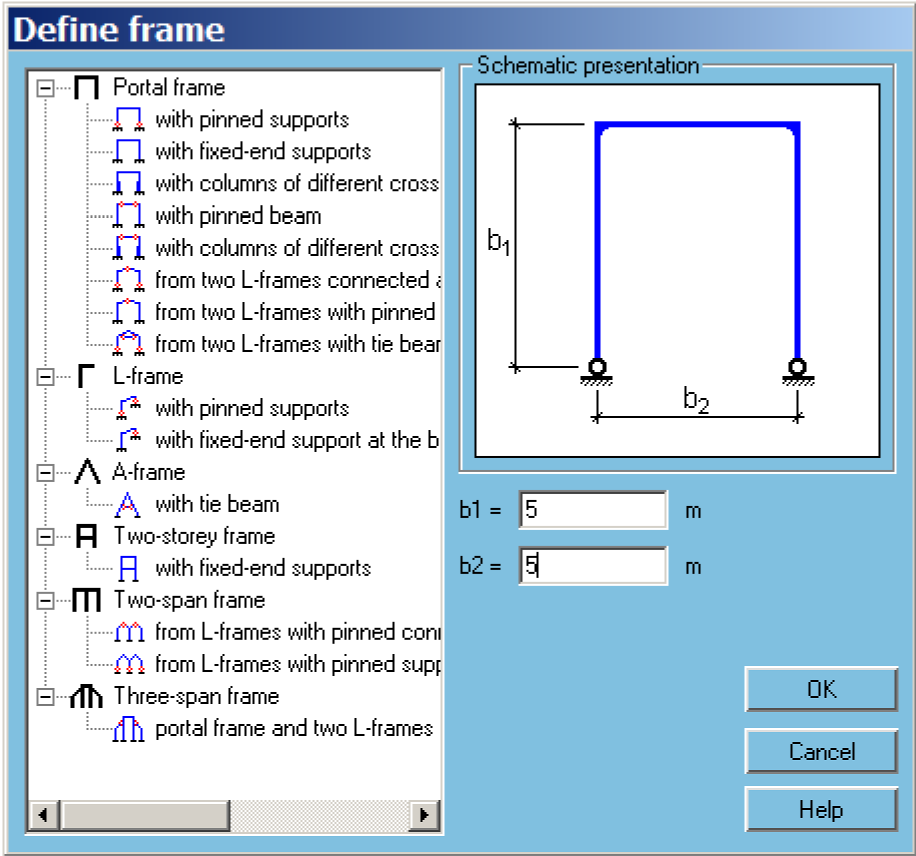
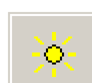


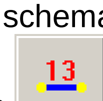
Figure 3.9

To define input data, follow these steps:

– Click the **Define frame** button. In the **Define frame** dialog box, select necessary frame from the list of main types of parametric frames. Define geometric properties of selected frame and then click **OK**.

– In the **Select objects** area, click the **Nodes** button  and select necessary nodes with the pointer on schematic presentation. In the **Information about supports** area, define boundary conditions (just click appropriate icon). If required, define stiffness of restraints.

– To delete restraints at any node, select this node with a pointer and click the **Delete support** button in the **Information about supports** area.
– To display numbers of nodes and elements on schematic presentation, in the **Display numbers** area, click **Nodes** 

or **Elements** 

.

– To define load at node, select appropriate node with the pointer, then in the **Load** area, define the load value (**P**) and angle of load application (**α**). Click **Add**.

– To modify or delete nodal load, click appropriate load in the table with specified loads at nodes. Modify values of (**P**) and (**α**), click **Edit** or click **Delete**.

– To define load on element, in the **Select objects** area, click the **Elements** button and select necessary elements either with the pointer on schematic presentation or in the **Information about stiffness** table. Then select appropriate type of load (just click appropriate icon), in another dialog box define parameters of load and click **OK**.

– If it is required to define additional hinges in bars of frames, select appropriate element and click **Define hinges** button . In the **Define hinges** dialog box, select define where to place hinges and click **OK**.

– To assign stiffness to elements, in the **Select objects** area, click **Elements** . Select appropriate elements either on schematic presentation or in the **Information about stiffness** table. In the **Stiffness** area, define axial and flexural stiffness values (EA and EI), then click **Apply**.

– Click **Calculate**.

Output data.

On the **Output data** tab you will see deformed shape, tables of forces in bars and nodal displacements. When you select node or element on schematic presentation, the row that corresponds to this node or element in appropriate table will be selected in the list.

To display force diagrams in elements, click appropriate buttons. If required, it is also possible to display applied loads together with diagrams.

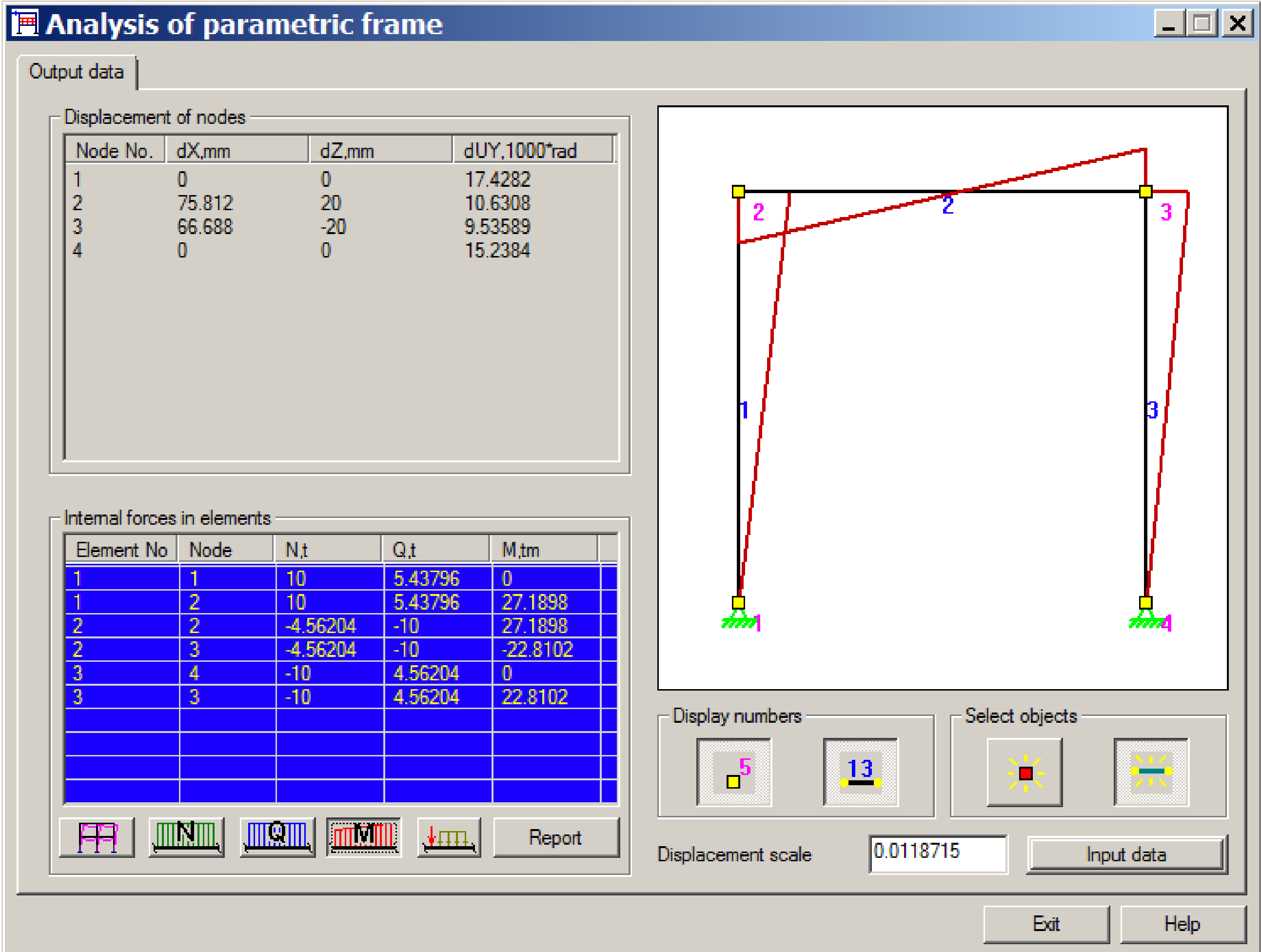


Figure 3.10

To present a report document in HTML format, click **Report**.

Arbitrary plane frame
Arbitrary plane frame

This module enables you to carry out static analysis of plane frames and trusses of various shapes.

Model of the frame is located in the XOZ-plane. X-axis is horizontal while Z-axis is vertical. Y-axis is directed out of the plane and generates the right Cartesian coordinate system with the X and Z-axes. Let us refer this coordinate system as global or principal one.

Every node of the frame has three degrees of freedom (DOF) – two translations (X, Z) and rotation (UY).

Nodal loads – forces and moments are directed relative to the global coordinate system. Force is considered to be positive if it is directed opposite appropriate axis. Moment is considered to be positive if it is directed clockwise when you look from the end of appropriate axis.

Initial translation is considered to be positive if it acts along direction of appropriate axis.

Initial rotation is considered to be positive if it is directed anti-clockwise when you look from the end of appropriate axis.

Restraints imposed at support nodes are directed relative to the global coordinate system.

Local coordinate system of every bar is also right Cartesian coordinate system. X1-axis is longitudinal axis of the bar; it passes from the beginning of the bar up to its end through the gravity centre of the section. As a rule, node of the model with the smaller number is considered as beginning of the bar while node of the model with the greater number is considered as the end of the bar.

Y1-axis and Z1-axis – principal central axes that also pass through the gravity centre.

Local axes are generated according to the rules below.

For vertical bars:
– if the X1-axis is directed upward, then the Y1-axis is horizontal and directed out of the plane at us and the Z1-axis is directed left-to-right;
– if the X1-axis is directed downward, then the Y1-axis is horizontal and directed out of the plane at us and Z1-axis is directed right-to-left.

For arbitrary oriented bars:
– the Z1-axis is always directed to the upper half-space;
– if the X1-axis is directed left-to-right, then the Y1-axis is horizontal and directed out of the plane from us;
– if the X1-axis is directed right-to-left, then the Y1-axis is horizontal and directed out of the plane at us.
Local load is oriented relative to local coordinate axes. Sign convention for the local load is similar to the sign convention for the nodal load.

The work with the program will be illustrated at example of two-storey frame with two spans.

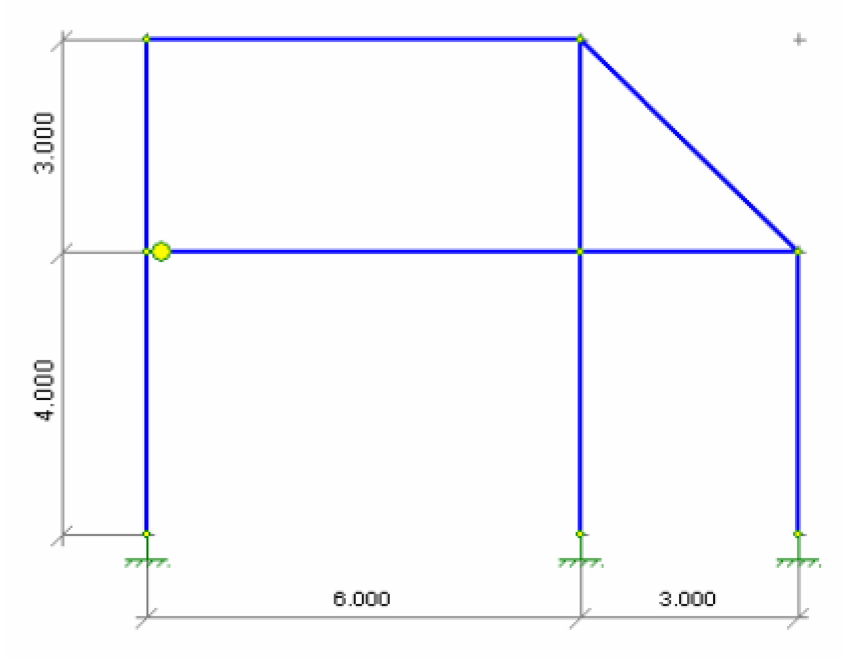


Figure 3.11

[Step 1. Generate model geometry]

[Step 2. Define boundary conditions (restraints) and hinges]

[Step 3. Define cross-sections for bars]

[Step 4. Define loads on design model]

[Options to edit and visualize the model]

[Output data]

[Sign conventions for displacements and forces]

Step 1. Generate model geometry

To start, on the MODEL menu, click **Grid** (button  on the toolbar). In the **Grid generation** dialog box, define steps along axis 1 (horizontal axis) 6 and 3 metres and steps along axis 2 (vertical axis) 4 and 3 metres. Number of steps for all cases define as equal to 1.

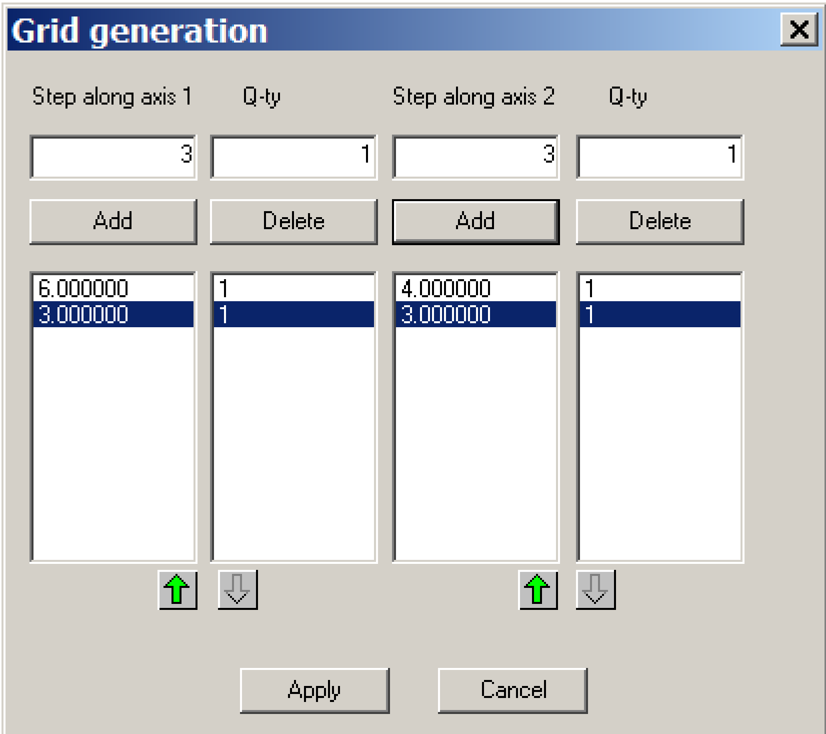
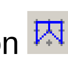


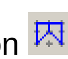
Figure 3.12

Click **Apply**. Nodes of the grid will be displayed on the screen. They are denoted with grey cross symbols.

To define bars, on the MODEL menu, click **Bars** (button  on the toolbar) and connect nodes of the grid.

When you define bars, the mouse buttons work in the following way.

Left mouse button: the first click – to select the first node, the second click – to select the second node. Right mouse button: to unselect the first node.

When model geometry is generated, click the **Bars** button  once again in order to make this command (mode) not active.

Step 2. Define boundary conditions (restraints) and hinges

To define the type of restraint, on the MODEL menu, point to **Add restraint** and click appropriate command from the list or click necessary button on the toolbar.

Define the nodes where the restraint should be imposed on.

In this example, select **Restrain all** option (button on the toolbar) and define three bottom nodes.

Then click this button once again in order to make this command (mode) not active.

To define hinges (connections between bars and nodes), on the MODEL menu, click **Add hinge** (button on the toolbar). Then click appropriate end of the bar. If there are hinges at both ends of the bar, place the pointer (and click) at the middle of the bar.

The hinges will be marked with yellow circle on the screen.

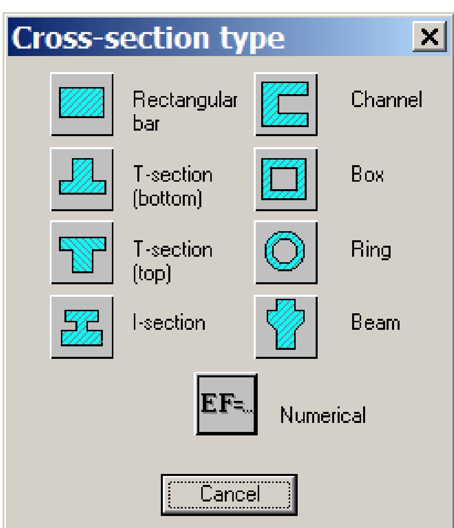
Click **Add hinge** button once again in order to make this command not active.

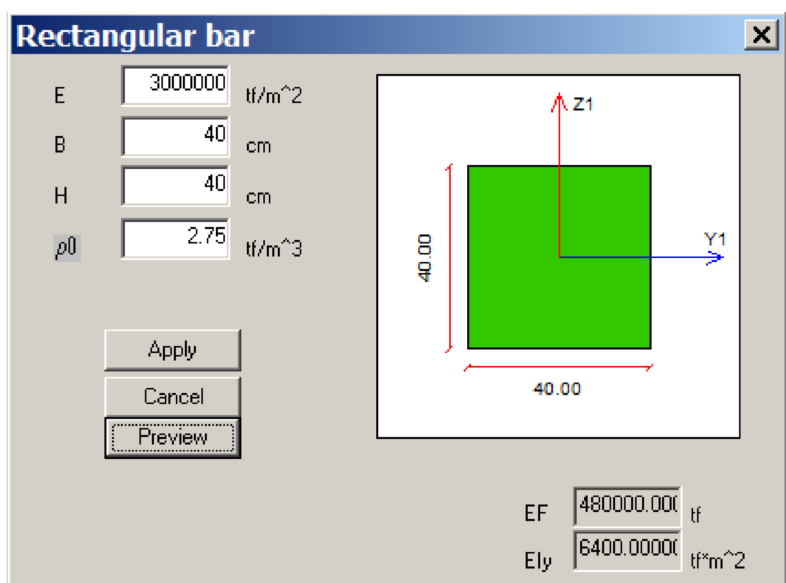
Step 3. Define cross-sections for bars

To define sections for bars, on the MODEL menu, point to **Section** and click **Select section** (button on the toolbar).

In the **Cross-section** dialog box, click **Add**.

In the **Cross-section type** dialog box, select **Rectangular bar** option and define necessary properties: rectangular bar with dimensions 40x40cm, modulus of elasticity 3000000 t/m^2 and material density 2.75t/m^3. Click **Apply**. Other types of sections are defined in the same way.





a) b)

Figure 3.13

In the **Cross-section** dialog box, select the section and click **Set as current**. Then click **Apply**.

To assign current section to bars of design model, follow one of these methods:

1) on the EDIT menu, click **Select bars** (button on the toolbar).
Select necessary bars with the pointer.

Then on the MODEL menu, point to **Section** and click **Assign section** (button on the toolbar).

On the MODEL menu, point to **Section** and click **Apply to selected bars** (button on the toolbar);

2) On the MODEL menu, point to **Section** and click **Assign section** (button on the toolbar).
Select necessary bars and assign the section to them.

Click **Assign section** button once again in order to make this command not active.

Note. The first method is helpful when you define section to large number of bars. Otherwise, when you define sections to separate bars, use the second method.

Step 4. Define loads on design model

Let us define two load cases: the first one is the dead weight and the second one is the load on nodes and bars of design model.

Dead weight may be defined if material density was specified for all sections or unit weight (if the section has numerical description). To define dead weight, on the MODEL menu, point to **Load case** and click **Add dead weight**. If material density or unit weight were not defined, then you will see appropriate message and dead weight is not added.

Load for the first load case is presented in the figure below.

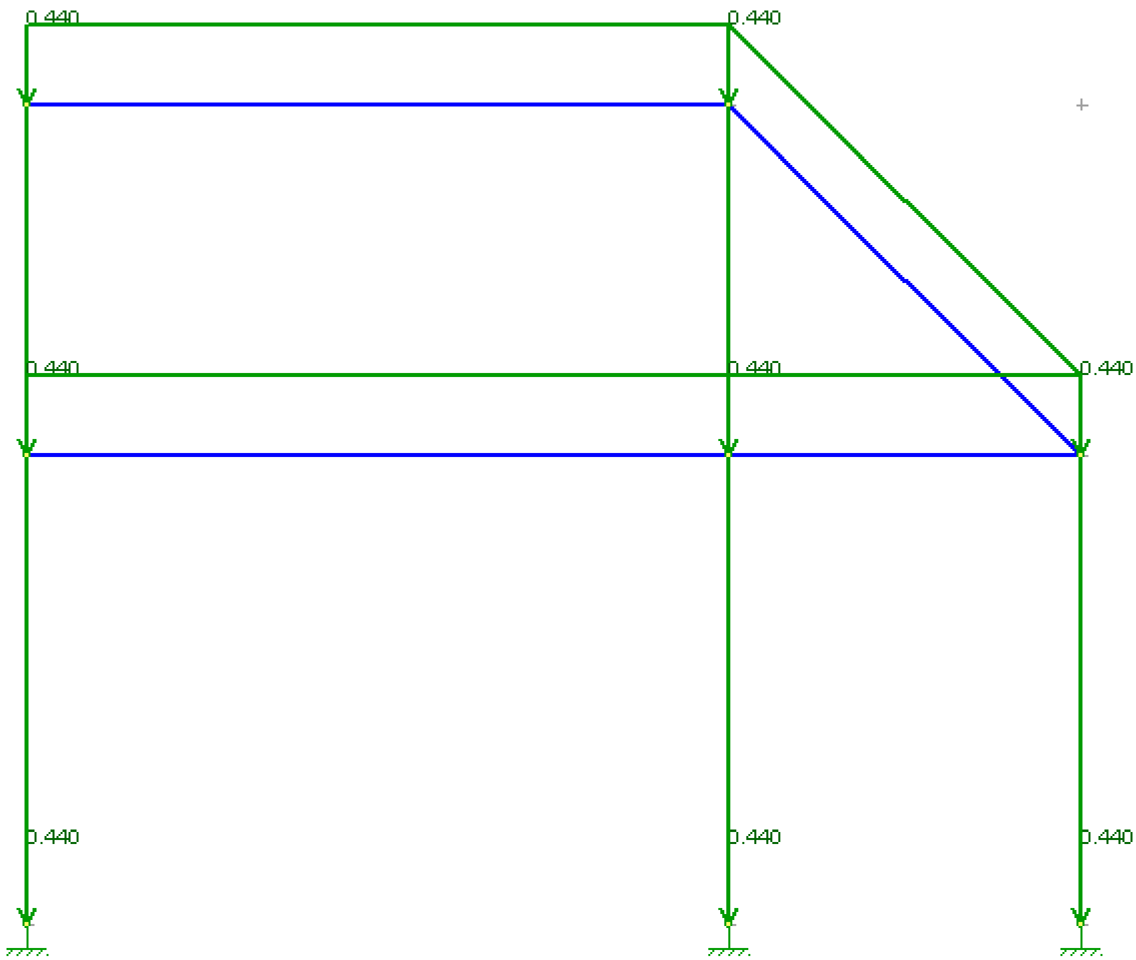


Figure 3.14

To define the second (and the following) load cases, on the MODEL menu, point to **Load case** and click **Current load case** (button on the toolbar).

Specify number of the load case in the **Current load case** dialog box.

To define nodal loads, on the MODEL menu, point to **Load case** and click **Define at node** (button on the toolbar).

In the **Load at nodes** dialog box, define horizontal force P = – 10 tf and click **Apply**.

On the MODEL menu, point to **Load case** and click **Apply load** (button on the toolbar). Then specify the node where the load should be applied to (in this example, it is the upper left node).

In the **Load at nodes** dialog box, buttons with green arrows correspond to force relative to appropriate axis in the global coordinate system while buttons with yellow arrows - to cinematic load (initial translation of nodes). Initial translations may be defined only to nodes where there are no supports.

To define local loads on bars, on the MODEL menu, point to **Load case** and click **Define on bar** (button  on the toolbar).

In the **Load on bars** dialog box, specify uniformly distributed load (5 tf/m) on inclined bar along the whole length of bar in local coordinate system along the Z-axis.

It is possible to define the following types of load:

1) Concentrated load in span in global or local coordinate system along the X-axis or Z-axis;

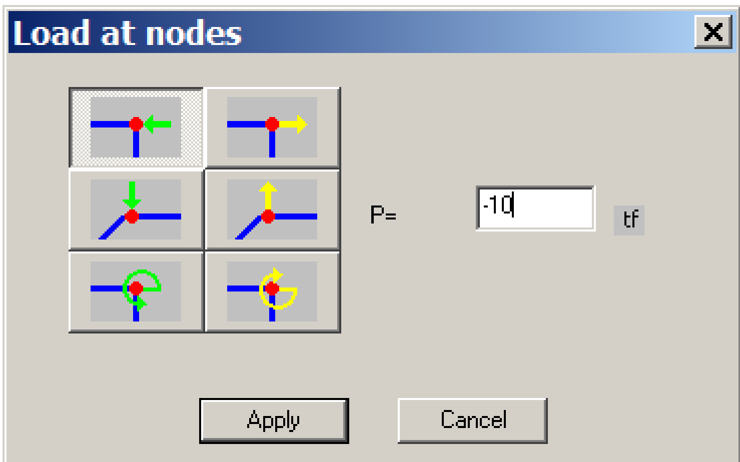


Figure 3.15

2) Concentrated moment in span;
3) Uniformly distributed load in span in global or local coordinate system along the X-axis or Z-axis;
4) Trapezoidal distributed load in span in global or local coordinate system along the X-axis or Z-axis;
5) Uniform heating along the local X-axis of the bar;
6) Non-uniform heating of the upper and lower fibre of the bar.

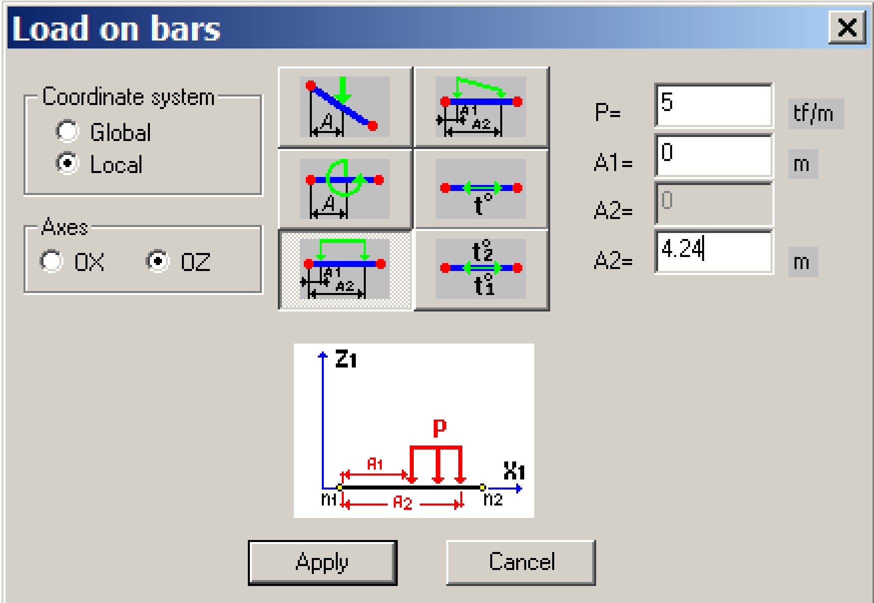


Figure 3.16

When the load is specified, click **Apply**.

On the MODEL menu, point to **Load case** and click **Apply load** (button on the toolbar).

Design model for the second load case is presented in the figure below.

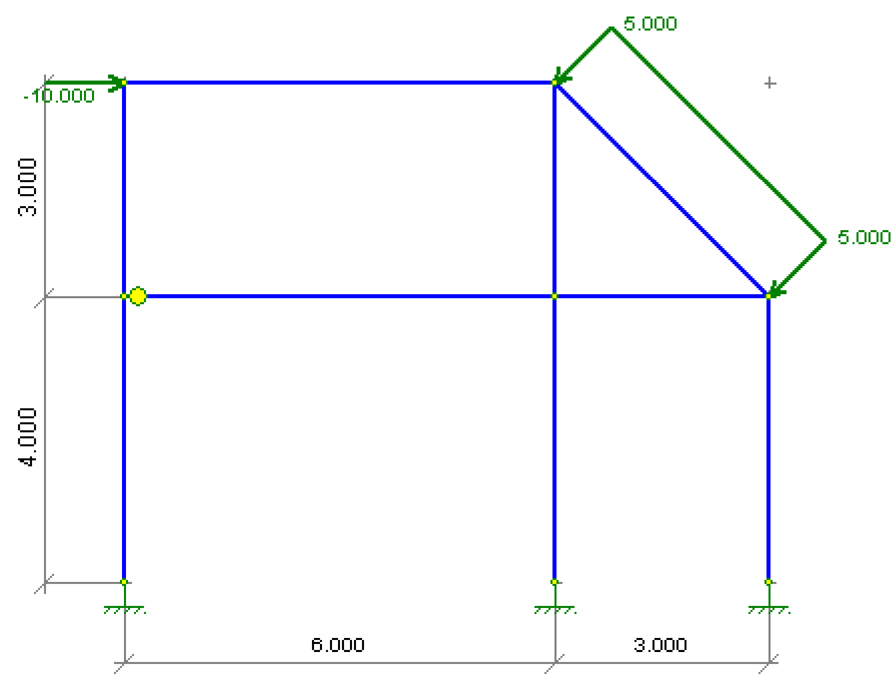


Figure 3.17

To start calculation, on the MODEL menu, point to **Calculation** and click **Analyse problem** (button on the toolbar).

When you see the message '*Calculation is complete*', you could evaluate analysis results.

Options to edit and visualize the model

Options to edit the model

Principal commands (modes) to edit the model are presented on the EDIT menu.

Undo

Select nodes

Select bars

Delete nodes

Delete bars

Remove restraint

Delete hinge

Delete load (MODEL menu / **Load case**)

Options to visualize the model

All commands to visualize the model are presented on the VISUALIZE menu. With the help of these commends you could show or hide the following data:

Numbers of nodes

Numbers of bars

Dimensions of the specified grid

Restraints

Hinges

Loads

Load values

Tools

Length between two nodes

Angle by three nodes .

Information about bar

Information about node

Zoom - to enlarge the model fragment

Fit in window - to display all objects on the screen

Output data

Define the number of the load case that you would like to evaluate (button № on the toolbar).

To display the output data, on the MODEL menu, point to **Calculation** and click one of the following commands to display appropriate data on the screen:

– Deformed shape;

– Axial force diagram N;

– Shear force diagram Qz ;

– Moment diagram My .

To display numerical values of nodal displacements and forces in bars, on the MODEL menu, point to **Calculation / Tables** and click **Nodal displacements** or **Internal forces at bar ends** .

Displacements/Load case 2

File

Node ...	X (mm)	Z (mm)	Uy (rad.x1000)
9	-3.31636	0.00755124	0.090772
10	-3.41715	-0.11866	0.590335
11	-3.37513	-0.0604073	-1.20252
12	0	0	0
13	0	0	0
14	-3.29009	0.00329099	-0.562206
15	0	0	0

a)

Forces/Load case 2

File

Element No.	Section	N (tf)	My (tf*m)	Qz (tf)
9	1	-8.06329	1.51205	-0.681639
9	2	-8.06329	-2.57778	-0.681639
10	1	-1.29823	-7.41465	11.8413
10	2	-1.29823	-2.1763	-9.35871
11	1	-7.24887	-0.404172	1.1641
11	2	-7.24887	4.25224	1.1641
12	1	-8.13671	-4.83687	1.22775
12	2	-8.13671	3.7574	1.22775
13	1	0.681639	1.51205	-1.93671
13	2	0.681639	-4.29809	-1.93671
14	1	0.394919	-4.29809	2.59881
14	2	0.394919	6.09715	2.59881
15	1	-4.53552	2.58048	-0.28672
15	2	-4.53552		-0.28672

b)

Figure 3.18

To save calculation results to a certain file, in the **Nodal displacements (Forces)** window, on the FILE menu, click **Save**.

To display information about certain bar (such as forces, diagrams of axial and shear forces, moments and displacements), click the **Information about bar** button.

To display information about certain node (such as nodal displacements), click the **Information about node** button.

Sign conventions for displacements and forces

Sign convention for displacements

Linear translations (X, Z) are considered positive if they are directed left-to right or upward along appropriate axes.

Rotation (UY) is considered positive if it rotates clockwise when you look at the screen.

Sign convention for forces

Axial forces (N) is considered positive if the bar is in tension.

Sign convention for other forces relates to the bar section that belongs to its end:

– positive bending moment (My) rotates anti-clockwise when you look from the end of the Y1-axis;

– positive shear force (Qz) acts along the Z1-axis.

Rectangular slab on elastic foundation

This module enables you to carry out static analysis of rectangular slab on elastic foundation. There may be one rectangular opening or a cutout in a slab.

Input data.

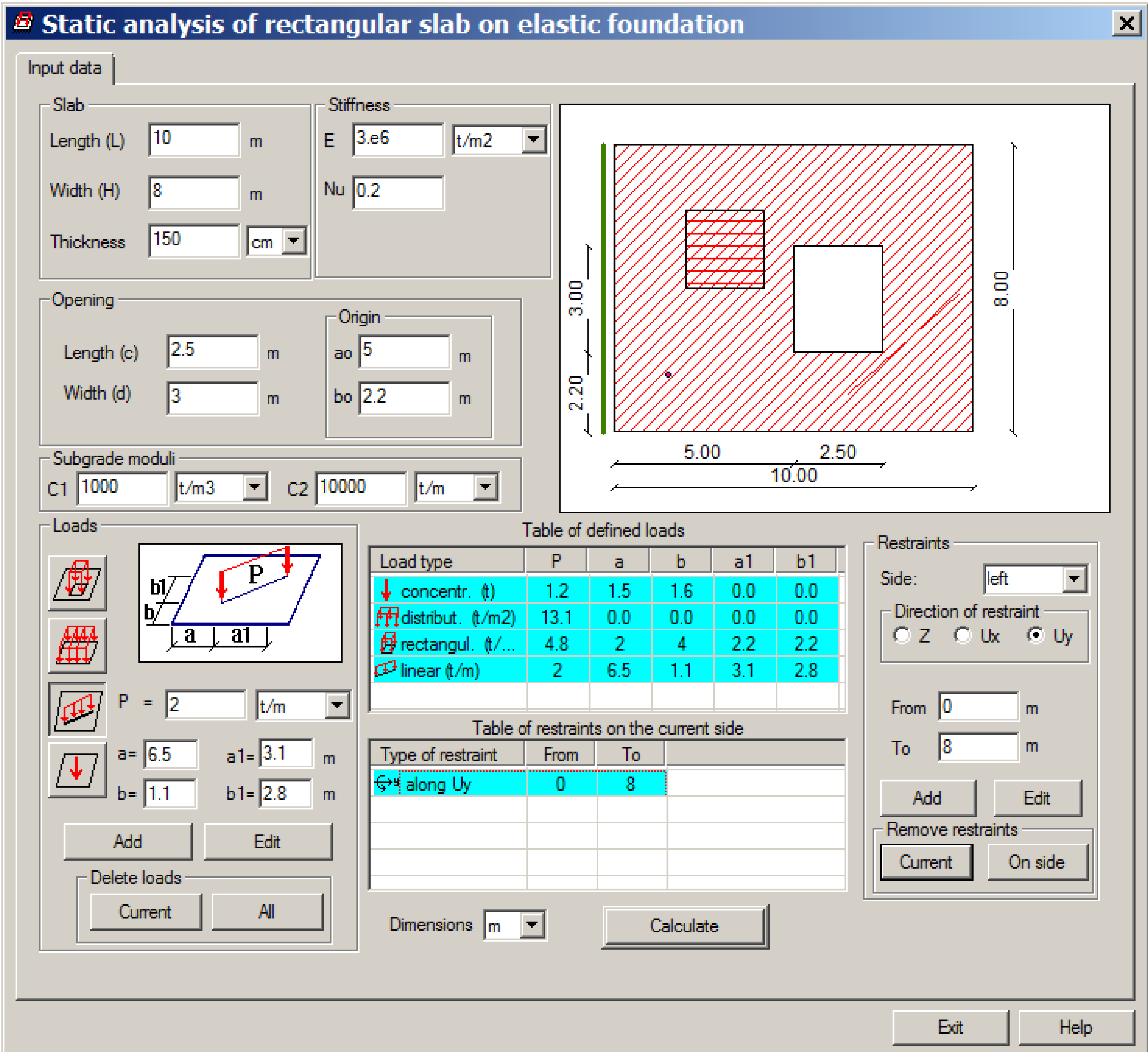


Figure 3.19

Slab

In the appropriate boxes define the following values:

– dimensions of slab — length, width, thickness;

– stiffness parameters — modulus of elasticity and Poisson's ratio;

– parameters of opening — length of opening, width of opening and distances from the opening to the nearest left corner of the slab. If there is no opening, you could define its dimensions as equal to zero.

Parameters of elastic foundation

In the appropriate boxes define moduli of subgrade reaction (also called subgrade modulus):

– C1 — subgrade modulus in compression;

– C2 — subgrade modulus in shear.

Loads

It is possible to define vertical concentrated and uniformly distributed loads. Uniformly distributed loads may be distributed across the whole slab, across arbitrary rectangular area or along the line on a slab.

– In the **Loads** area, select certain type of load from the four available types (just click appropriate icon), define necessary parameters — load value and other parameters depending on the load type and then click **Add**;

– To modify parameters of load, in the **Table of defined loads** select the row for appropriate load, input new parameters and click **Edit**;

– To delete load, in the **Table of defined loads** select the row for appropriate load, under **Delete loads** click **Current**. To delete all loads, click **All**.

Boundary conditions

Define line segments with certain types of boundary conditions on one side of a slab or on several sides of a slab, that is, restraints should be imposed. The following types of restraints are allowed:

– Z — linear restraint along the axis that is orthogonal to the slab surface;

– Ux — restraint against rotation about the axis that is parallel to the slab length;

– Uy — restraint against rotation about the axis that is parallel to the slab width.

– In the **Restraints** area, in the drop-down list, define the side of the slab where boundary conditions should be applied.

– With an option button, define direction along which displacements should not be allowed. In the **From** and **To** boxes, define distances to restraints, then click **Add**. If the values in the **From** and **To** boxes coincide, restraint is supposed at this point.

– To modify distances to restraint, select appropriate row in the **Table of restraints on the current side** table, define new parameters and click **Edit**.

– To remove restraints, in the **Table of restraints on the current side** table, select appropriate row and click **Current**. To remove all restraints on the current side of a slab, click **On side**.

When all input data is defined, click **Calculate**.

Output data.

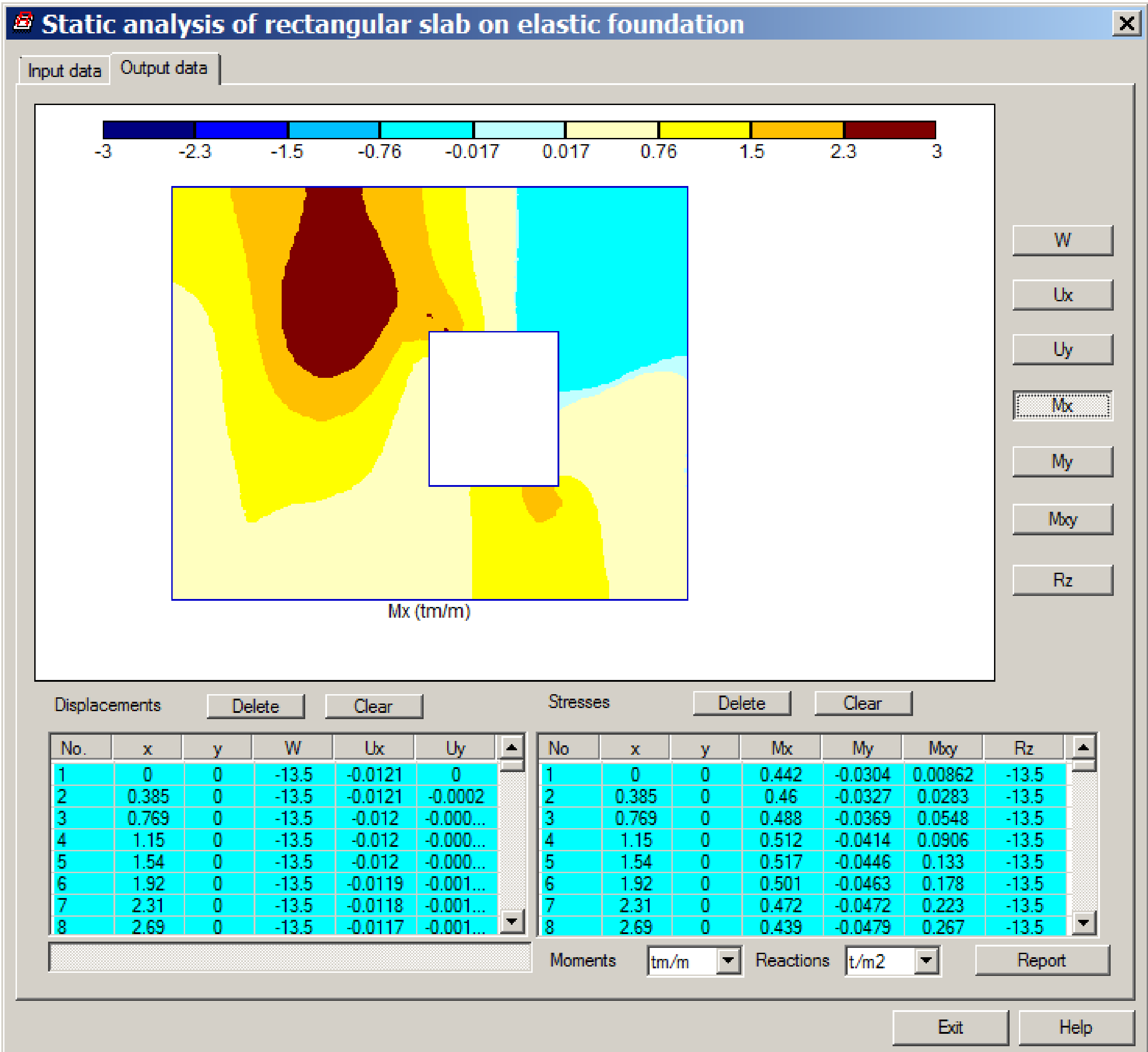


Figure 3.20

Calculation is made by finite element method in displacements. When calculation procedure is complete, the **Output data** tab will be displayed in the dialog box. The following data is calculated:

– vertical displacements **W** and rotation angles **Ux** , **Uy** ;
– bending moments **Mx**, **My** и **Mxy;**
– reaction of foundation **Rz**.

This data is presented as contour plots, the values are presented in the table.

– To generate and view certain results, just click appropriate button. All images presented on the screen are automatically placed to the report. When you move the pointer across the contour plot on the image, at the bottom of the dialog box you will see information about location of the pointer and the value of appropriate contour plot. When you click at any point on the contour plot image, the value of this contour plot will be added to the table.

– **Displacements** and **Stresses** tables for all elements of the model are presented after calculation. To delete all rows in the table, click **Clear** button above the table. Then you could input values of contour plots at certain points (with the help of the pointer), it will reduce the report. It is possible to vary measurement units for moments and reaction of foundation.

– To delete one row from the table, select the row with the pointer and click **Delete** button above the table.

To generate the report file in HTML format, click **Report**.

Rectangular slab

This module enables you to carry out static analysis of rectangular roof slabs and floor slabs. There may be one rectangular opening or a cutout in a slab.

Input data.

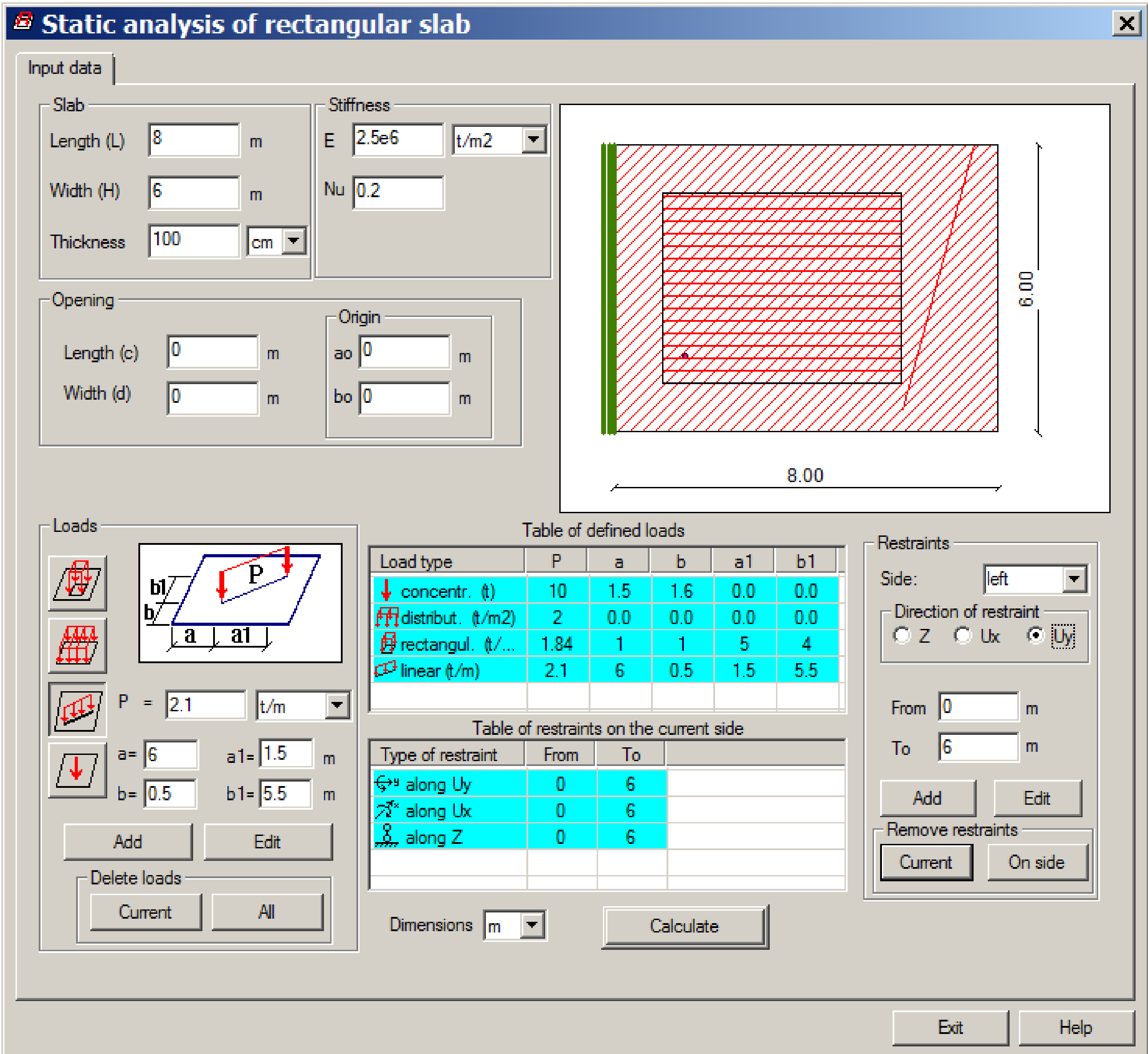


Figure 3.21

Slab

In the appropriate boxes define the following values:

– dimensions of slab — length, width, thickness;

– stiffness parameters — modulus of elasticity and Poisson's ratio;

– parameters of opening — length of opening, width of opening and distances from the opening to the nearest left corner of the slab. If there is no opening, you could define its dimensions as equal to zero.

Loads

It is possible to define vertical concentrated and uniformly distributed loads. Uniformly distributed loads may be distributed across the whole slab, across arbitrary rectangular area or along the line on a slab.

– In the **Loads** area, select certain type of load from the four available types (just click appropriate icon), define necessary parameters — load value and other parameters depending on the load type and then click **Add**;
– To modify parameters of load, in the **Table of defined loads** select the row for appropriate load, input new parameters and click **Edit**;
– To delete load, in the **Table of defined loads** select the row for appropriate load, under **Delete loads** click **Current**. To delete all loads, click **All**.

Boundary conditions

Define line segments with certain types of boundary conditions on one side of a slab or on several sides of a slab, that is, restraints should be imposed. The following types of restraints are allowed:
– Z — linear restraint along the axis that is orthogonal to the slab surface;
– Ux — restraint against rotation about the axis that is parallel to the slab length;
– Uy — restraint against rotation about the axis that is parallel to the slab width.

– In the **Restraints** area, in the drop-down list, define the side of the slab where boundary conditions should be applied.
– With an option button, define direction along which displacements should not be allowed. In the **From** and **To** boxes, define distances to restraints, then click **Add**. If the values in the **From** and **To** boxes coincide, restraint is supposed at this point.
– To modify distances to restraint, select appropriate row in the **Table of restraints on the current side** table, define new parameters and click **Edit**.
– To remove restraints, in the **Table of restraints on the current side** table, select appropriate row and click **Current**. To remove all restraints on the current side of a slab, click **On side**.

When all input data is defined, click **Calculate**.

Output data.

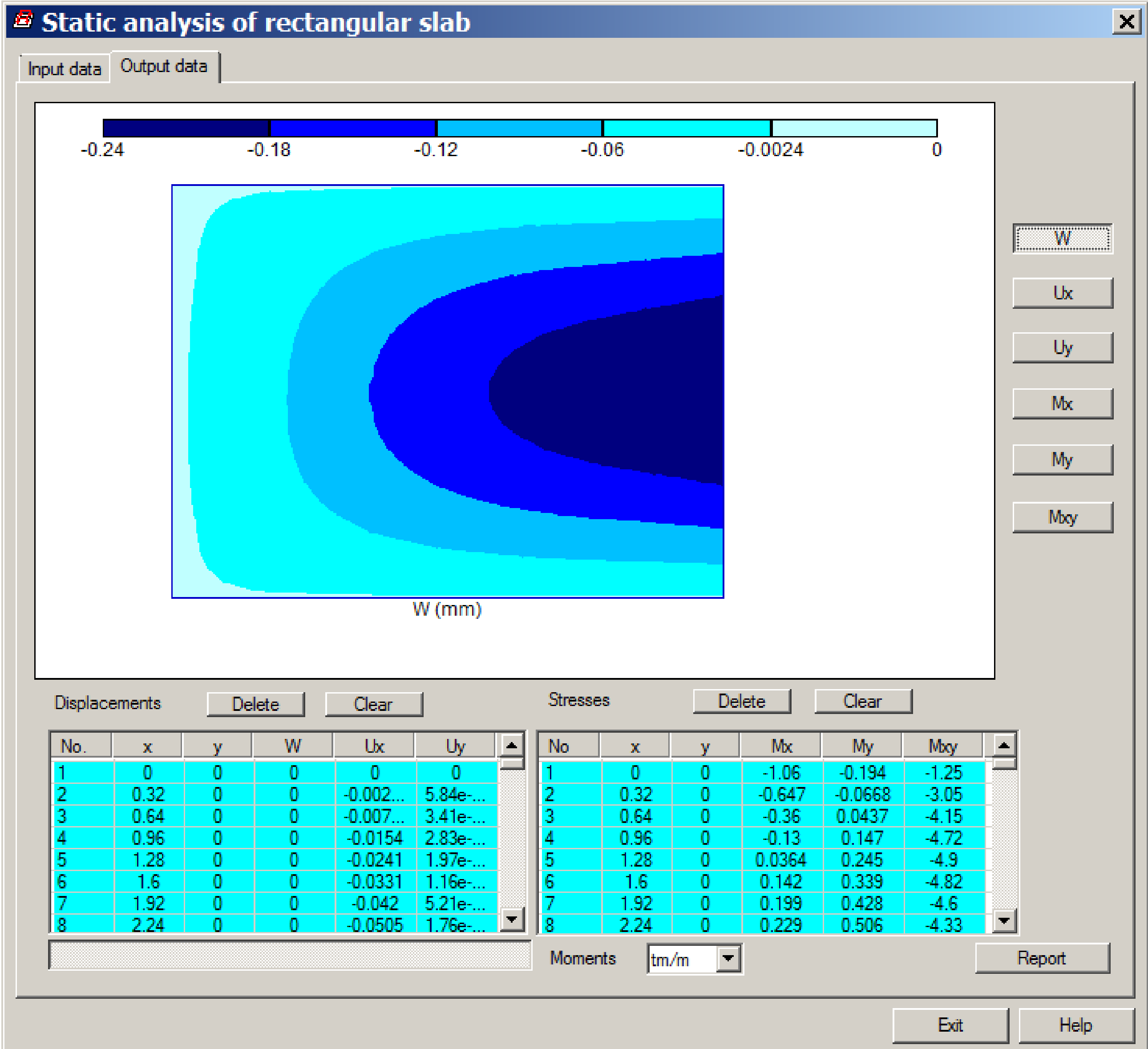


Figure 3.22

Calculation is made by finite element method in displacements. When calculation procedure is complete, the **Output data** tab will be displayed in the dialog box. The following data is calculated:

– vertical displacements **W** and rotation angles **Ux** , **Uy** ;

– bending moments **Mx**, **My** и **Mxy**.

This data is presented as contour plots, the values are presented in the table.

– To generate and view certain results, just click appropriate button. All images presented on the screen are automatically placed to the report. When you move the pointer across the contour plot on the image, at the bottom of the dialog box you will see information about location of the pointer and the value of appropriate contour plot. When you click at any point on the contour plot image, the value of this contour plot will be added to the table.

– **Displacements** and **Stresses** tables for all elements of the model are presented after calculation. To delete all rows in the table, click **Clear** button above the table. Then you could input values of contour plots at certain points (with the help of the pointer), it will reduce the report. It is possible to vary measurement units.

– To delete one row from the table, select the row with the pointer and click **Delete** button above the table.

To generate the report file in HTML format, click **Report**.

Wall-beam

This module enables you to carry out static analysis of rectangular wall-beam. There may be one rectangular opening or a cutout.

Input data.

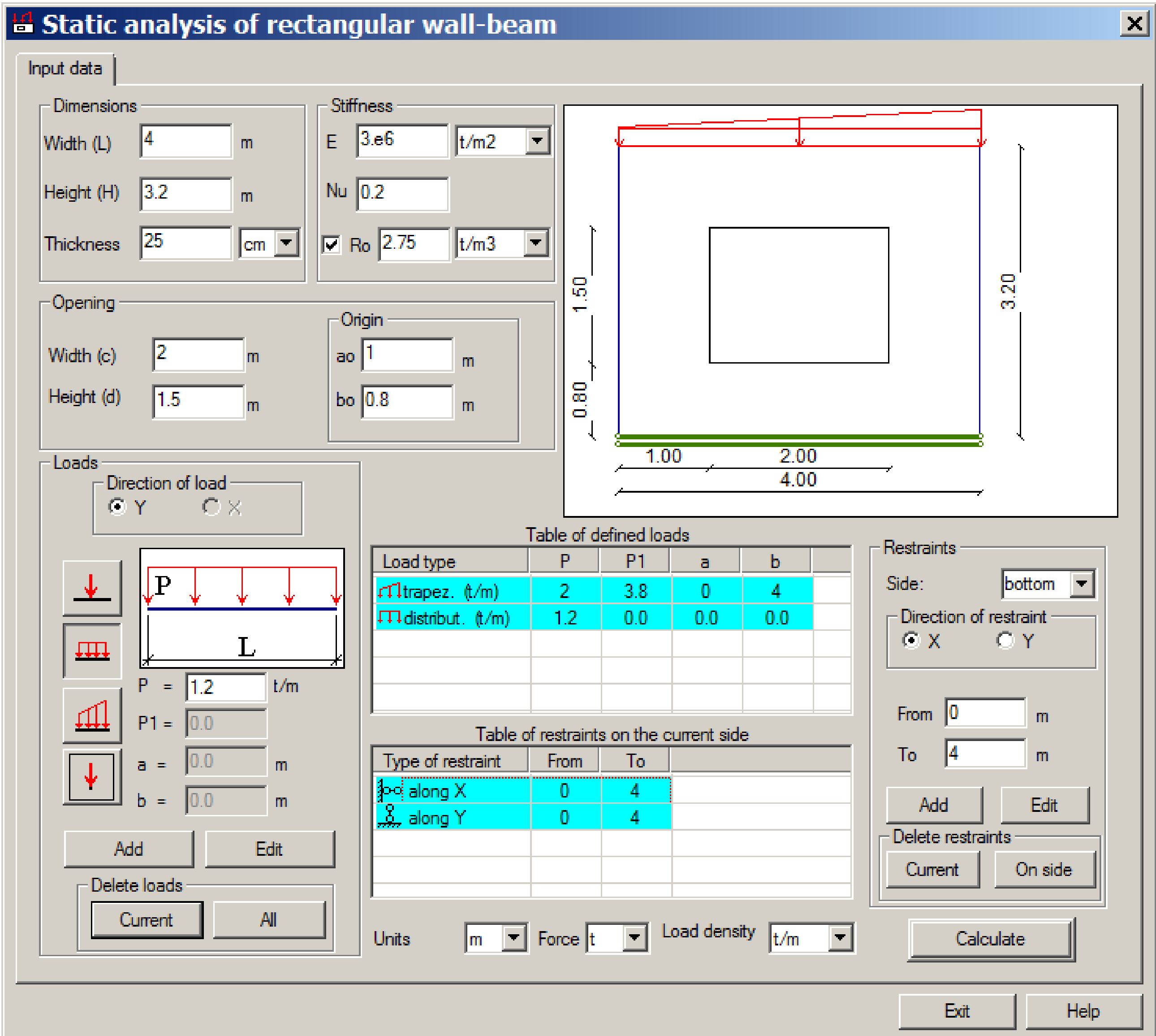


Figure 3.23

Wall-beam

In the appropriate boxes define the following values:

– dimensions of wall-beam — length, width, thickness;

– stiffness parameters — modulus of elasticity and Poisson's ratio; material density (to automatically take account of dead weight);

– parameters of opening — length of opening, width of opening and distances from the opening to the bottom left corner of the wall-beam. If there is no opening, you could define its dimensions as equal to zero.

Loads

– In the **Loads** area, define direction of load, then select certain type of load from the four available types (just click appropriate icon), define necessary parameters — load value and other parameters depending on the load type and then click **Add**;

– To modify parameters of load, in the **Table of defined loads** select the row for appropriate load, input new parameters and click **Edit**;

– To delete load, in the **Table of defined loads** select the row for appropriate load, under **Delete loads** click **Current**. To delete all loads, click **All**.

Boundary conditions

– In the **Restraints** area, in the drop-down list, define the side of the wall-beam where boundary conditions should be applied.
– With an option button, define direction along which displacements should not be allowed (X — linear restraint along the horizontal axis, Y — linear restraint along the vertical axis). In the **From** and **To** boxes, define distances to restraints, then click **Add**. If the values in the **From** and **To** boxes coincide, restraint is supposed at this point.
– To modify distances to restraint, select appropriate row in the **Table of restraints on the current side** table, define new parameters and click **Edit**.
– To remove restraints, in the **Table of restraints on the current side** table, select appropriate row or click **Current**. To remove all restraints on the current side of a slab, click **On side**.

When all input data is defined, click **Calculate**.

Output data.

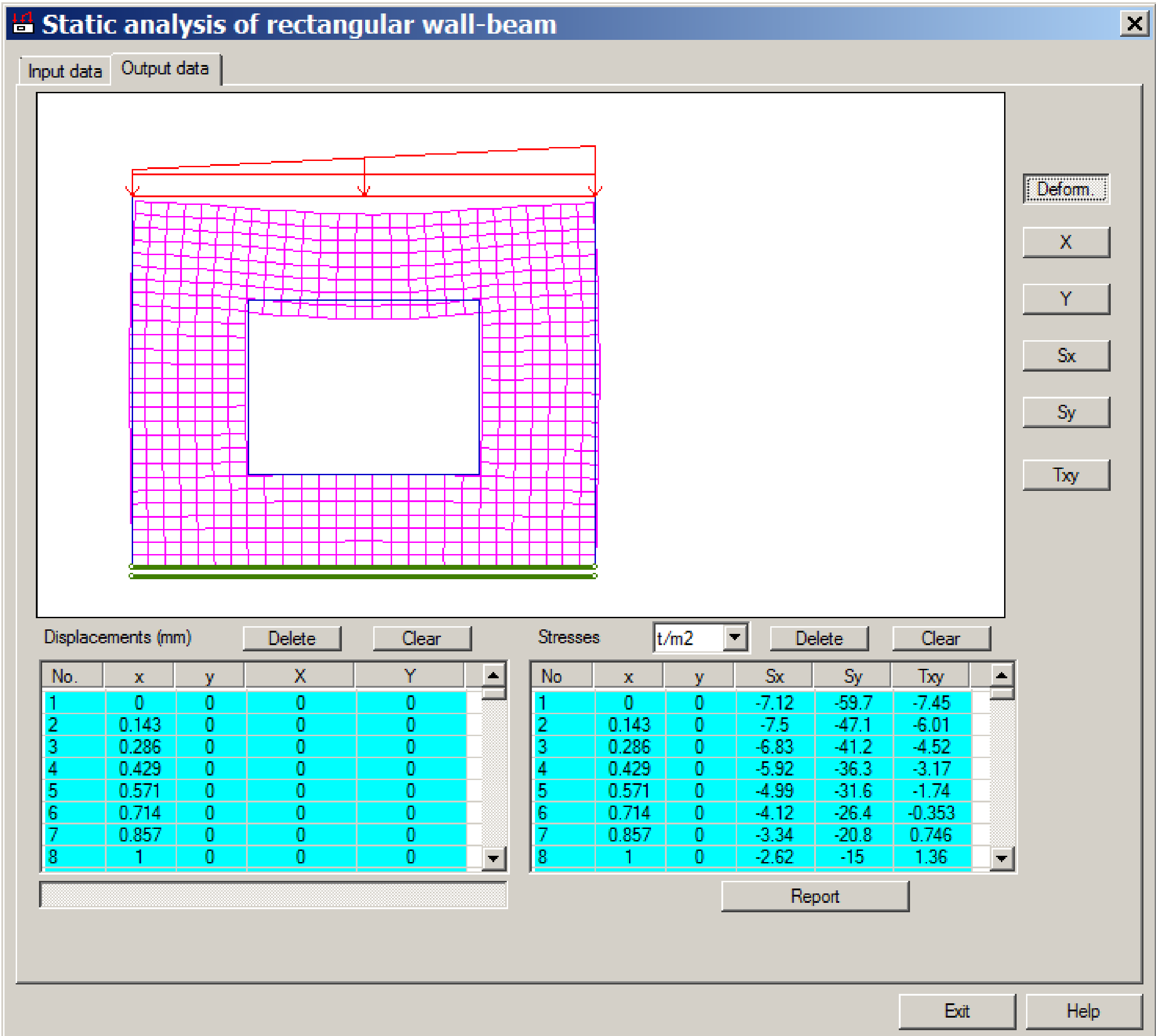


Figure 3.24

– To generate and view certain results (such as deformed shape, contour plots of vertical Y and horizontal X displacements, contour plots of normal Sx, Sy and shear Txy stresses), just click appropriate button. All images presented on the screen are automatically placed to the report. When you move the pointer across the contour plot on the image, at the bottom of the dialog box you will see information about location of the pointer and the value of appropriate contour plot. When you click at any point on the contour plot image, the value of this contour plot will be added to the table.

– **Displacements** and **Stresses** tables for all elements of the model are presented after calculation. To delete all rows in the table, click **Clear** button above the table. Then you could input values of contour plots at certain points (with the help of the pointer), it will reduce the report. It is also possible to vary measurement units.

– To delete one row from the table, select the row with the pointer and click **Delete** button above the table.

To generate the report file in HTML format, click **Report**.

Shell on rectangular plan

This module enables you to carry out static analysis of convex parabolic and spherical shells that are rectangular in plan.

Input data.

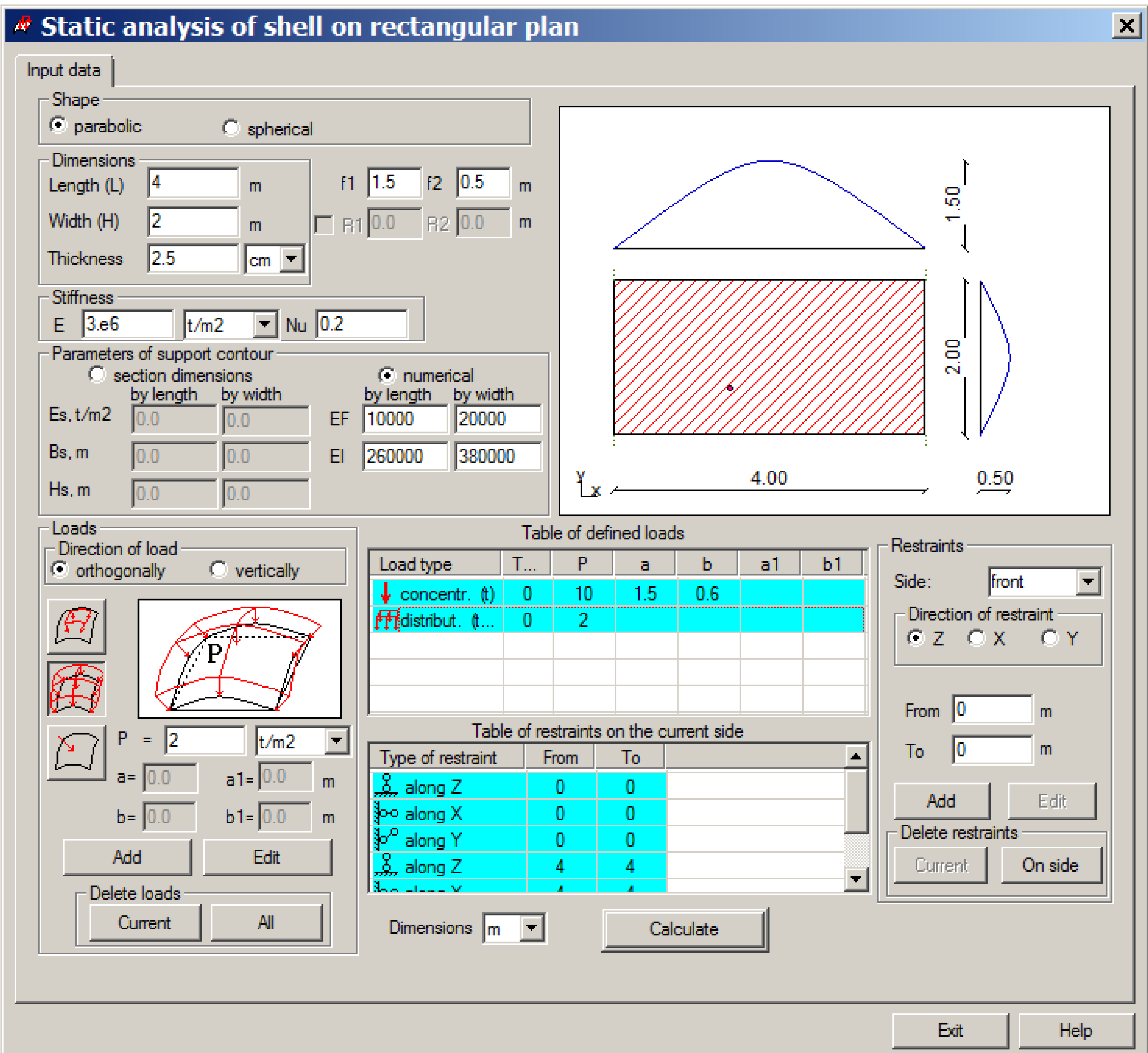


Figure 3.25

Shell

In the appropriate boxes define the following values:

 www.liraland.com

– dimensions in plan (length, width), thickness of shell;
– stiffness parameters — modulus of elasticity and Poisson's ratio;
– parameters for the surface — shape of the surface, rise and radius for left and right sides (f2, R2) or for front and back sides (f1, R1).
If there is a support contour, define its stiffness properties – modulus of elasticity, width and height or numerical stiffness.

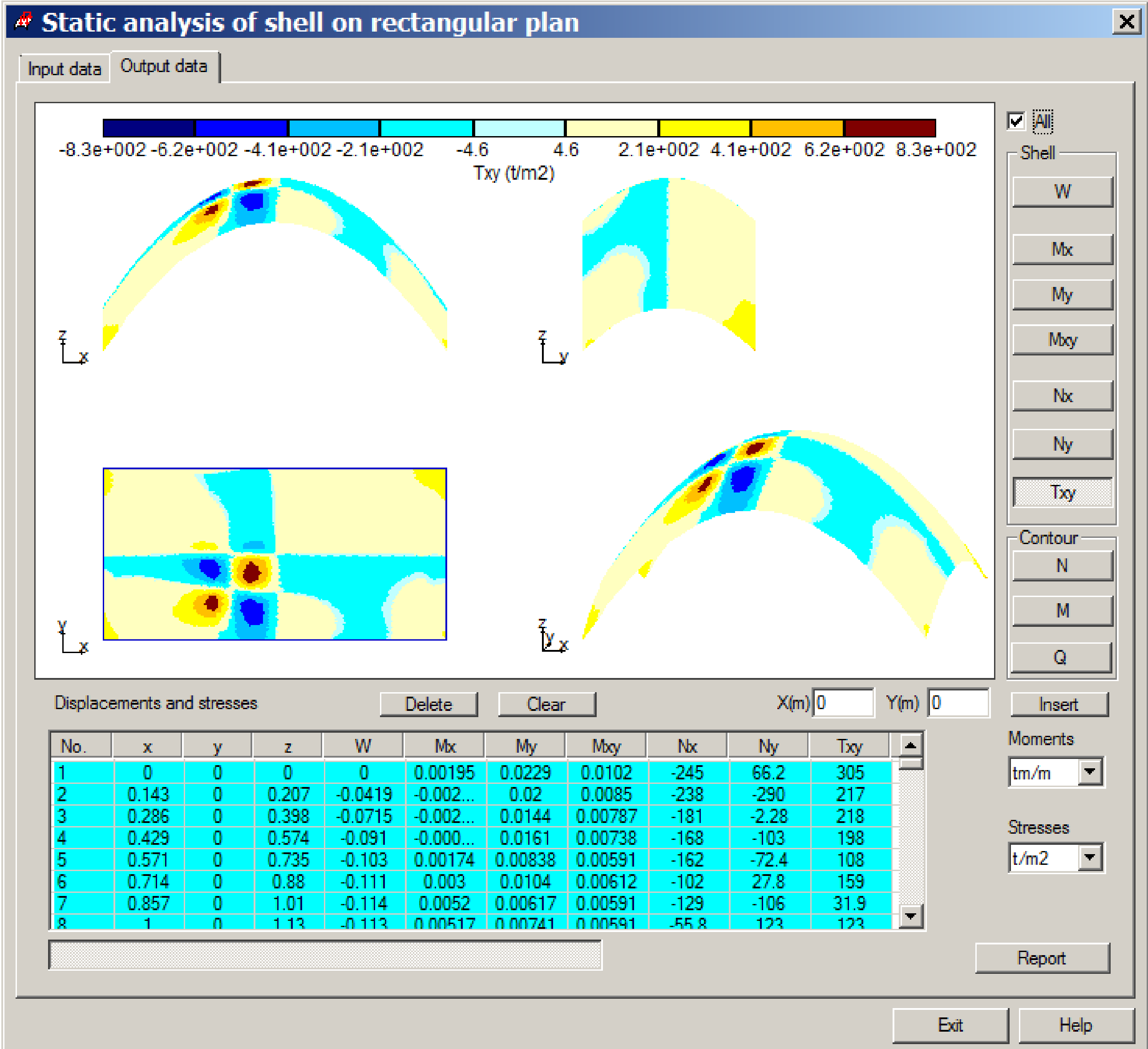
Loads
– In the **Loads** area, define direction of load, then select certain type of load from the three available types (just click appropriate icon), define necessary parameters — load value and other parameters depending on the load type and then click **Add**;
– To modify parameters of load, in the **Table of defined loads** select the row for appropriate load, input new parameters and click **Edit**;
– To delete load, in the **Table of defined loads** select the row for appropriate load, under **Delete loads** click **Current**. To delete all loads, click **All**.

Boundary conditions
Define line segments with certain types of boundary conditions on one side or on several sides, that is, restraints should be imposed. The following types of restraints are allowed:
– Z — linear restraint along the vertical axis;
– X, Y — linear restraints along appropriate horizontal axis.

– In the **Restraints** area, in the drop-down list, define the side where boundary conditions should be applied.
– With an option button, define direction along which displacements should not be allowed. In the **From** and **To** boxes, define distances to restraints, then click **Add**. If the values in the **From** and **To** boxes coincide, restraint is supposed at this point.
– To modify distances to restraint, select appropriate row in the **Table of restraints on the current side** table, define new parameters and click **Edit**.
– To remove restraints, in the **Table of restraints on the current side** table, select appropriate row or click **Current**. To remove all restraints on the current side of a slab, click **On side**.

When all input data is defined, click **Calculate**.

Output data.



No.	x	y	z	W	Mx	My	Mxy	Nx	Ny	Txy
1	0	0	0	0	0.00195	0.0229	0.0102	-245	66.2	305
2	0.143	0	0.207	-0.0419	-0.002...	0.02	0.0085	-238	-290	217
3	0.286	0	0.398	-0.0715	-0.002...	0.0144	0.00787	-181	-2.28	218
4	0.429	0	0.574	-0.091	-0.000...	0.0161	0.00738	-168	-103	198
5	0.571	0	0.735	-0.103	0.00174	0.00838	0.00591	-162	-72.4	108
6	0.714	0	0.88	-0.111	0.003	0.0104	0.00612	-102	27.8	159
7	0.857	0	1.01	-0.114	0.0052	0.00617	0.00591	-129	-106	31.9
8	1	0	1.13	-0.113	0.00517	0.00741	0.00591	-55.8	123	123

Figure 3.26 a)

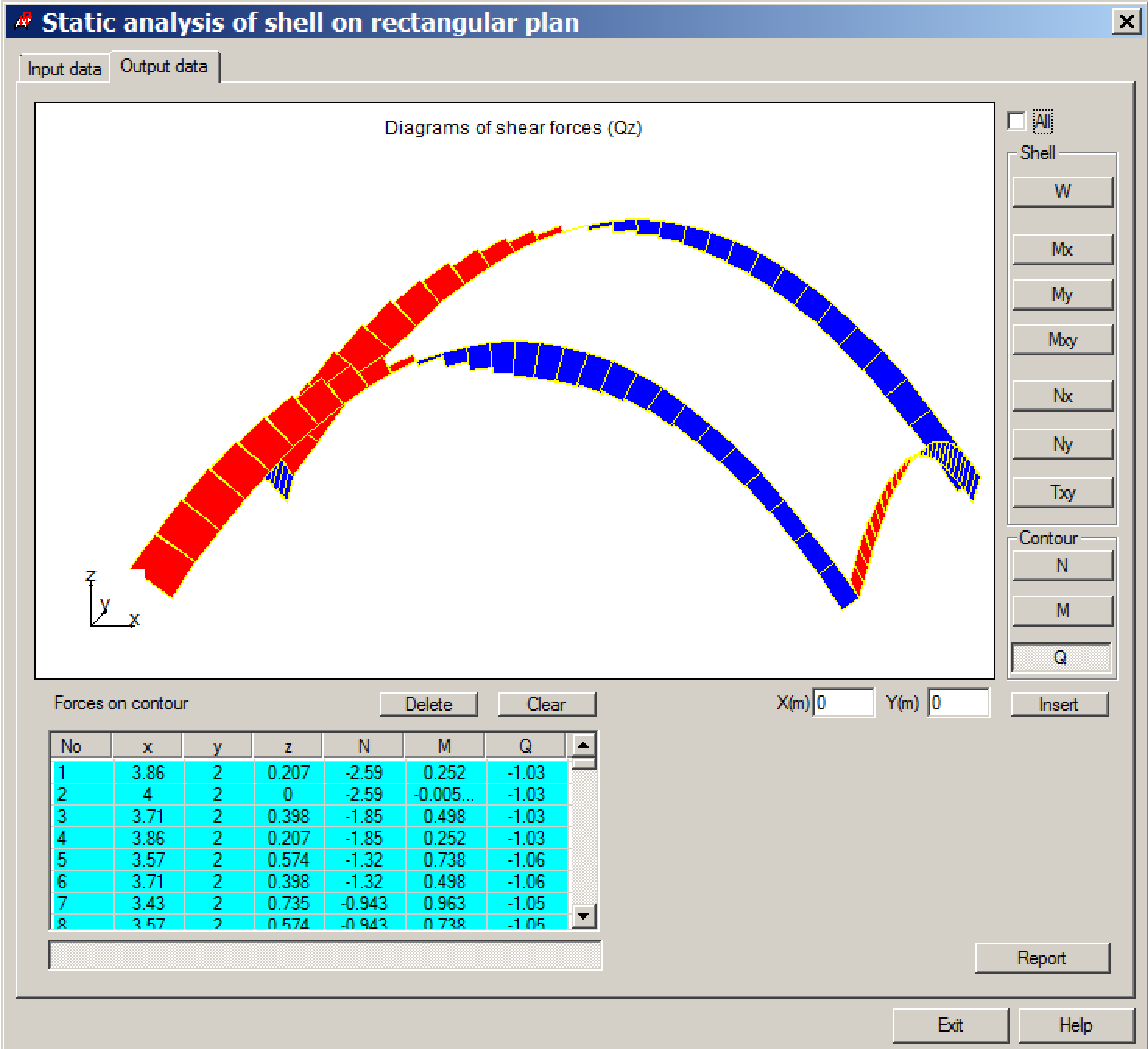


Figure 3.26 b)

Calculation is made by finite element method in displacements. When calculation procedure is complete, the **Output data** tab will be displayed in the dialog box. The following data is calculated:

– vertical displacements W ;
– bending moments Mx, My and torsion moment Mxy;
– normal stresses Nx, Ny and shear stresses Txy;
– forces N, M, Q in support contour.

This data is presented as contour plots (W, Mx, My, Mxy, Nx, Ny, Txy) or diagrams (N, M, Q), the values are presented in the table.

– Contour plots may be presented in different ways (it depends on the **All** check box status - on/off): 1) one by one in certain projection (XOY, XOZ, YOZ) or in dimetric projection; 2) all contour plots together. To generate certain contour plot, just click appropriate button. If the **All** check box is not selected, then when you click any button for a certain contour plot, the plots will change each other. To simultaneously display contour plots or diagrams in all three planes and in dimetric projection, select the **All** check box. The data may be added to the table only from projection on the XOY-plane (in plan). In this mode when you move the pointer across the contour plot on the image, at the bottom of the dialog box you will see information about location of the pointer and the value of appropriate contour plot. When you click at any point on the contour plot image, the value of this contour plot will be added to the table. To add result values to the table, you could also define coordinates of a point in appropriate boxes and click **Insert**. The report file contains only one projection for every contour plot or diagram, that is, the latest image visualized on the screen for the certain parameter.

– **Displacements and stresses** and **Forces on contour** tables for all elements of the model are presented after calculation. To delete all rows in the table, click **Clear** button above the table. Then you could input values of contour plots at certain points (with the help of the pointer), it will reduce the report. It is possible to vary measurement units for moments and stresses.

– To delete one row from the table, select the row with the pointer and click **Delete** button above the table.

To generate the report file in HTML format, click **Report**.

Shell on circular plan

This module enables you to carry out static analysis of convex parabolic, spherical and conic shells that are circular in plan.

Input data.

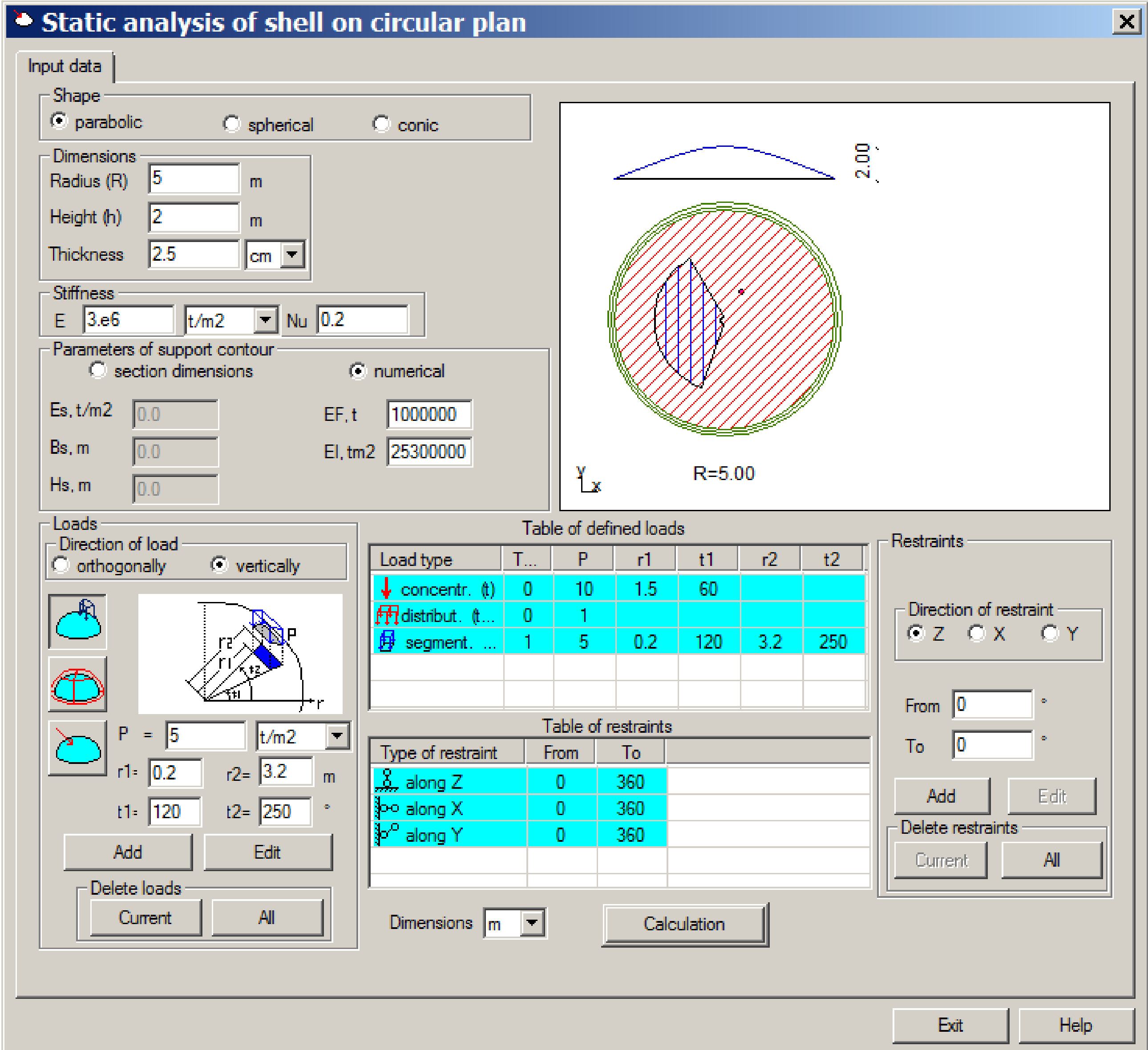


Figure 3.27

Shell

In the appropriate boxes define the following values:

– dimensions — radius in plan, rise (height), thickness of shell;

– stiffness parameters — modulus of elasticity and Poisson's ratio;

– parameters for the surface — shape of the surface (parabolic, spherical, conical).

If there is a support contour, define its stiffness properties – modulus of elasticity, width and height or numerical stiffness.

Loads

– In the **Loads** area, define direction of load, then select certain type of load from the three available types (just click appropriate icon), define necessary parameters — load value and other parameters depending on the load type and then click **Add**;

– To modify parameters of load, in the **Table of defined loads** select the row for appropriate load, input new parameters and click **Edit**;

– To delete load, in the **Table of defined loads** select the row for appropriate load, under **Delete loads** click **Current**. To delete all loads, click **All**.

Boundary conditions

Define segments with certain types of boundary conditions, that is, restraints should be imposed. The following types of restraints are allowed:
– Z — linear restraint along the vertical axis;
– X, Y — linear restraints along appropriate horizontal axis.

– With an option button, define direction along which displacements should not be allowed. In the **From** and **To** boxes, define distances to restraints, then click **Add**. If the values in the **From** and **To** boxes coincide, restraint is supposed at this point.
– To modify distances to restraint, select appropriate row in the **Table of restraints on the current side** table, define new parameters and click **Edit**.
– To remove restraints, in the **Table of restraints on the current side** table, select appropriate row or click **Current**. To remove all restraints on the current side of a slab, click **On side**.

When all input data is defined, click **Calculate**.

Output data.

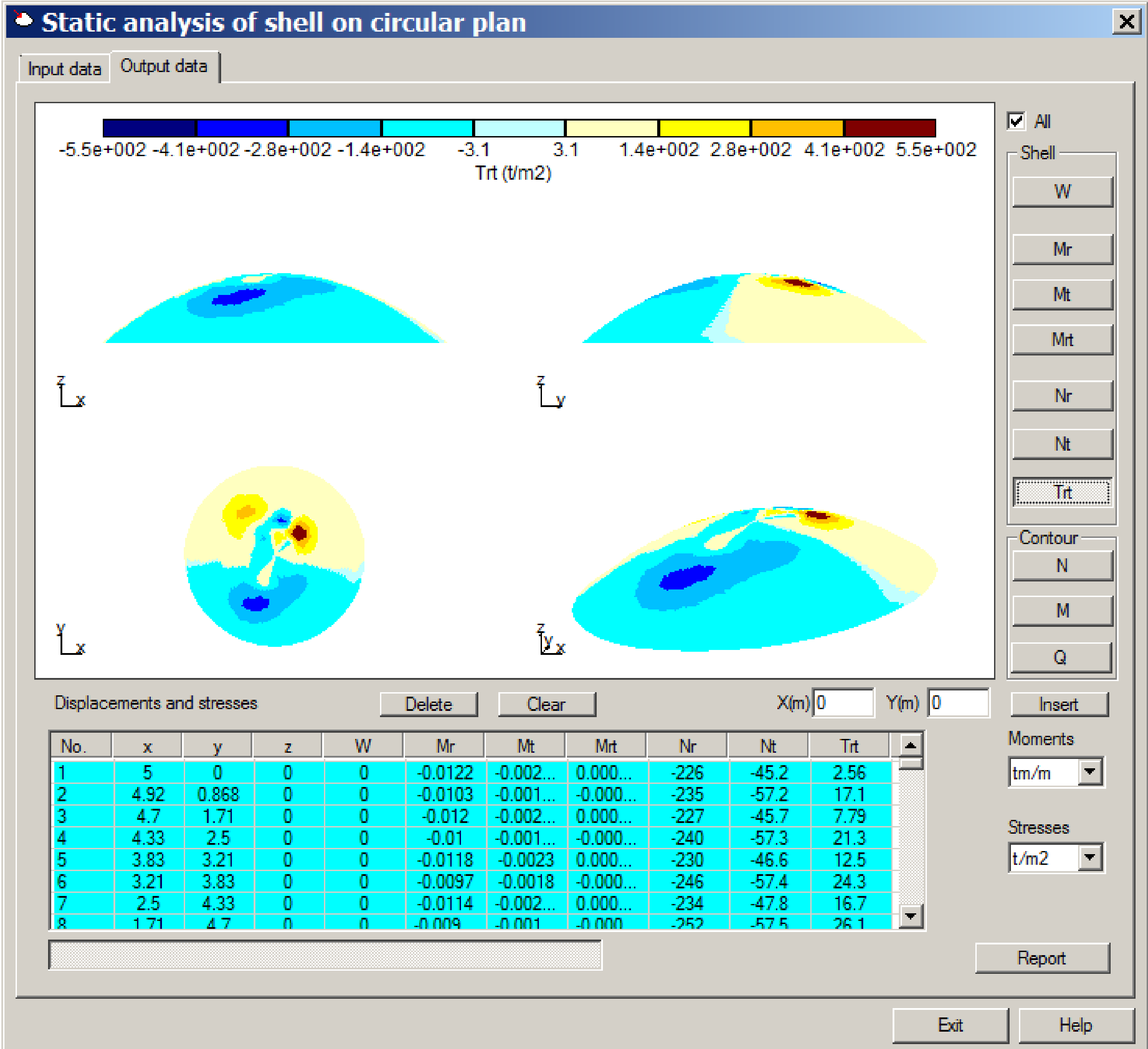

Static analysis of shell on circular plan
Input data
Output data
-5.5e+002 -4.1e+002 -2.8e+002 -1.4e+002 -3.1 3.1 1.4e+002 2.8e+002 4.1e+002 5.5e+002
Trt (t/m2)
All
Shell
W
Mr
Mt
Mrt
Nr
Nt
Trt
Contour
N
M
Q
Displacements and stresses
Delete
Clear
X(m) 0
Y(m) 0
Insert
Moments
tm/m
Stresses
t/m2
Report
Exit
Help
No. x y z W Mr Mt Mrt Nr Nt Trt
1 5 0 0 0 -0.0122 -0.002... 0.000... -226 -45.2 2.56
2 4.92 0.868 0 0 -0.0103 -0.001... -0.000... -235 -57.2 17.1
3 4.7 1.71 0 0 -0.012 -0.002... 0.000... -227 -45.7 7.79
4 4.33 2.5 0 0 -0.01 -0.001... -0.000... -240 -57.3 21.3
5 3.83 3.21 0 0 -0.0118 -0.0023 0.000... -230 -46.6 12.5
6 3.21 3.83 0 0 -0.0097 -0.0018 -0.000... -246 -57.4 24.3
7 2.5 4.33 0 0 -0.0114 -0.002... 0.000... -234 -47.8 16.7

Figure 3.28 a)

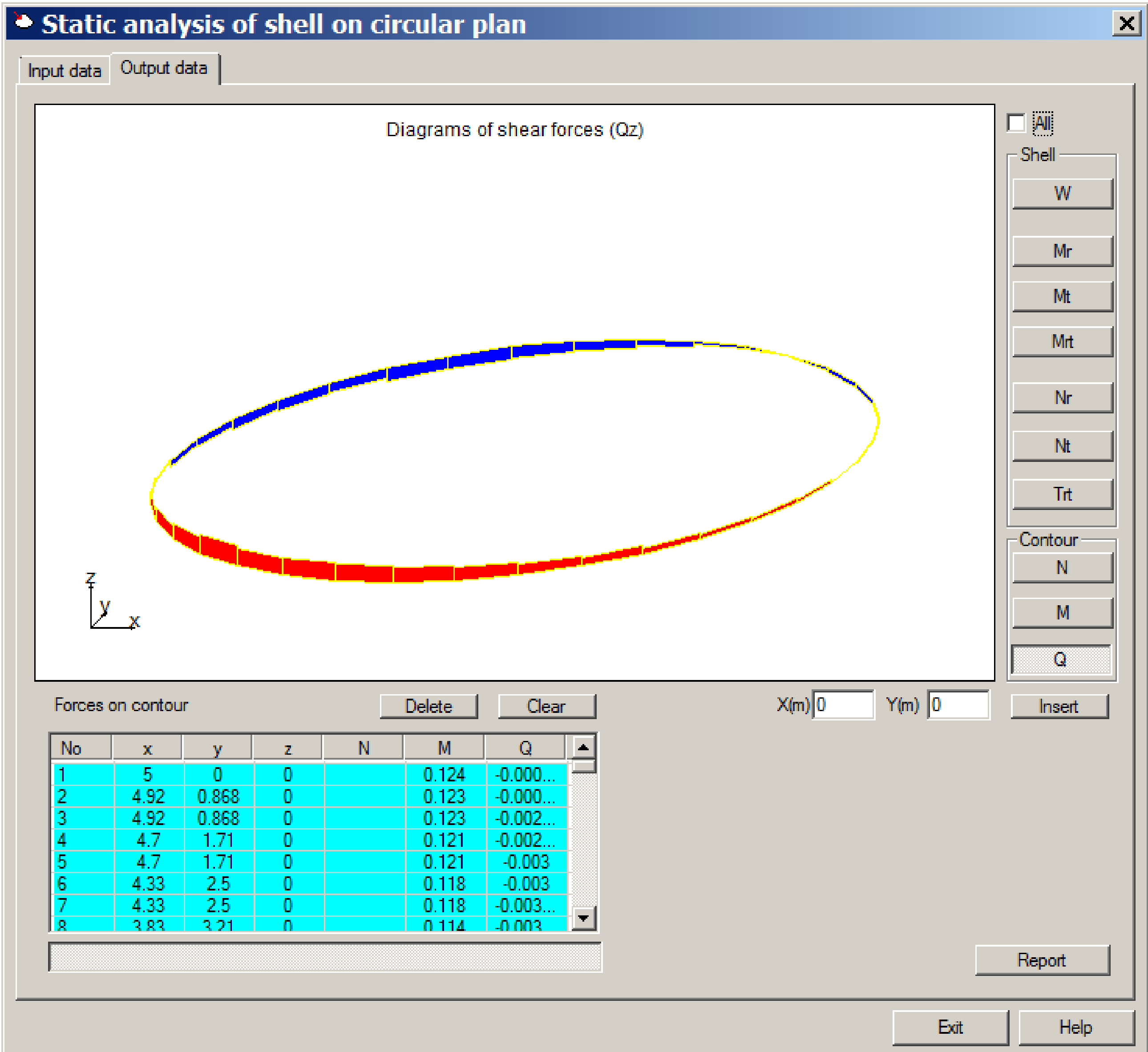


Figure 3.28 b)

Calculation is made by finite element method in displacements. When calculation procedure is complete, the **Output data** tab will be displayed in the dialog box. The following data is calculated:

– vertical displacements W ;
– bending moments Mr, Mt and torsion moment Mrt;
– normal stresses Nr, Nt and shear stresses Trt;
– forces N, M, Q in support contour.

This data is presented as contour plots (W, Mr, Mt, Mrt, Nr, Nt, Trt) or diagrams (N, M, Q), the values are presented in the table.

– Contour plots may be presented in different ways (it depends on the **All** check box status - on/off): 1) one by one in certain projection (XOY, XOZ, YOZ) or in dimetric projection; 2) all contour plots together. To generate certain contour plot, just click

 www.liraland.com

appropriate button. If the **All** check box is not selected, then when you click any button for a certain contour plot, the plots will change each other. To simultaneously display contour plots or diagrams in all three planes and in dimetric projection, select the **All** check box. The data may be added to the table only from projection on the XOY-plane (in plan). In this mode when you move the pointer across the contour plot on the image, at the bottom of the dialog box you will see information about location of the pointer and the value of appropriate contour plot. When you click at any point on the contour plot image, the value of this contour plot will be added to the table. To add result values to the table, you could also define coordinates of a point in appropriate boxes and click **Insert**. The report file contains only one projection for every contour plot or diagram, that is, the latest image visualized on the screen for the certain parameter.

– **Displacements and stresses** and **Forces on contour** tables for all elements of the model are presented after calculation. To delete all rows in the table, click **Clear** button above the table. Then you could input values of contour plots at certain points (with the help of the pointer), it will reduce the report. It is possible to vary measurement units for moments and stresses.

– To delete one row from the table, select the row with the pointer and click **Delete** button above the table.

To generate the report file in HTML format, click **Report**.

Mode shapes and frequencies of natural vibrations in cantilever

This module enables you to calculate mode shapes and frequencies of natural vibrations in cantilever. Method for iteration of subspace is applied to solve the eigenvalue problem.

Input data.

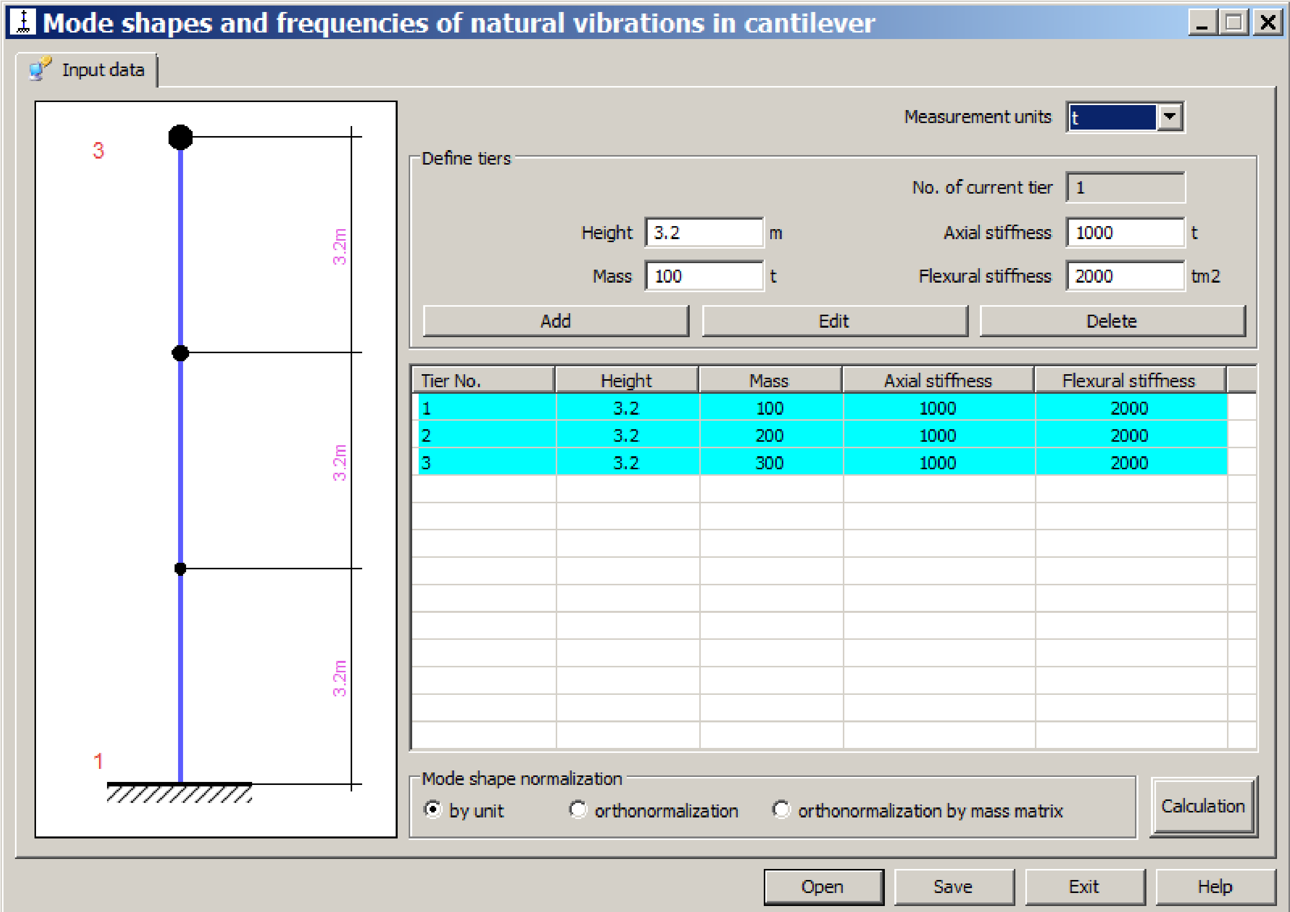


Figure 3.29

To define input data, follow these steps:

– Define measurement units for stiffness and mass - **t** or **kN**.
– In the **Height** box, define the height of the tier in **m**.
– In the **Mass** box, define the mass at the level of tier top.
– In the **Axial stiffness** box, define the axial stiffness for the tier.
– In the **Flexural stiffness** box, define the flexural stiffness for the tier.
– To add defined tier, click **Add**.
– To modify parameters for the current tier, click **Edit**.
– To select a tier, just click appropriate row in the table of tiers.
– To delete the current tier, click **Delete**.
– To define normalization for mode shapes of natural vibrations, select appropriate option.
– When the input data is defined, click **Calculate**.
– Select the problem, if required.

Output data.

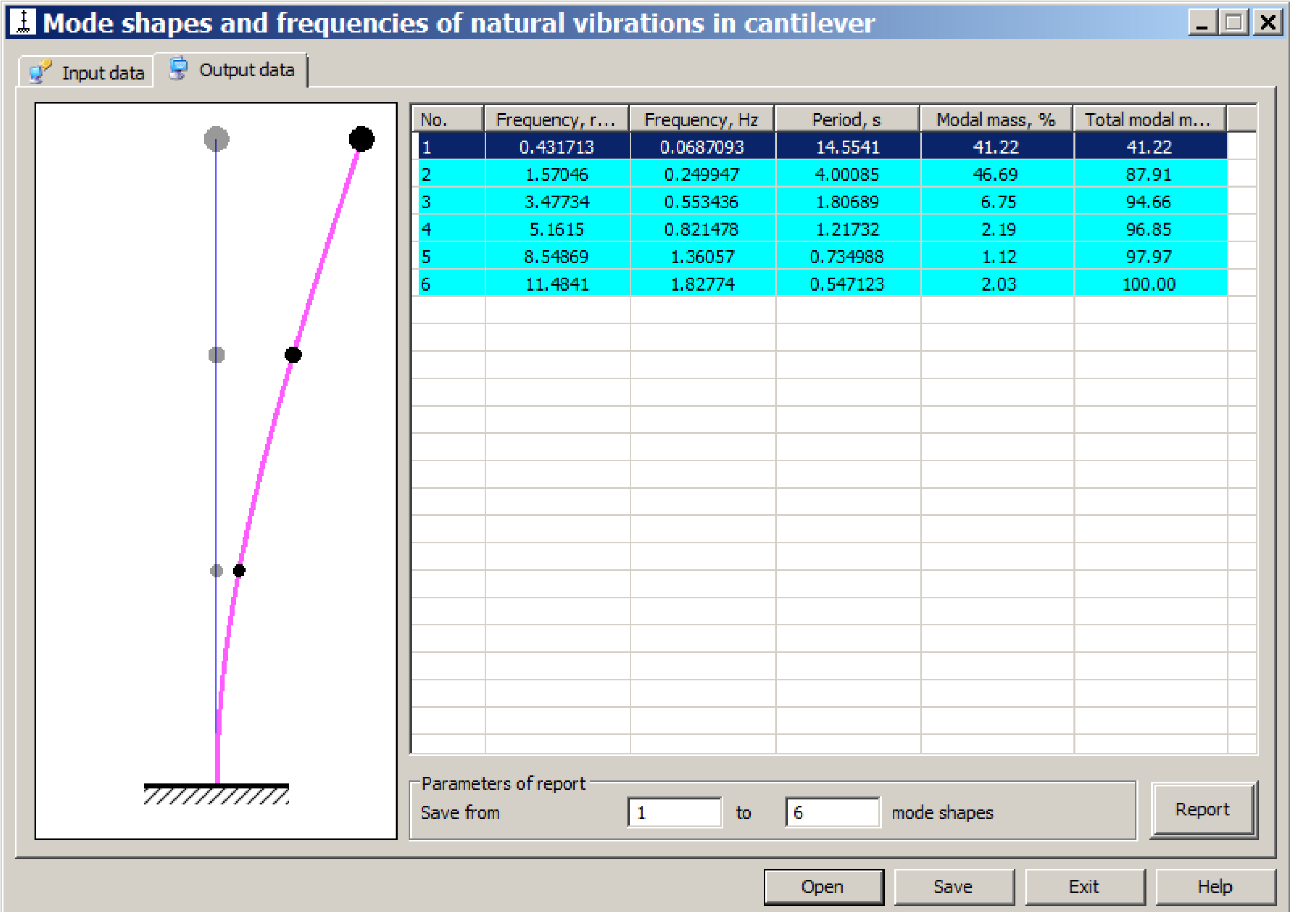


Figure 3.30

– On the **Output data** tab you will find mode shapes of natural vibrations. For every mode shape the following data is presented: angular frequency **ω** (rad/s), technical frequency **f** (Hz), period of vibrations **T** (s), modal mass (%) and total modal mass (%).

– In the **Parameters of report** area it is possible to define mode shapes for which results should be presented.

– To present a report document in HTML format, click **Report**.

Stability factors and buckling modes in cantilever

This module enables you to calculate stability factors and buckling modes. Buckling modes are computed in classical definition for elastic system. It is supposed that displacements are rather small and all external loads applied to the model (as well as internal forces) will increase in proportion to parameter λ. Minimum value of parameter λ (for which stiffness matrix for the system will not be positive-defined any more) is critical; this λ value is called 'stability factor'. Three stability factors and mode shapes appropriate for them are computed.

Input data.

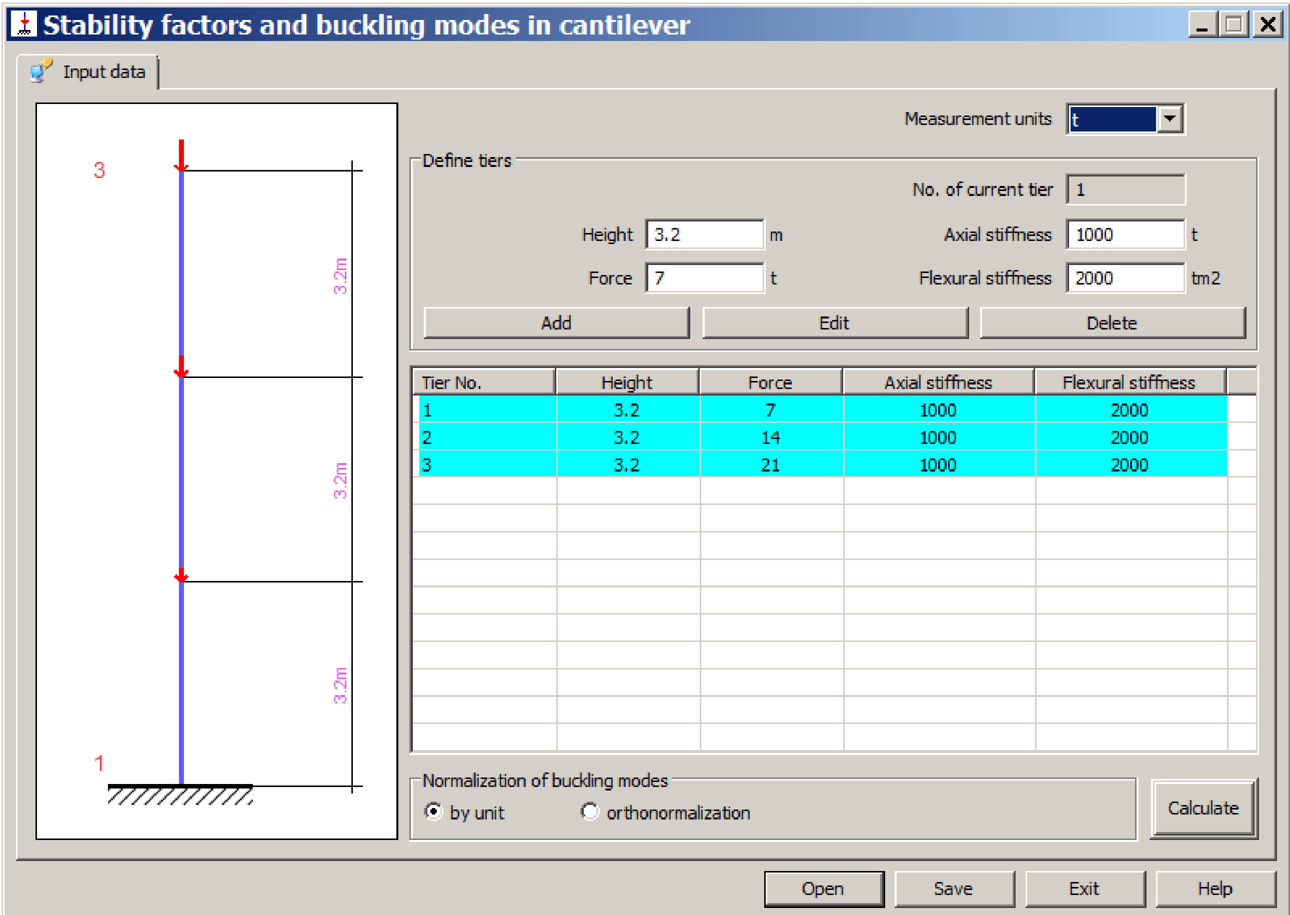


Figure 3.31

To define input data, follow these steps:

– Define measurement units for stiffness and mass - **t** or **kN**.
– In the **Height** box, define the height of the tier in **m**.
– In the **Force** box, define the force at the level of tier top.
– In the **Axial stiffness** box, define the axial stiffness for the tier.
– In the **Flexural stiffness** box, define the flexural stiffness for the tier.
– To add defined tier, click **Add**.
– To modify parameters for the current tier, click **Edit**.
– To select a tier, just click appropriate row in the table of tiers.
– To delete the current tier, click **Delete**.
– To define normalization for mode shapes of natural vibrations, select appropriate option.
– When the input data is defined, click **Calculate**.
– Select the problem, if required.

Output data.

 www.liraland.com

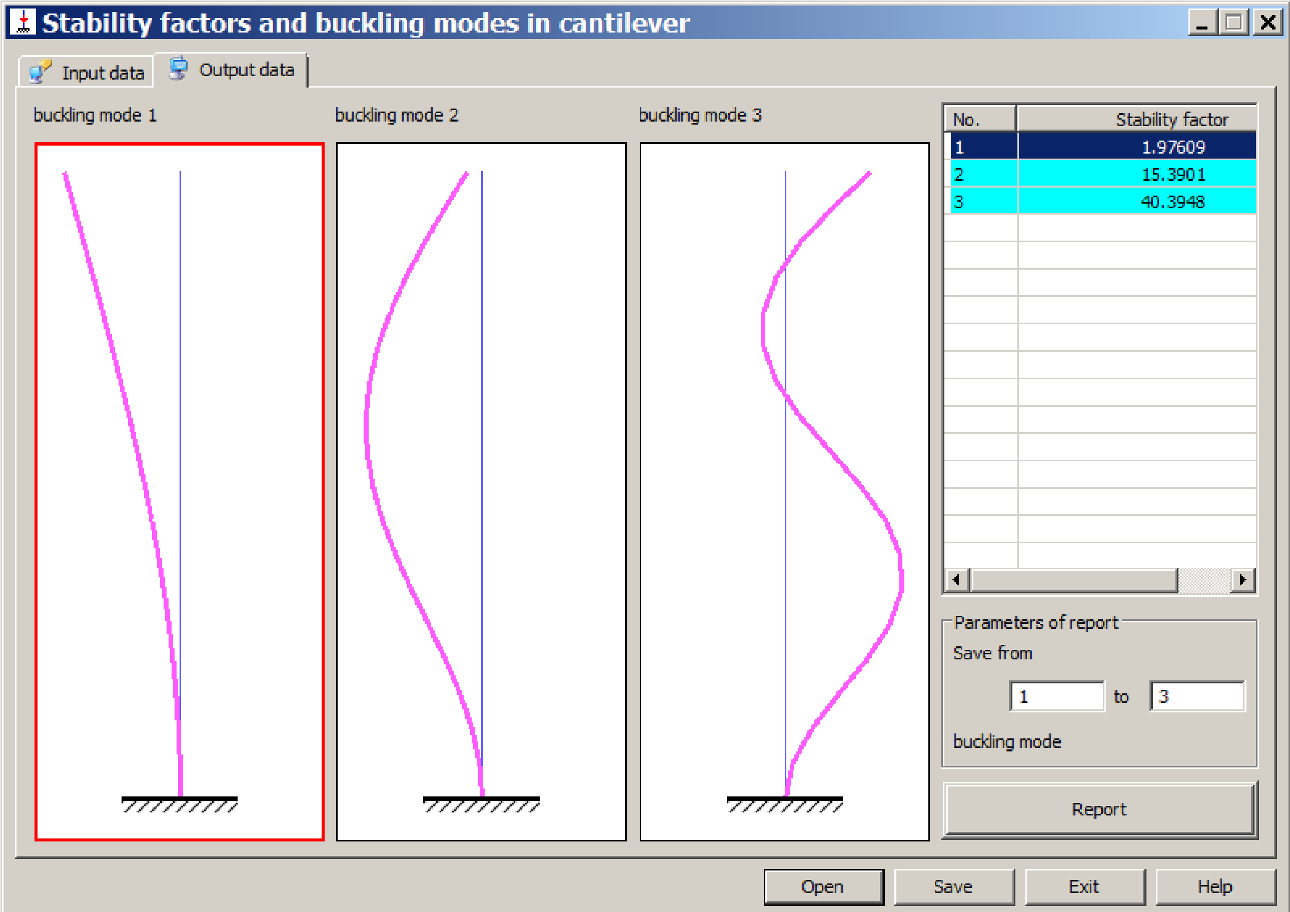


Figure 3.32

– On the **Output data** tab you will find up to three buckling modes. Stability factor is presented in the table for every buckling mode.

– In the **Parameters of report** area it is possible to define buckling modes for which results should be presented.

– To present a report document in HTML format, click **Report**.

Mode shapes and frequencies of natural vibrations in continuous beam

This module enables you to calculate mode shapes and frequencies of natural vibrations in multispan continuous beam (up to five spans with two cantilevers). Sections of spans may differ. It is possible to consider compliance of supports. Method for iteration of subspace is applied to solve the eigenvalue problem.

Input data.

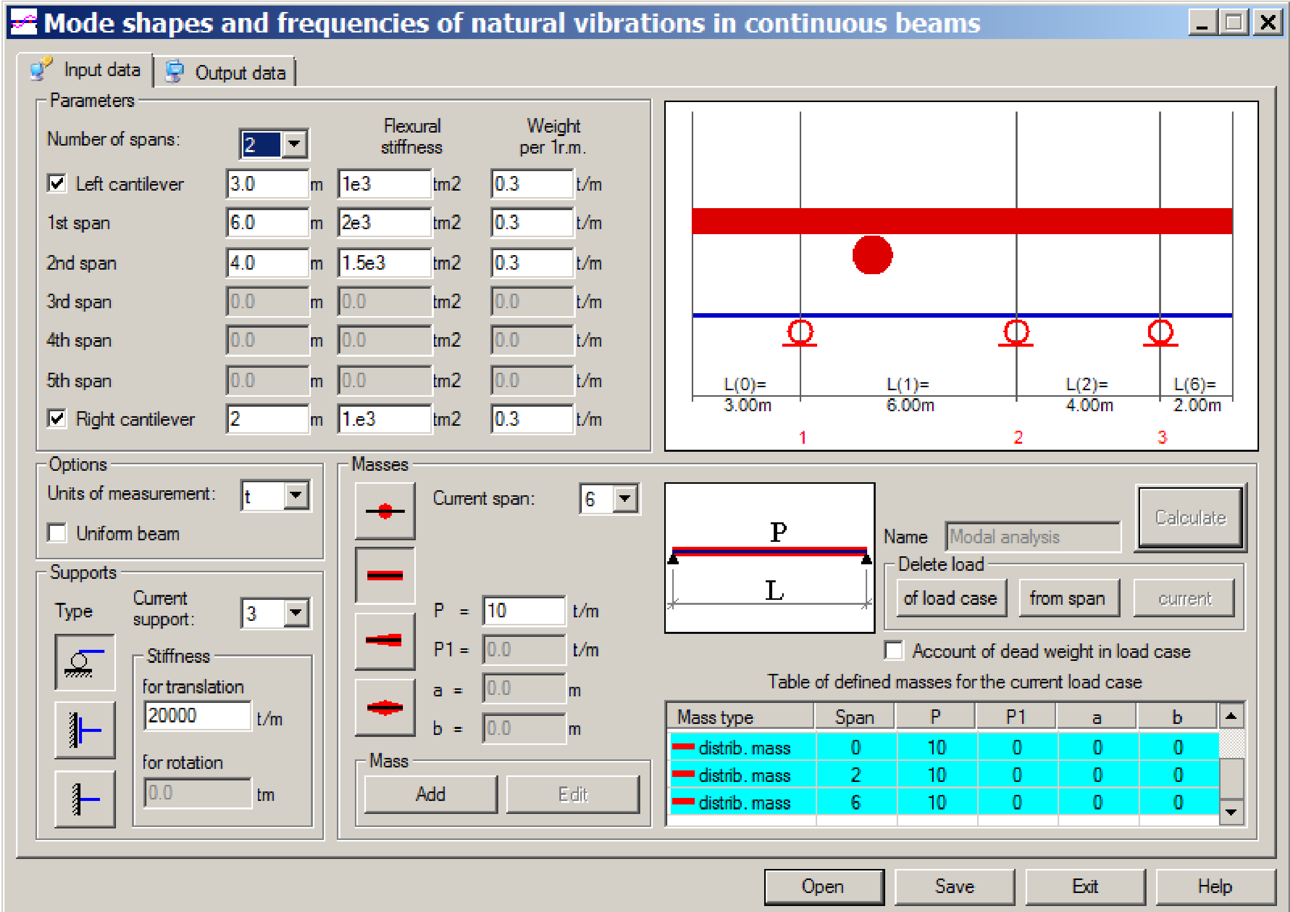


Figure 3.33

To define input data, follow these steps:

– In the **Number of spans** drop-down list, define number of spans and, if required, define left or right cantilevers with appropriate check boxes.

– In the **Options** area, define measurement units for load and stiffness - **t** or **kN**.

– In the appropriate boxes, define the span (cantilever) length in **m** and flexural stiffness in a span. To define the same flexural stiffness for all spans and cantilevers, select the **Uniform beam** check box.

– Define support to which boundary conditions should be applied. To define such support, either click it directly in the schematic presentation or select it in the **Supports** area, in the **Current support** drop-down list.

– Then define the type of boundary condition (just click appropriate icon in the **Supports** area) and it will be displayed in the schematic presentation. If required, in the **Supports / Stiffness** area, define stiffness for translation or rotation.

– In the **Current span** box or in schematic presentation, select spans where the mass should be applied to (it will be displayed in schematic presentation window).

– Select one of the four available mass types (just click appropriate button).

– Define mass values (P, P1), distance to beginning/end of mass application (a,b) (if required) and click **Add**.

– To modify parameters for the mass, select the mass in the table, modify its parameters in the appropriate boxes and click **Edit**.

– To delete mass, select the mass in the table. To delete current mass, in the **Delete load** area, click **current**. To delete all masses from the current span, in the **Delete load** area, click **from span**.
– To consider the dead weight in the current load case, select the **Account of dead weight in load case** check box.
– When the input data is defined, click **Calculate**.
– Select the problem, if required.

Output data.

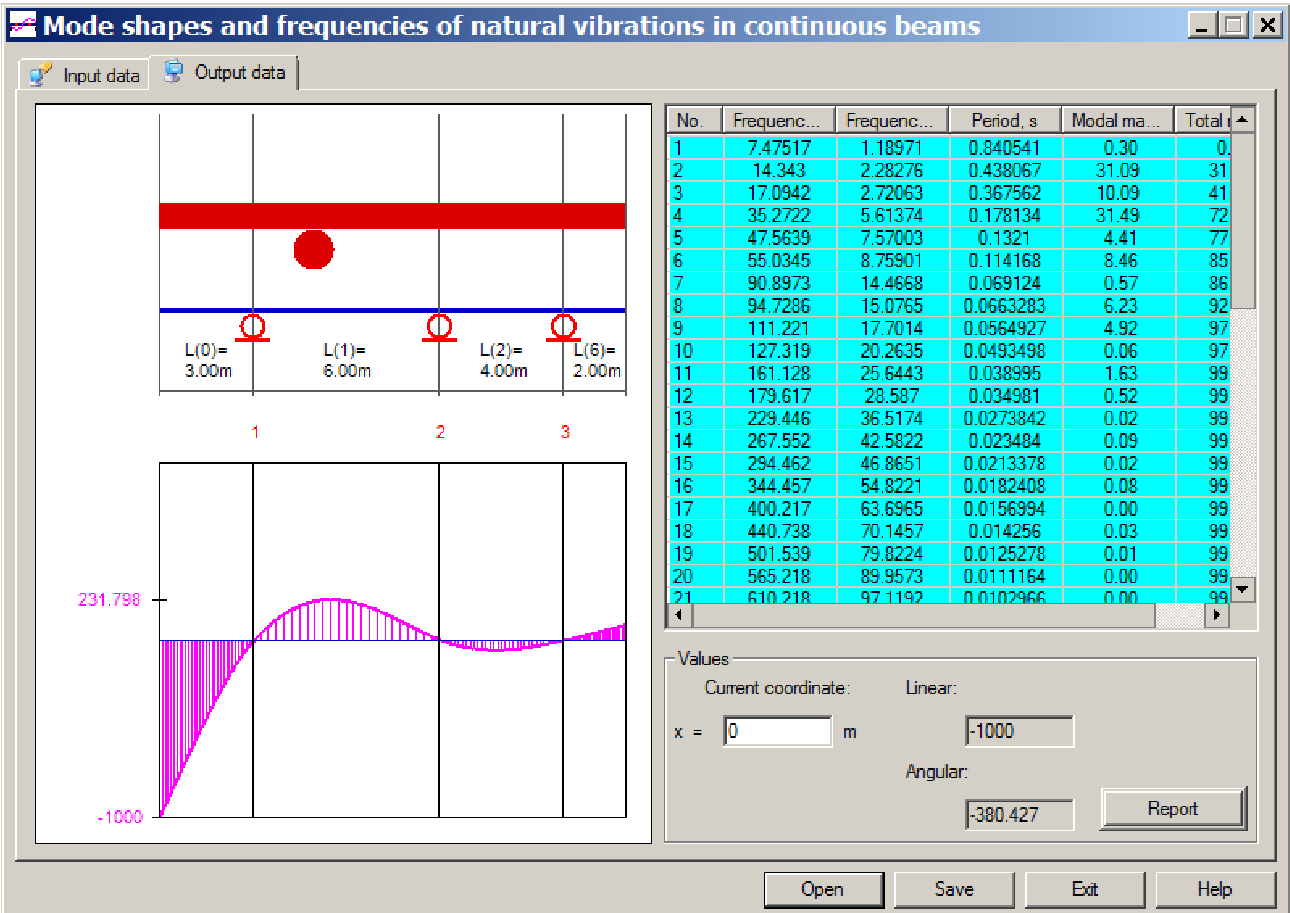


Figure 3.34

– On the **Output data** tab you will find mode shapes of natural vibrations. For every mode shape the following data is presented: angular frequency **ω** (rad/s), technical frequency **f** (Hz), period of vibrations **T** (s), modal mass (%) and total modal mass (%).

– To present a report document in HTML format, click **Report**.

Cable and string

Cable and string

This module enables you to analyse cable and strings according to requirements presented in the Design-theoretical reference book for structural designer.

There are five types for calculation.

For cables:
– calculation type 1 – by specified length of blank S_0;
– calculation type 2 – by dip f_q from dead weight q at mid-span;
– calculation type 3 – by dip f_K from arbitrary load at arbitrary point K .

For strings:
– calculation type 4 – string with arbitrary load;
– calculation type 5 – string prestressed by force N, arbitrary load.

The following load patterns are available: concentrated load, different types of distributed loads, as well as their combinations.

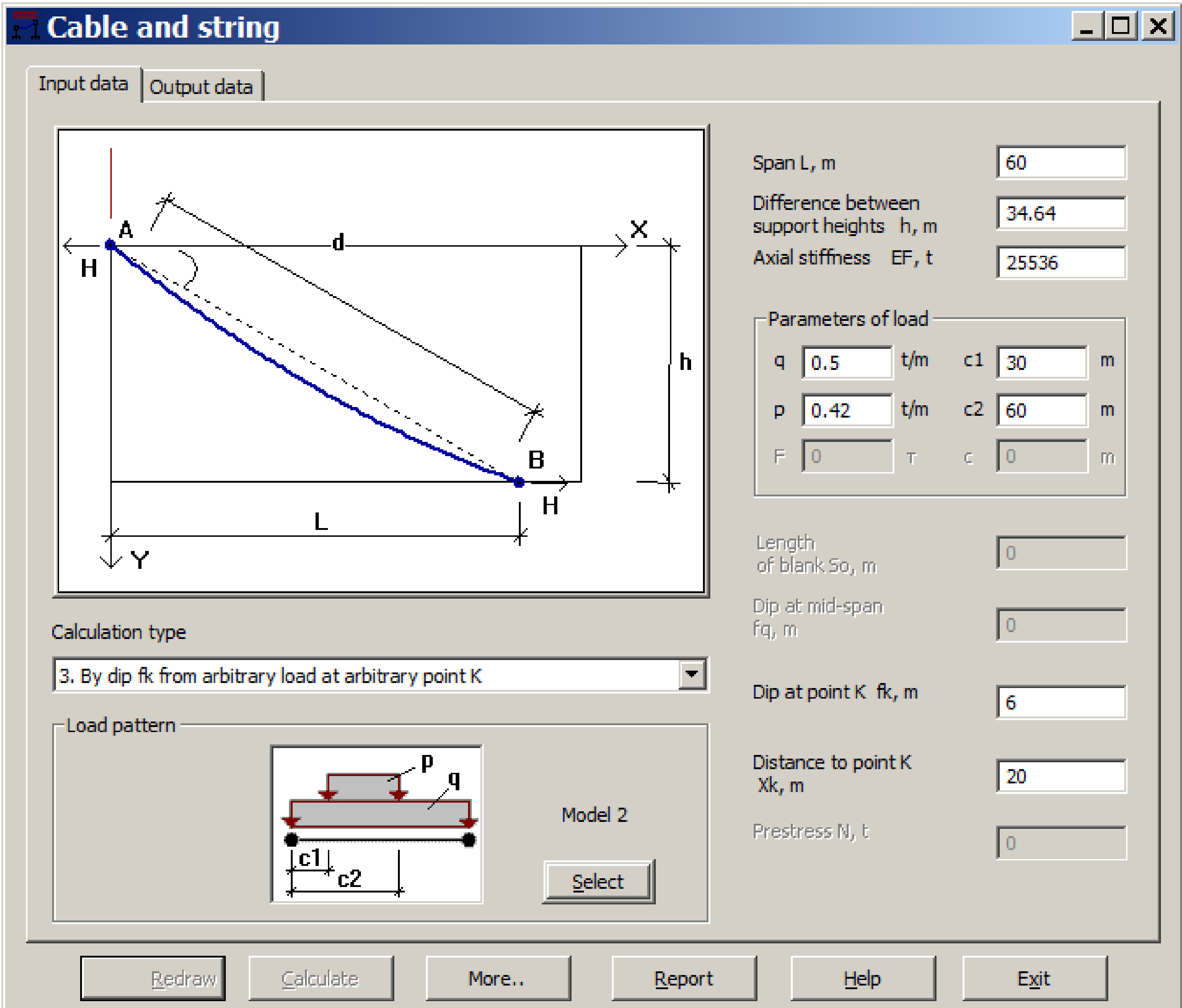


Figure 3.35

Calculation type 1.

Cable. By length of blank $\boldsymbol{S_0} > \boldsymbol{d}$ (***d*** – length of chord) and allowed load pattern, determine thrust, form of equilibrium and axial force.

Calculation type 2.

Cable. By dip $\boldsymbol{f_q}$ at midspan from the dead weight and allowed load pattern, determine initial dip, thrust, length of blank, form of equilibrium and axial force.

Calculation type 3.

Cable. By dip $\boldsymbol{f_K}$ at arbitrary point ***K*** with arbitrary load pattern, determine form of equilibrium, initial thrust, length of blank and axial force.

Calculation type 4.

String. Length of blank is equal to length of chord $\boldsymbol{S_0 = d}$. Determine thrust and force in string for the specified load pattern.

Calculation type 5.

String. Length of blank is equal to length of chord $S_0 = d$. Determine thrust and force in prestressed string for the specified load pattern.

Design model of the cable is located in the X and Y axes. X-axis is directed left-right. Y-axis is directed top-down.

Important.
If calculation type 3 is specified, then dip of the cable at point ***K*** is calculated automatically for the specified data. If the specified dip f_K is less than dip of the same cable at point ***K***, then calculation procedure is terminated and the program displays the following message: *'For selected type of calculation with the specified parameters of cable and load pattern, specified value of dip at point K should be not less than [Yk] m'*. In this case, new value will be displayed in the box for dip value; it is suggested for calculation procedure.

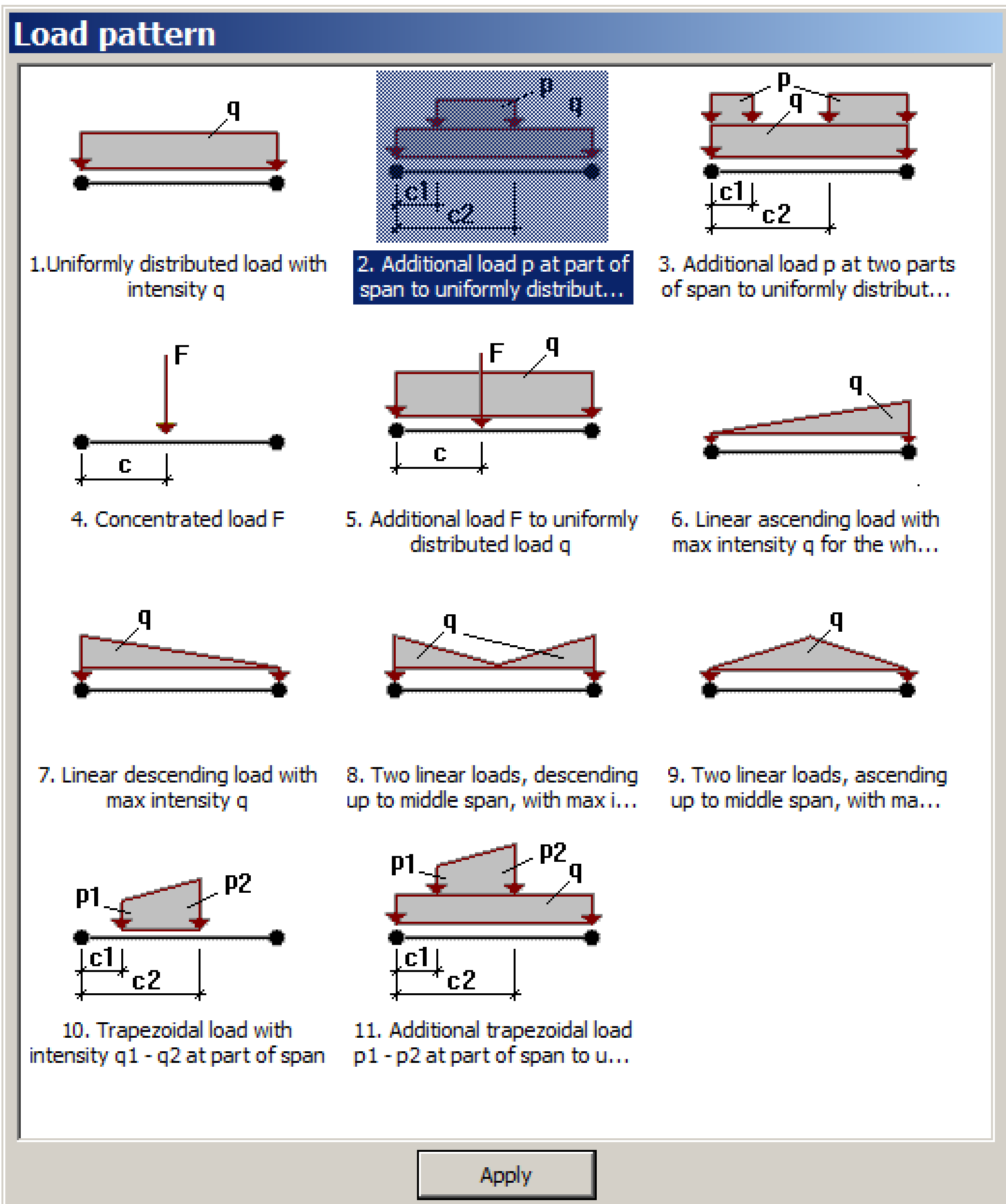


Figure 3.36

The following load patterns are realized:

1 – uniformly distributed load with intensity ***q***;
2 – additional load ***p*** at part of span to uniformly distributed load ***q***;
3 – additional load ***p*** at two parts of span to uniformly distributed load ***q*** ;
4 – concentrated load ***F***;
5 – additional load ***F*** to uniformly distributed load ***q***;
6 – linear ascending load with max intensity ***q*** for the whole span;
7 – linear descending load with max intensity ***q***;
8 – two linear loads, descending up to middle span, with max intensity ***q***;
9 – two linear loads, ascending up to middle span, with max intensity ***q***.
10 – trapezoidal load with intensity ***q1-q2*** at part of span with length ***b*** , distance to the beginning c_1.
11 – additional trapezoidal load with intensity ***p1-p2*** at part of span with length ***b*** and distance to the beginning c_1 to uniformly distributed load ***q*** .

When you start the program, the dialog box will be displayed with the **Input data** tab where you could define input data for calculation.

Cable and string. Input data

The input data depends on the type of calculation you select.

Required parameters:

L – span length (distance between supports along the X-axis);
h – difference between support heights (distance between supports along the Y-axis);
EF – axial stiffness.

To present the cable on schematic presentation zone according to specified parameters, click **Redraw**.

From the **Calculation** box, select appropriate type of calculation procedure.

Boxes for additional parameters become active according to type of calculation you select.

Additional parameters depending on the type of calculation:

Calculation type 1 – length of blank S_0. Available load patterns: 1, 2, 3, 5,11.
Calculation type 2 – dip f_q at midspan. Available load patterns: 1, 2, 3, 5,11.
Calculation type 3 – dip f_K at arbitrary point ***K*** and distance X_K to the left support. All load patterns are allowed.
Calculation type 4 – additional data is not required. All load patterns are allowed.
Calculation type 5 – prestress ***N***. All load patterns are allowed.

To select load pattern, click **Select** button. You will find load patterns available for certain type of calculation.

To start calculation procedure, click **Calculate**.

Cable and string. Output data

When calculation is complete, the **Output data** tab and the **Report** button become available.

The output data contains:
1. Projection of cable (string) onto horizontal axis; load pattern is indicated.
2. Diagram of dips for the cable (string) ***f = Mb(X) / H***.
3. Form of equilibrium for the cable (string) ***Y = f(X)***.
4. Diagram of tg values of slope angles for the cable (string).
5. Diagram of beam moment and shear forces onto horizontal projection.
6. Diagram of axial force.

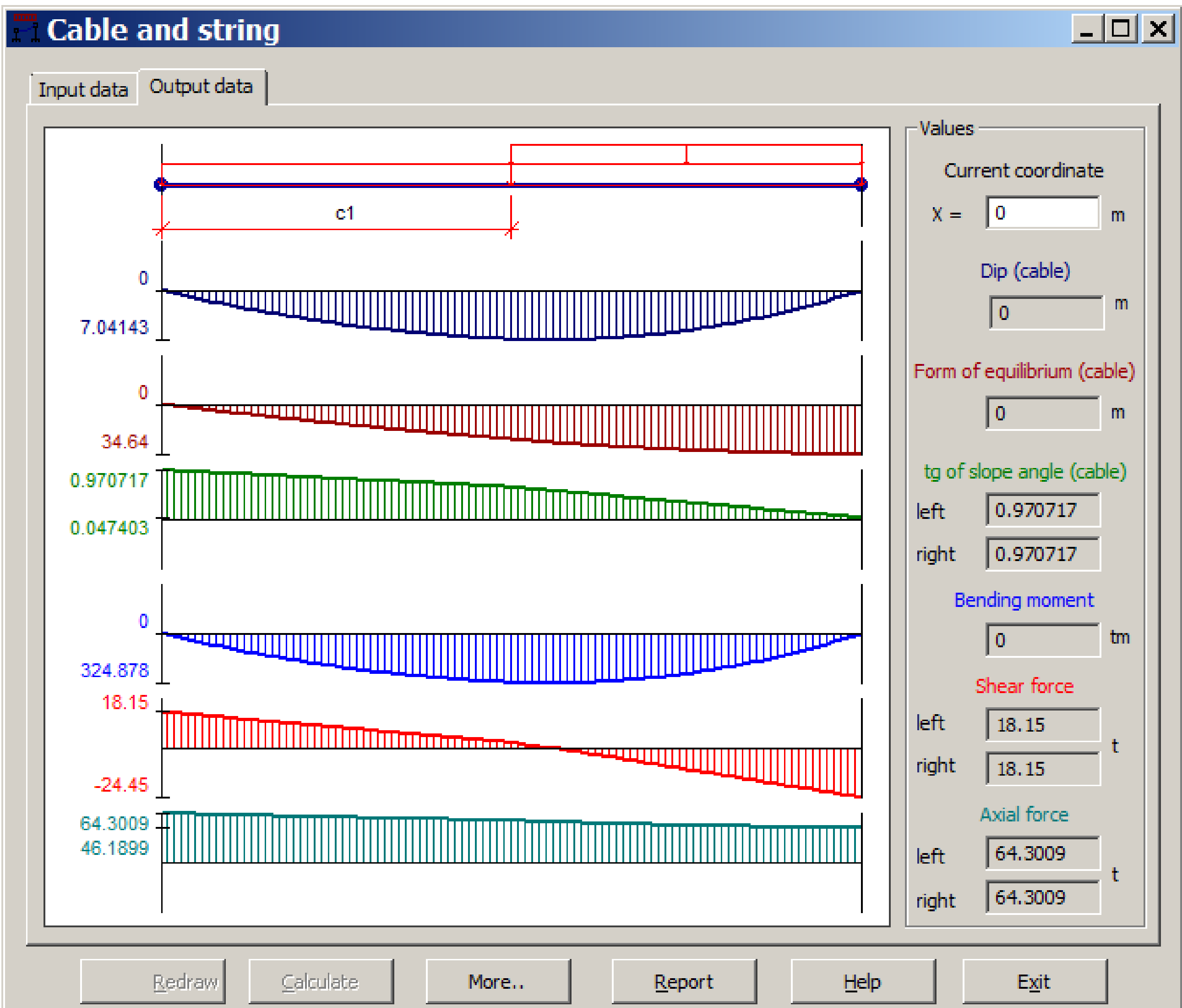


Figure 3.37

When you define **Current coordinate X**, ordinate values for the above-mentioned diagrams will be displayed in appropriate boxes.

When you drag the pointer along the image, you will also see ordinate values for the above-mentioned diagrams. Colour for the diagram corresponds to the colour for the text at appropriate boxes.

To present the text file with temporary parameters for calculation and comments on them, click **More** button.
If calculation was terminated by any reason, then temporary parameters calculated up to this moment are saved to this file with a message about the cause of such termination.

To generate report file (HTML) with input and output data, click **Report**. It is possible to save the report file for further work.

Cable and string. Notation

Notation for input data

For geometry:
f_q – dip at midspan from the dead weight q;
f_K – dip at arbitrary point K;
X_K – distance from left support to point K .

For load:
q – dead weight or load uniformly distributed along the whole span or max value of triangular load;
$q1$ – the first (from left support) ordinate of trapezoidal load at part of a span;
$q2$ – the second (from left support) ordinate of trapezoidal load at part of a span;
p – additional uniformly distributed load at part of a span;
$p1$ – the first (from left support) ordinate of additional trapezoidal load at part of a span;
$p2$ – the second (from left support) ordinate of additional trapezoidal load at part of a span;
F – force in span;
c_1 – distance from the left support to the first ordinate of trapezoidal load;
c_2 – distance from the left support to the second ordinate of trapezoidal load;
c – distance from the left support to the force.

Notation for output data and temporary parameters

H_0 – thrust of nonstretchable cable;
S_0 – length of blank;
H – thrust of elastic cable;
S – length of loaded elastic cable;
d – length of chord;
D – parameter of load;

φ – slope angle between cable and the X-axis;
R1 – reaction at left support;
R2 – reaction at right support;
Mb(Xk) – beam moment at point ***Xk***;
Eps – relative (along the X-axis) distance from the left support to resultant;
Yc – dip at point where resultant is applied;
Yk – dip at point ***K***.
nju – ratio of dip (at point where resultant is applied) to span length;
A, B – coefficients in cubic equation;
v, w, w1, J, J1, t1, t2, z – temporary parameters for solving cubic equation;
r1, r2, k – temporary parameters.

Influence lines in continuous beam

The module enables you to generate influence lines of displacements, rotation angles, bending moments and shear forces from moving loads in multispan continuous beam (up to five spans with two cantilevers).

Input data.

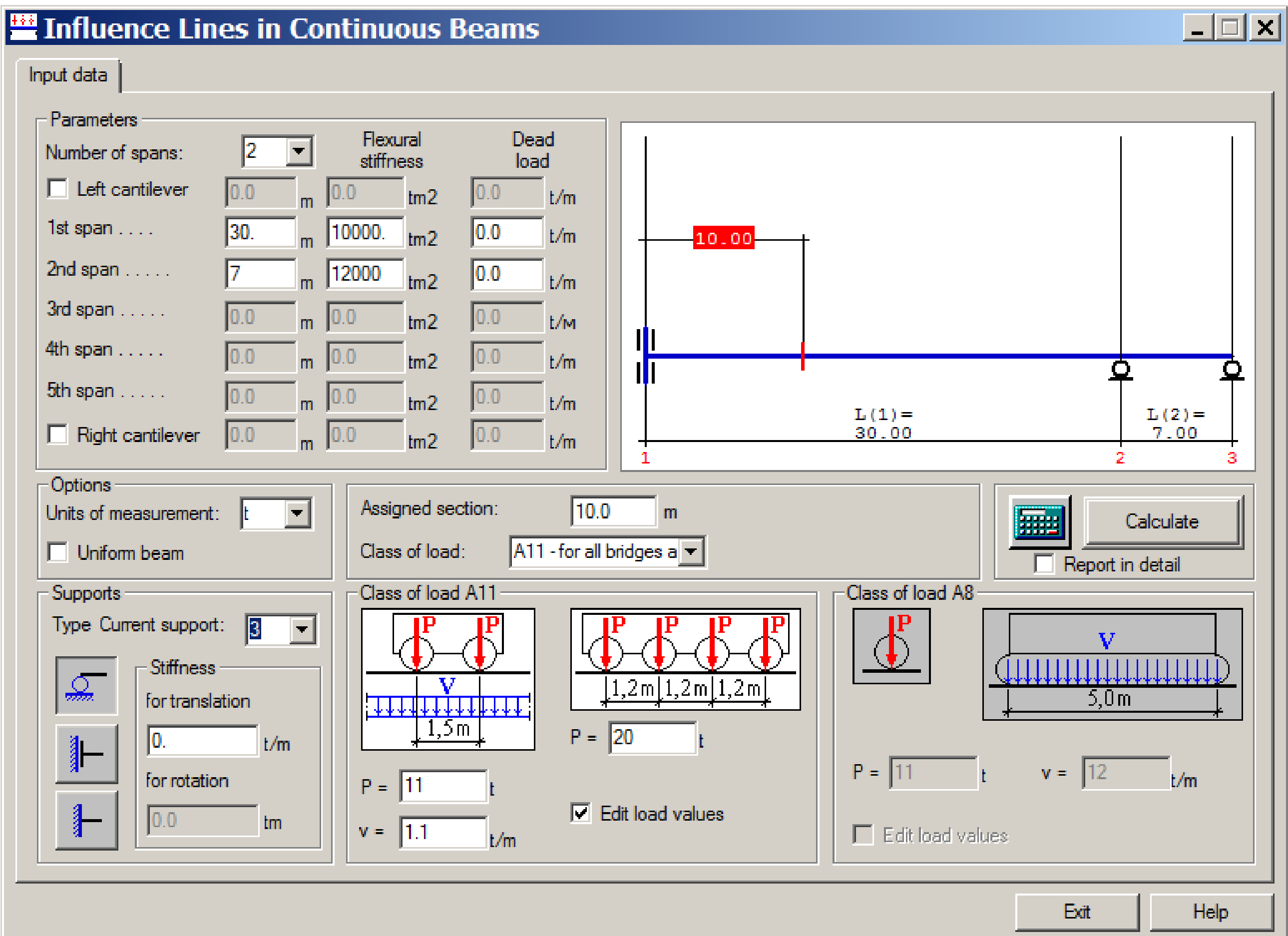


Figure 3.38

 www.liraland.com

To define input data, follow these steps:

– In the **Number of spans** drop-down list, define number of spans and, if required, define left or right cantilevers with appropriate check boxes.

– In the **Options** area, define measurement units for load and stiffness - **t** or **kN**.

– In the appropriate boxes, define the span length in m, flexural stiffness in a span, and the dead load. To define the same flexural stiffness for all spans and cantilevers, select the **Uniform beam** check box.

– Define support to which boundary conditions should be applied. To define such support, either click it directly in the schematic presentation or select it in the **Supports** area, in the **Current support** drop-down list. Then define the type of boundary condition (just click appropriate icon) and it will be displayed in the schematic presentation. If required, define stiffness for translation or rotation.

– In the **Assigned section** box, define ordinate of beam for which influence lines should be generated.

– Define the **Class of load** in the appropriate box. You could also define your own moving load that differ from normative values of load classes A11 and A8 (SNIP 2.05.03-84 'Bridges and pipes'). To do this, select the **Edit load values** check box and input appropriate values.

– When input data is defined, click **Calculate**.

Output data.

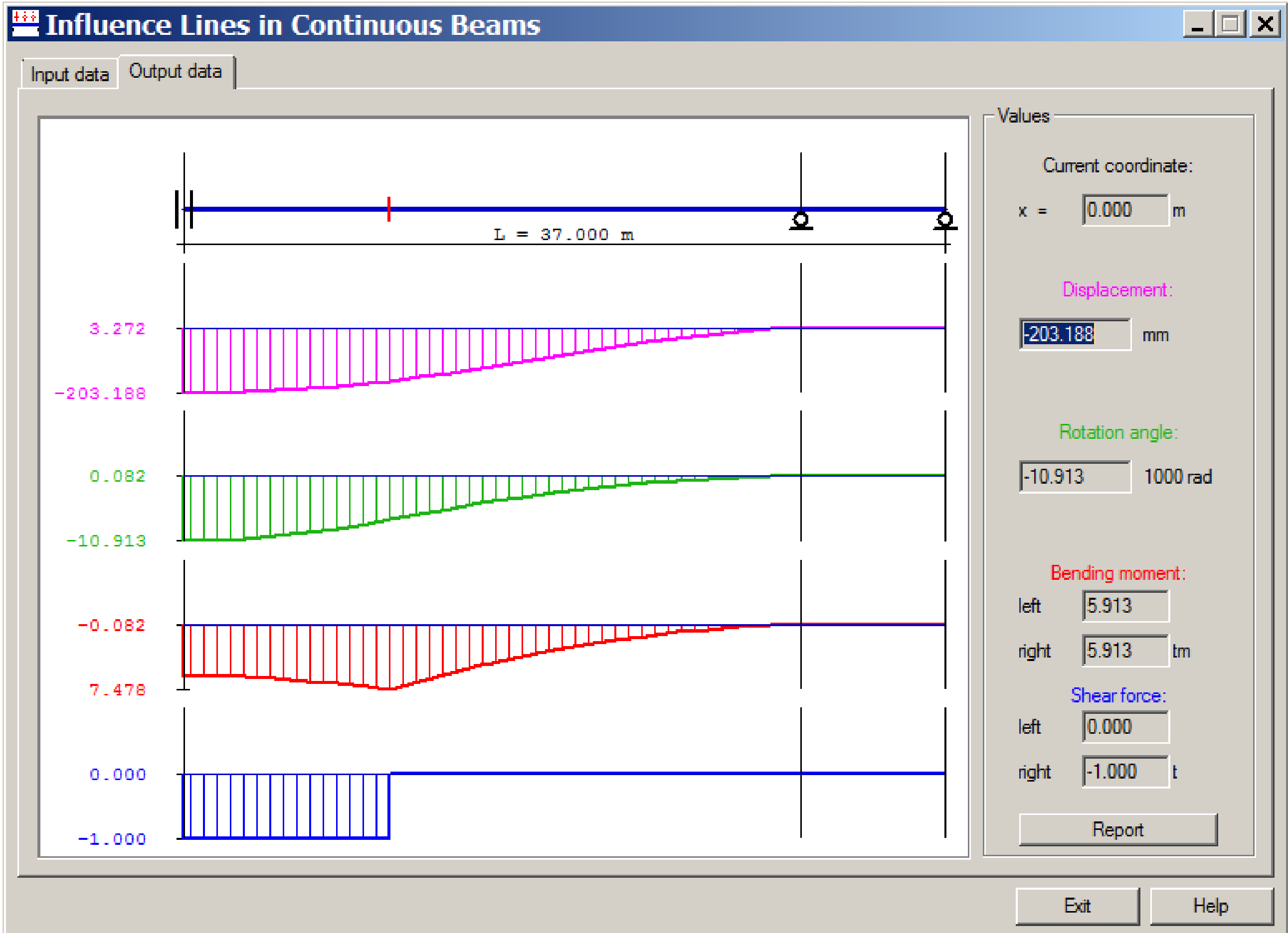


Figure 3.39

On the **Output data** tab you will see influence lines of displacements, rotation angles, bending moments and shear forces. To review values of influence lines at any point of beam, just drag the beam ordinate with the pointer. The values will be displayed at the appropriate boxes.

To present a report document in HTML format, click **Report**.

To generate a report with detailed tables of results, on the **Input data** tab, select the **Report in detail** check box.

Steel Structures

Steel structures

This chapter contains modules for analysis of elements and joints of steel structures as well as reference modules necessary for analysis and design of steel structures.

Steel table
Analysis of steel elements
Principal and equivalent stresses in steel structures
Effective lengths of steel structure elements
Parametric joints of steel structures
Analysis of welds
Bolted connections
Cold-formed shapes

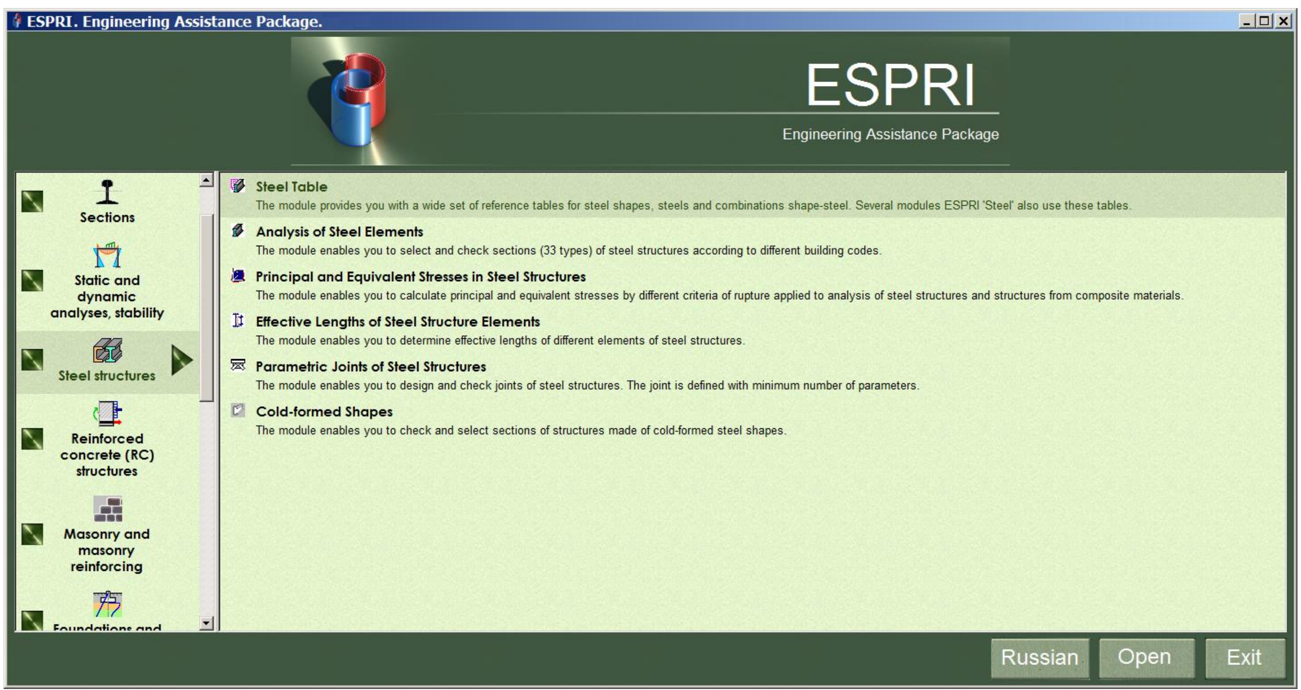


Figure 4.1

Steel table

This module is a version of SRS-SAPR (Steel Rolled Shapes) module in LIRA-SAPR program but it is adapted for the work in ESPRI environment.

The module enables you to view and edit existing steel tables and create new ones.

The module is mentioned to review and edit available steel tables and to create new ones.

Steel tables contain: table of *profiles*, table of *materials* and table of *combinations*. Dimensions and geometric properties of profiles are presented in the table of profiles. Physical and mechanical properties of materials that profiles are made of are presented in the table of materials.
Table of combinations presents data about compatibility of every combination profile-material, that is, whether the profile may be made of certain material, and properties of such combination.

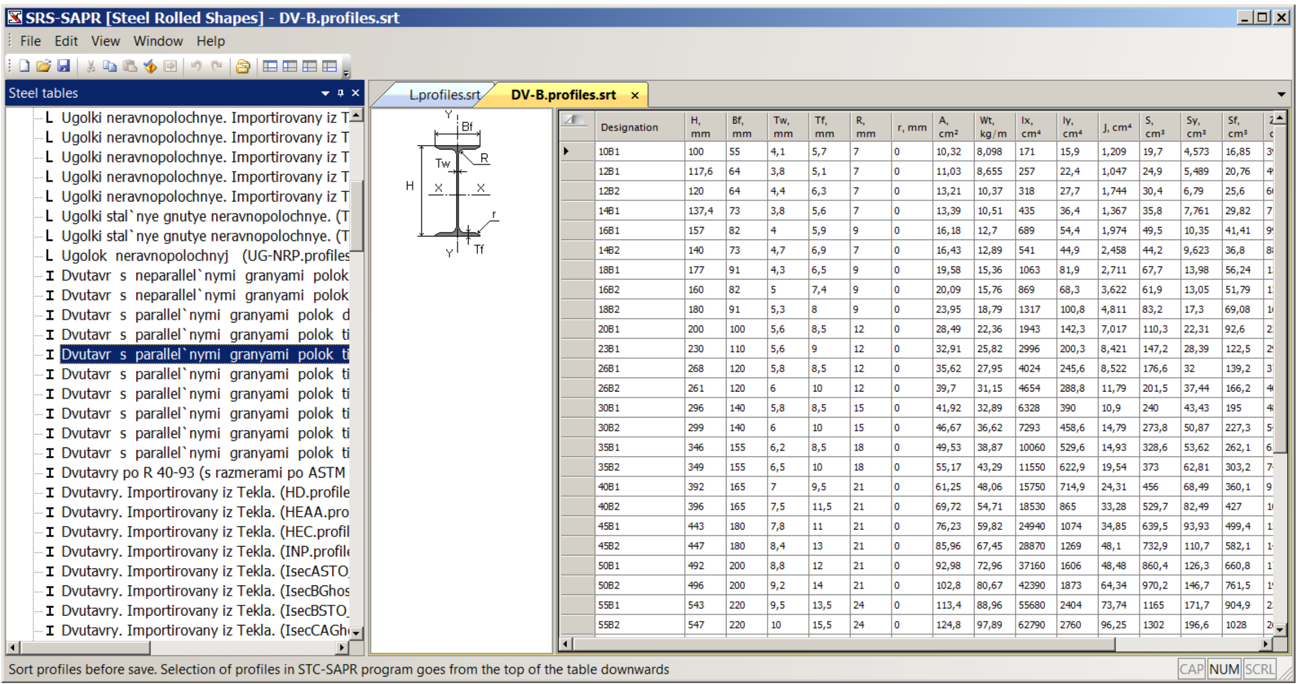


Figure 4.2

Main options

Preview steel table
In the list of steel tables, double-click appropriate item. Table of profiles presented in the steel table will be opened. To preview the table of materials, on the VIEW menu, click **Materials table**. To display the general data, on the VIEW menu, click **Show annotations**.
Note: If the list of steel tables is hidden, to present the list, on the VIEW menu, point to **Toolbars and docking windows** and click **Steel tables** (button on the toolbar). To open file, you could also use the **Open** command on the FILE menu.

Edit existing steel table
Open the file for preview. On the EDIT menu, click **Edit mode**. Now you could edit the table. When you edit the table, it is possible to type the values or use copy & paste commands: you could prepare fragment of the table in the table editor like MS Excel, then copy it to the Clipboard and paste to appropriate steel table in the SRS-SAPR. To do this, on the EDIT menu, click **Paste**. You could also copy selected fragment from SRS-SAPR table (**Edit / Copy**) and paste it to the table editor.

 www.liraland.com

Open folder that contains files of steel tables
To open folder that contains steel tables, on the FILE menu, click **Open folder**. Then double-click appropriate steel table in the list.

Create new steel table
To create new file of steel table, on the FILE menu, point to **New** and click the type of profile for which the steel table is created. Then fill in the table of profiles and table of materials. For ropes, it is necessary to fill in the table of combinations as well.

Add new profile to the table of profiles
Open the table of profiles as described above in the 'Edit existing steel table' or 'Create new steel table' sections. Select the last row in the table (the last row is denoted with an asterisk *). Define the name for the new profile in the **Designation** column and profile dimensions in the appropriate cells according to schematic presentation. To determine geometric properties of these profiles according to their dimensions, on the EDIT menu, click **Autocomplete now** (or click button on the toolbar or click F12). If geometric properties of a certain profile are known, you could define them in the table manually. To check them, on the EDIT menu, click **Validate now** (button on the toolbar). If there are any mistakes in values (program check), symbol will be displayed in the cells. If the value is correct, just ignore the warning. The additional check option is the option to generate the sketch of cross section of profile to scale. To generate the sketch, on the VIEW menu, click **Show sketch**.

Notes:
1. To cancel or customize check carried out by the program, on the EDIT menu, click **Validation options**.
2. Before the new steel table is saved, it is recommended to sort the table of profiles by area (just click the title of appropriate column).

Add new material to the table of materials
Open the table of profiles as described above in the 'Edit existing steel table' or 'Create new steel table' sections. To open materials table, on the VIEW menu, click **Materials table**. Select the last row in the table (the last row is denoted with an asterisk *). Type the name of new steel to the **Designation** cell, its GOST or TU – to **GOST** cell. Define design and normative strength of new steel in the **Ry1, Ru1, Ryn1, Run1** cells and define thickness of steel **Tmin1, Tmax1**, for which steel strength values are actual. If for another range of steel thicknesses it is necessary to define another strength values, fill in **Ry2, Ru2, Ryn2, Run2** and **Tmin2, Tmax2** cells, etc. The procedure for check of defined data is the same as for the table of profiles.

Notes:
1. To customize units of measurement, on the VIEW menu, click **Units**.

2. Table of combinations is filled in automatically as you define other data. That's why it should be edited manually only in case you edit the steel table of ropes.

Important. Several modules ESPRI 'Steel' also use these steel tables.

Analysis of steel elements

This module represents STC-SAPR module adapted to work in ESPRI environment.
The module enables you to select and check sections of steel elements according to different building codes.
The program supports the following normative documents:
– SNIP II–23–81* 'Steel structures',
– Manual on design of steel structures (to SNIP II–23–81* 'Steel structures'),
– Eurocode 3 'Design of steel structures'.

The work with the program is illustrated by an example: analysis of steel beam with I-section shape loaded in two planes.

Input data

On the FILE menu, point to **New** and click **Element** (button on the toolbar).
In the **Units of measurement** dialog box, define appropriate units for input data and output data and click **OK**.

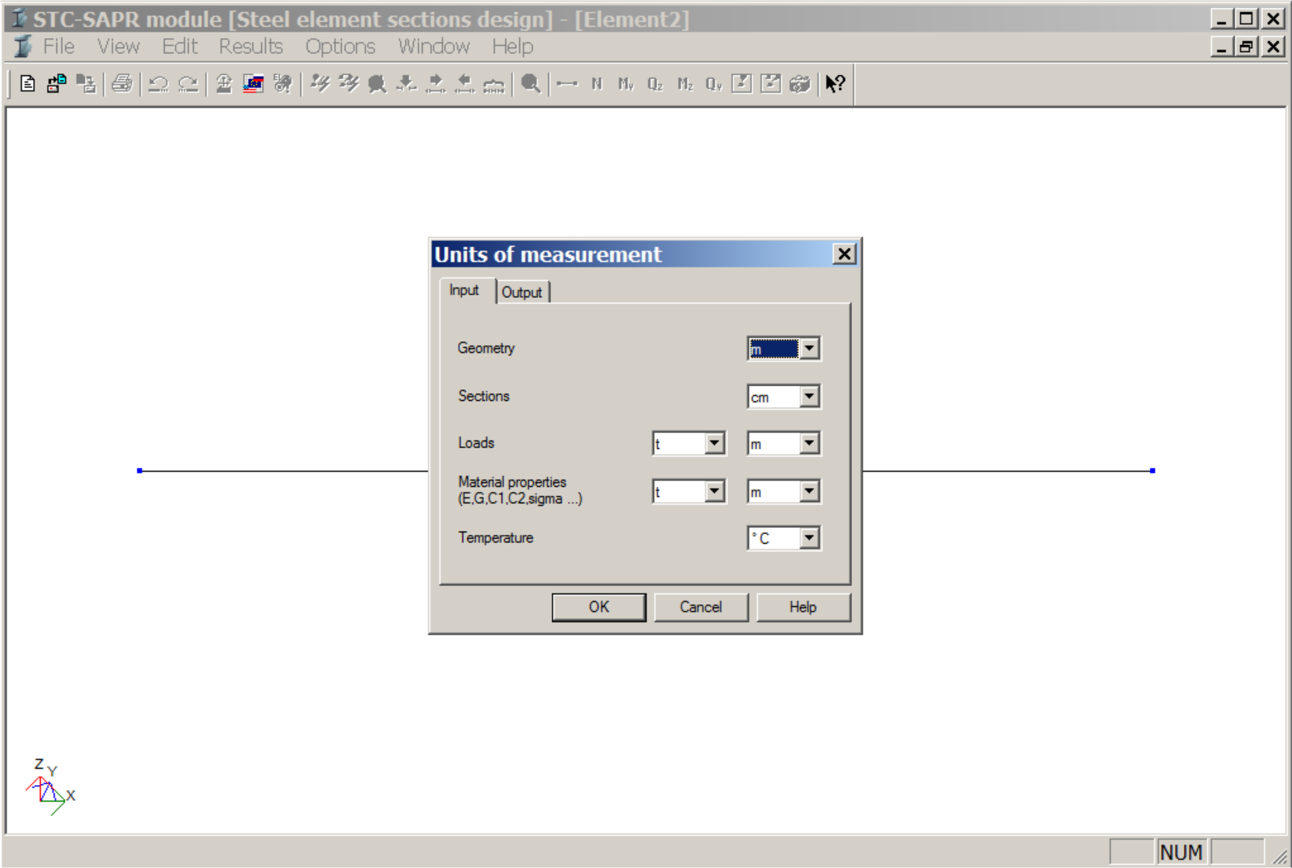


Figure 4.3 a)

In the **Problem info** dialog box, define the following data:
– problem code (comment) (not necessary);
– number of element (by default, it will be the file name);
– length of element – for this problem, define 6 m;
– element location – define either vertical or horizontal (specifies how the diagrams should be located in the report).
For flexure elements, it is possible to select and check according to serviceability limit states (SLS) by enveloping diagrams of DCF.
Select the **Calculate deflection by envelope diagrams DCF** check box and then define appropriate **Fixities at beam ends** (Y1, Z1 – translation along appropriate local axes of bar are not allowed, UY1, UZ1 – rotations about appropriate local axes of element are not allowed).

In the present case, we define beam with hinge support at the left end in both directions (Y1, Z1) and at the right end - fixed in its own plane and with hinge support out-of-plane (Y1, Z1, UY1).

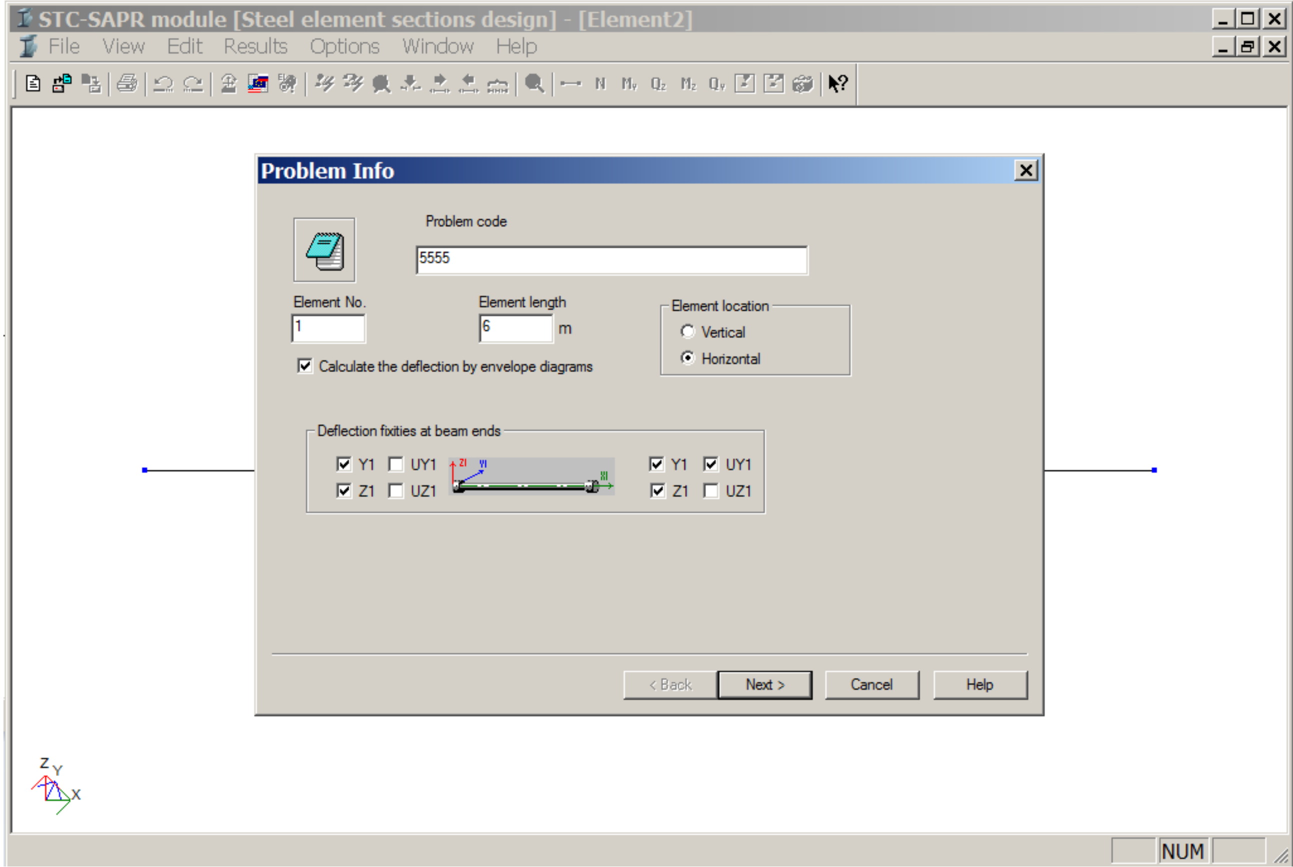
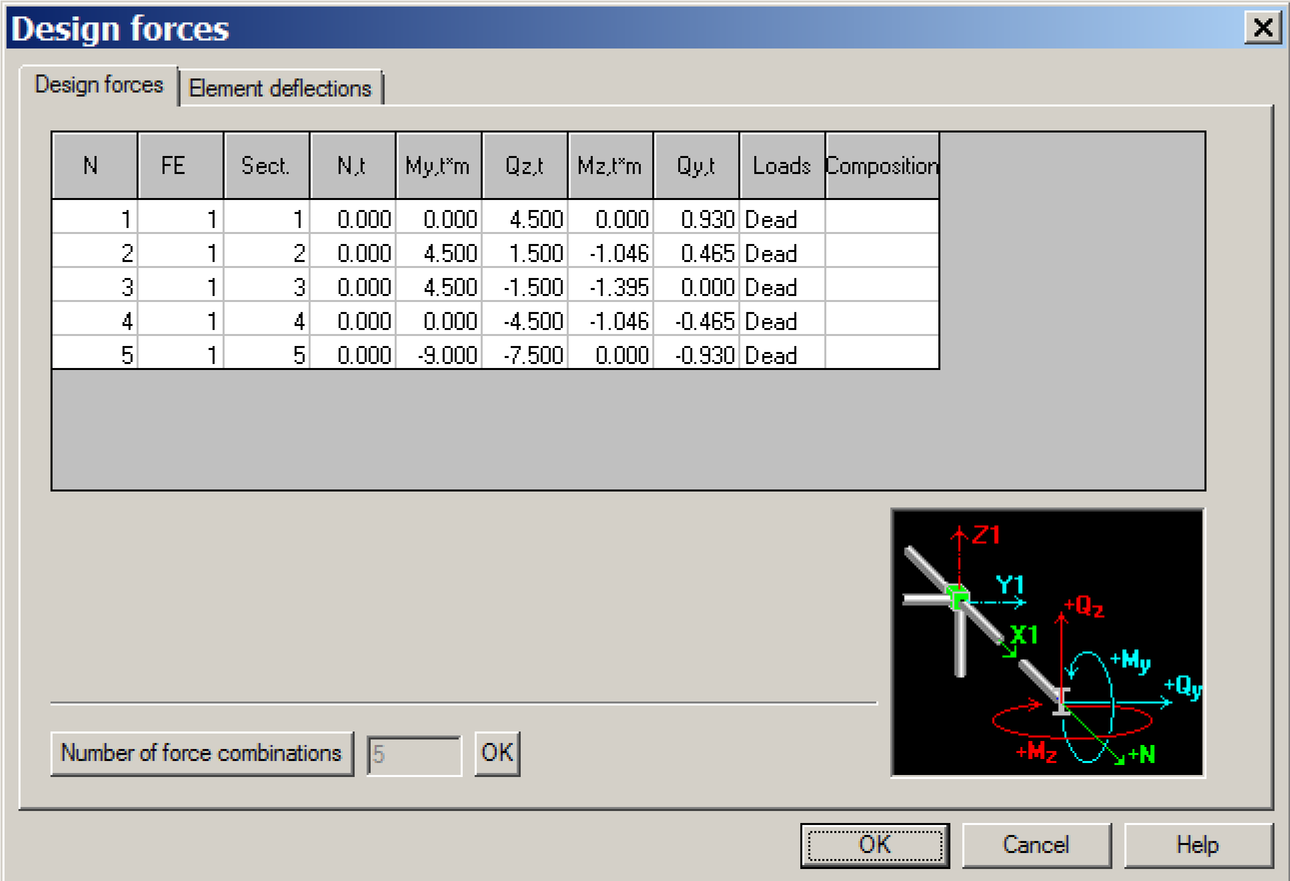


Figure 4.3 b)

Click **Next**.
In the **Design forces** dialog box, define **Number of force combinations** as equal to 5 and click **OK**.
In the table fill in the column of section numbers (Sect.) with sequence numbers 1, 2, 3, 4, 5 and for every section define appropriate forces in the table (for beam – bending moments and shear forces) according to sign convention presented in the right bottom corner of the dialog box.
When you fill in values in the **Design forces** table, click **OK**.
In the **Select section type** dialog box, select the type of section - **I-section**. To select the section type, either double-click its icon or select the section with the pointer and click **OK**.



N	FE	Sect.	N,t	My,t*m	Qz,t	Mz,t*m	Qy,t	Loads	Composition
1	1	1	0.000	0.000	4.500	0.000	0.930	Dead	
2	1	2	0.000	4.500	1.500	-1.046	0.465	Dead	
3	1	3	0.000	4.500	-1.500	-1.395	0.000	Dead	
4	1	4	0.000	0.000	-4.500	-1.046	-0.465	Dead	
5	1	5	0.000	-9.000	-7.500	0.000	-0.930	Dead	

Figure 4.4

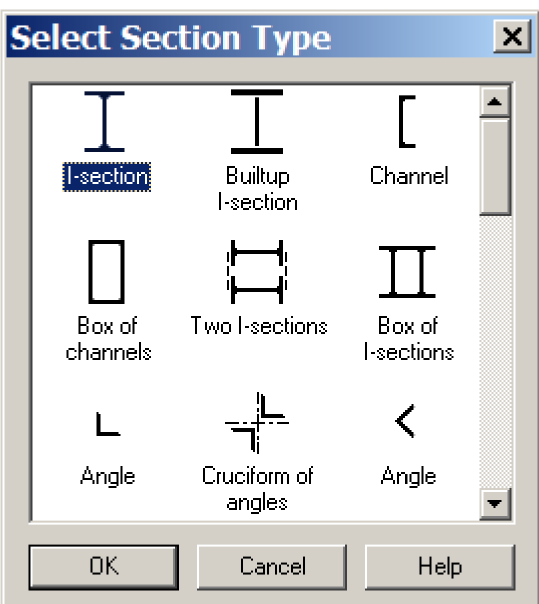


Figure 4.5

In the **Steel cross-section** dialog box, on the **Components** tab, in the drop-down lists **Profile**, select the profile table and number (name) for profile: *I-section with non-parallel* *flanges (45)*. Then, in the drop-down lists **Steel**, select the steel table and steel grade (*C245*).

To use forbidden combinations profile-steel (combinations may be found in the Steel Table module), select appropriate check box.

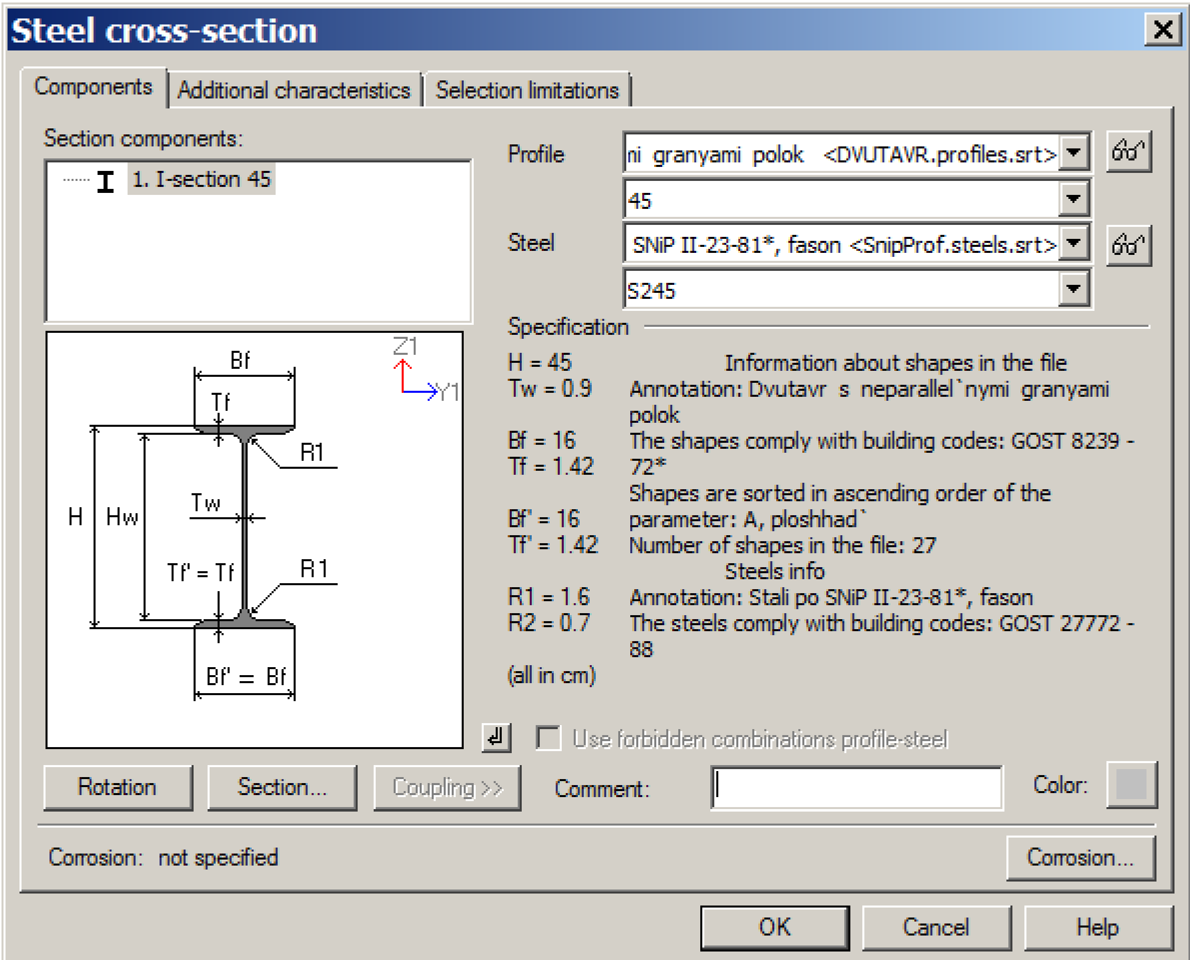


Figure 4.6

On the **Additional characteristics** tab, in the **Element type** area, click **Beam** (corresponds to analysis of flexure elements by SNIP II–23–81* 'Steel structures'). Input necessary data according to SNIP II–23–81* 'Steel structures' for the problem:
– service condition factor for strength – 1.1;
– safety factor – 0.95;
– in the **Analysis** area, click **Elastic**;
– in the **Deflection analysis** area, define **Max allowed deflection** as equal to 1/250;
– in the **Data for buckling analysis** area, under **Bracing of compressed flange**, for buckling analysis define effective length Lef = 1.5 m (suppose that beam is fixed out-of-plane with purlins of step 1.5 m).

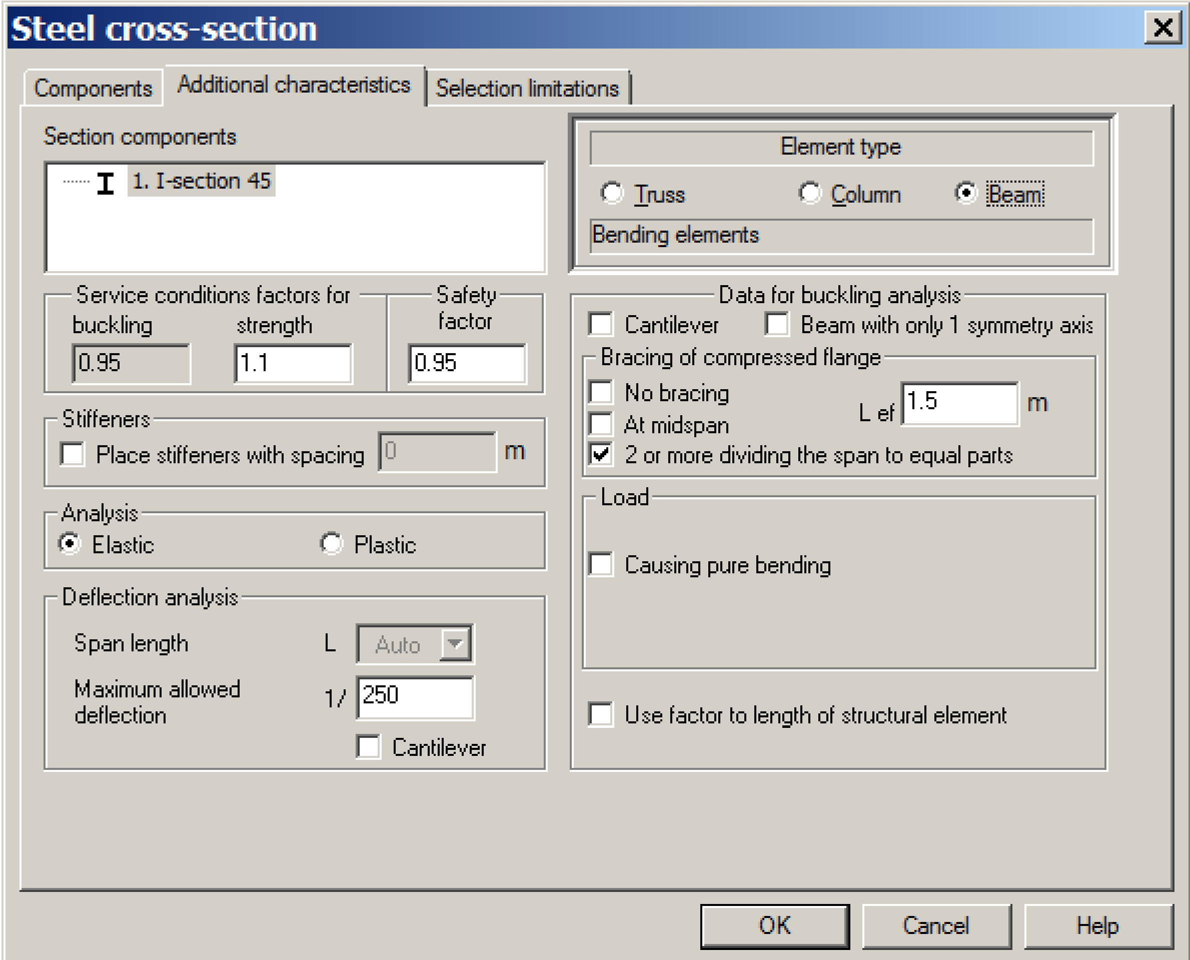
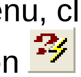


Figure 4.7

On the **Selection limitations** tab you could define parameters for limitation in selection procedure. In the Limitations area, define overall dimensions along axes and min thickness. It enables you to limit min and max height and width of sections during selection procedure. This option is frequently used for selection of weld composed sections.
Click **OK**.

Output data

To check or select section of element in steel structure, on the RESULTS menu, click **Check element** (button  on the toolbar) or **Select element section** (button  on the toolbar).

If you have to check or select separate section of an element, on the RESULTS menu, click **Choose section** (button  on the toolbar) and click any of section diagrams where selection or check should be made.

To switch to the next or previous section or to return to the mode of results for element as a whole, on the RESULTS menu, use the **Next section** (button), **Previous section** (button) or **Element as a whole** commands (button).

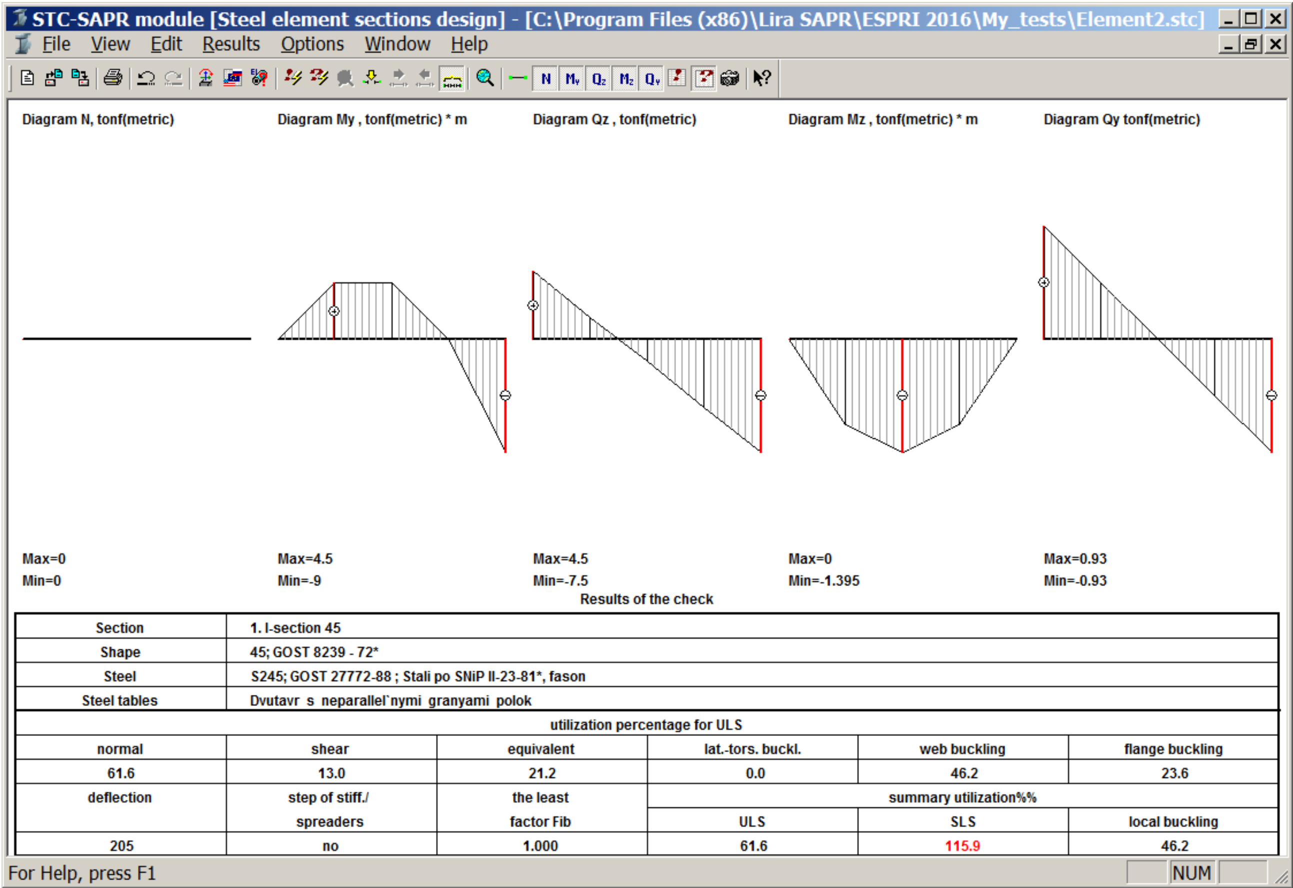


Figure 4.8

Output data (for the check option) presents utilization ratio values by appropriate checks (ratio of bearing capacity of section to real stress, slenderness, etc. multiplied by 100%). Output data (for the selection option) presents determined section with utilization percentage by appropriate checks.

To carry analysis by another building code, on the OPTIONS menu, click **Building code** (button on the toolbar).

To print the output data directly from the program window, on the FILE menu, click **Print** (button). It is also possible to copy data to other applications that work with graphic objects. To do this, use the **Copy** button on the toolbar .

The program contains additional tools that enhance visualization options.

Principal and equivalent stresses in steel structures

The module enables you to calculate principal and equivalent stresses by different criteria of rupture applied for analysis of steel structures and structures of composite materials. Principal stresses σ_1, σ_2, σ_3 are determined by the certain stress tensor $\{\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{xz}, \tau_{yz}\}$.

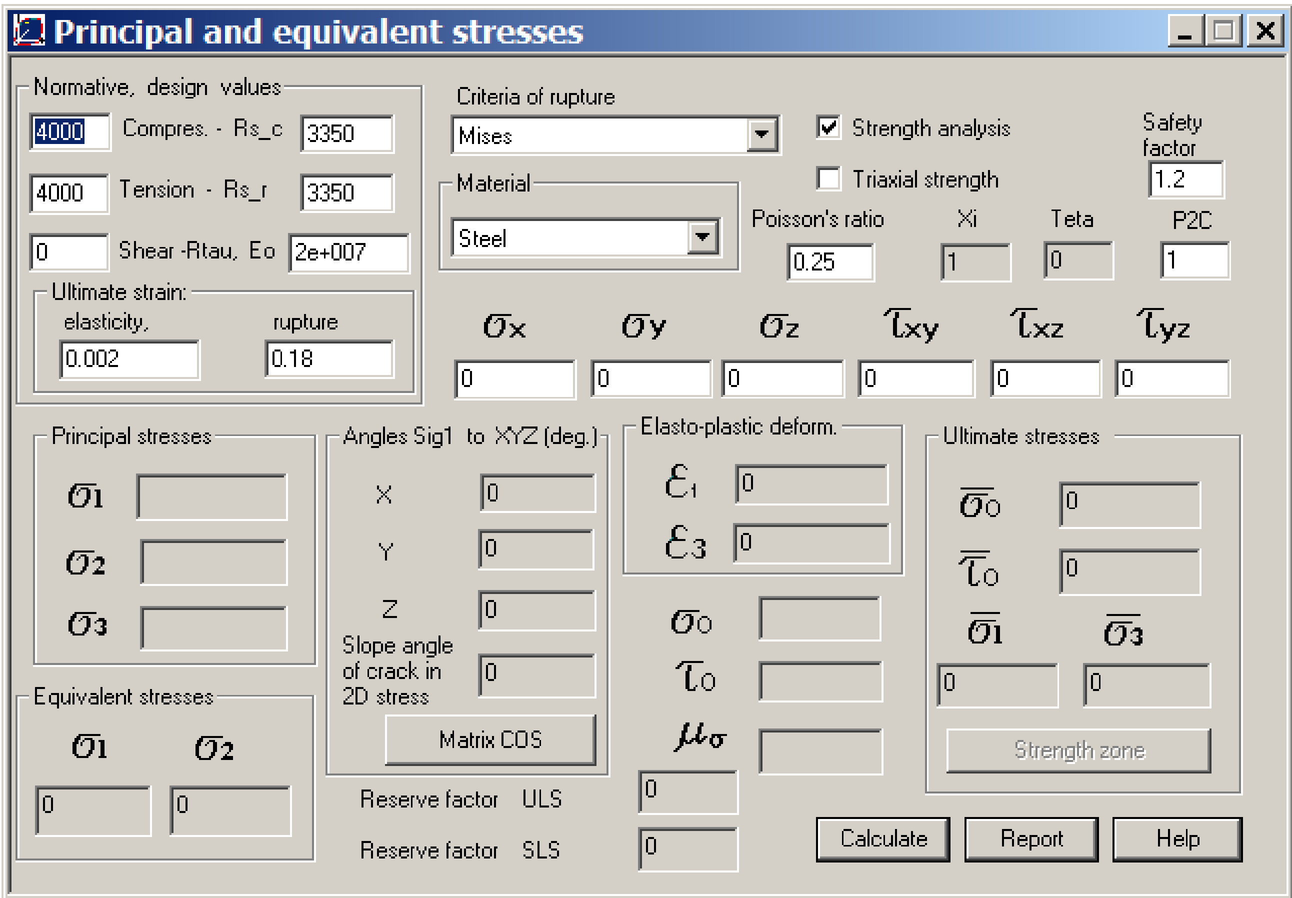


Figure 4.9

To find out roots of principal stresses, cubic equation is solved based on stress matrix [σ] in the form:

$$[\sigma] = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

Input data

Besides components of the stress tensor, it is necessary to define the following data:

– strength in compression **Rc** (tf/m^2);
– strength in tension **Rr <= Rc** (tf/m^2);
– for calculation of composite materials, strength in pure shear should be also defined – **Rtau**, if **Rtau=0** it is taken as:

Rtau = 0.5*Rc(1+c)/$\sqrt{3}$ **,**
where
c=Rr/Rc – fragility parameter of material;
v – Poisson's ratio;
– additional parameters for calculation by different strength criteria **c, θ** (or other parameter), **P2C** are determined automatically according to defined strength parameters: **Rc, Rr, Rtau**.

Output data

When the **Strength analysis** check box is selected, after strength analysis, the following data is displayed:

σ_1, σ_3 – ultimate allowed values of principal stresses;
σ_0, τ_0 – ultimate axial and shear octahedral stresses;
– reserve factor (analysis by serviceability limit state is supposed, that is, normative strength parameters of material are considered in analysis);
– Lode-Nadai parameter is computed as: $\mu_\sigma = \dfrac{(2\sigma_2 - \sigma_1 - \sigma_3)}{(\sigma_1 - \sigma_3)}$;

To generate report file in HTML format, click **Report**. The report file contains the following data:
– principal stresses as: σ1 > σ2 > σ3;
– matrix of directional cosines MC(3,3) – orientation of principal stresses in the space relative to the X,Y,Z axes:

$$MC = \begin{bmatrix} l1 & m1 & n1 \\ l2 & m2 & n2 \\ l3 & m3 & n3 \end{bmatrix}$$

– slope angles AL1, AL2, AL3 (l1, l2, l3) of principal stress σ_1 to the X, Y, Z-axes; angles define orientation of the crack plane in the space;
– slope angle α3 of principal stress σ_3 to the X-axis; angle defines orientation of the crack plane relative to the X-axis ;
– Euler angles TETA, PSI, FI (θ, ψ, φ) that define location of principal stresses σ_1, σ_2, σ_3 relative to the X, Y, Z-axes of global coordinate system;
– additional strength parameters:
  – strength in biaxial uniform compression – R2c = Rc*P2c,
  – strength in biaxial uniform tension – R2p.

Stress tensor		
Sig_X =1500.000	Sig_Y =-2500.000	Sig_Z = 0.000
Tau_XY= 450.000	Tau_XZ= 0.000	Tau_YZ= 0.000
Principal stress		
SIG_1=1550.000	SIG_2= 0.000	SIG_3=-2550.000
Equivalent stress by theory	Mises	
SIG_1E=1550.000	SIG_2E=-2550.000	
Octahedral stress - Normal	Sig_O =-333.333	
Octahedral stress - Shear	Tau_O =1690.332	
Lode-Nadai coefficient	Param_Lode = 0.2439	
Slope angle of crack	Alfa_crack_2x= -83.660	(degrees)
Slope angle Sig_1	to X, Y, Z-axes	of principal coordinate system
AL1= 3.031 (Radians)	AL2= 1.681	AL3= 1.571
AL1= 167.320 (Degrees)	AL2= 12.680	AL3= 90.000
Matrix of direction cosines	Orientation SIG_1,2,3 in triaxial stress strain state	MC(3,3)
L1=-0.99388	m1= 0.00000	n1= 0.00000
L2=-0.11043	m2= 0.00000	n2= 0.00000
L3=-0.00000	m3=-1.00000	n3= 1.00000
Material properties:	Steel	
Initial Poisson's ratio	Nu_o =0.250	
Normative strength in compression	Rc_n=4000.000	(tf/m2)
Normative strength in tension	Rp_n=4000.000	(tf/m2)
Strength in pure shear	R_tau=2309.401	(tf/m2)
Strength in biaxial compression	R_2c=4000.000	(tf/m2)
Strength in biaxial tension	R_2p=4000.000	(tf/m2)
Safety factor by stress	kRs =1.200	
Design compression strength	Rs_c =3350.000	(tf/m2)
Design tensile strength	Rs_r =3350.000	(tf/m2)
Initial modulus of elasticity	Es_o =2.0000e+007	(tf/m2)
Secant modulus, elastic-plastic	Es_cut =4.5276e+003	(tf/m2)
Ultimate strain elastic, rupture	Def_u =2.000000e-003	Def_pl =1.800000e-001
Principal strain, elastic	Def_1 =1.151316e-004	Def_3 =-1.342105e-004
Principal strain, elastic-plastic	Def_1n=3.423454e-001	Def_3n=-3.990769e-001
Ultimate principal stress		
SG_1L =4000.000	SG_2L = 0.000	SG_3L= -2844.524
SG_OL =-371.844	TAU_L = 2309.401	
Presence of cracks:	TB=1,2,3 - cracks;	TB=10 - crack and yield in compression;
TB=20 - yielding of steel in biaxial compression;	TB=5 - yielding of steel by ULS;	TB=50 - rupture by ultimate strain
State of steel in biaxial stress strain state	TB=50	
Reserve factor by SLS	1.116	
Reserve factor by ULS	0.930	
Utilization factor by SLS	0.89643	

Figure 4.10

Effective lengths of steel structure elements

The module enables you to determine effective lengths of the following elements of steel structures:

– Idealized conditions (columns) of uniform section;
– elements of uniform section with semi-rigid joints;
– elements of uniform section for one- or multi-srorey frames;
– intersecting elements of uniform section;
– continuous beams (top chord of truss) and columns (crane laced columns);
– elements of 3D laced structures;
– lower part of single-stage (crane) columns.

The following building code are implemented in the program:
1. SNIP II-23-81* 'Steel Structures';
2. Manual to design of steel structures (to SNIP II-23-81* 'Steel Structures').

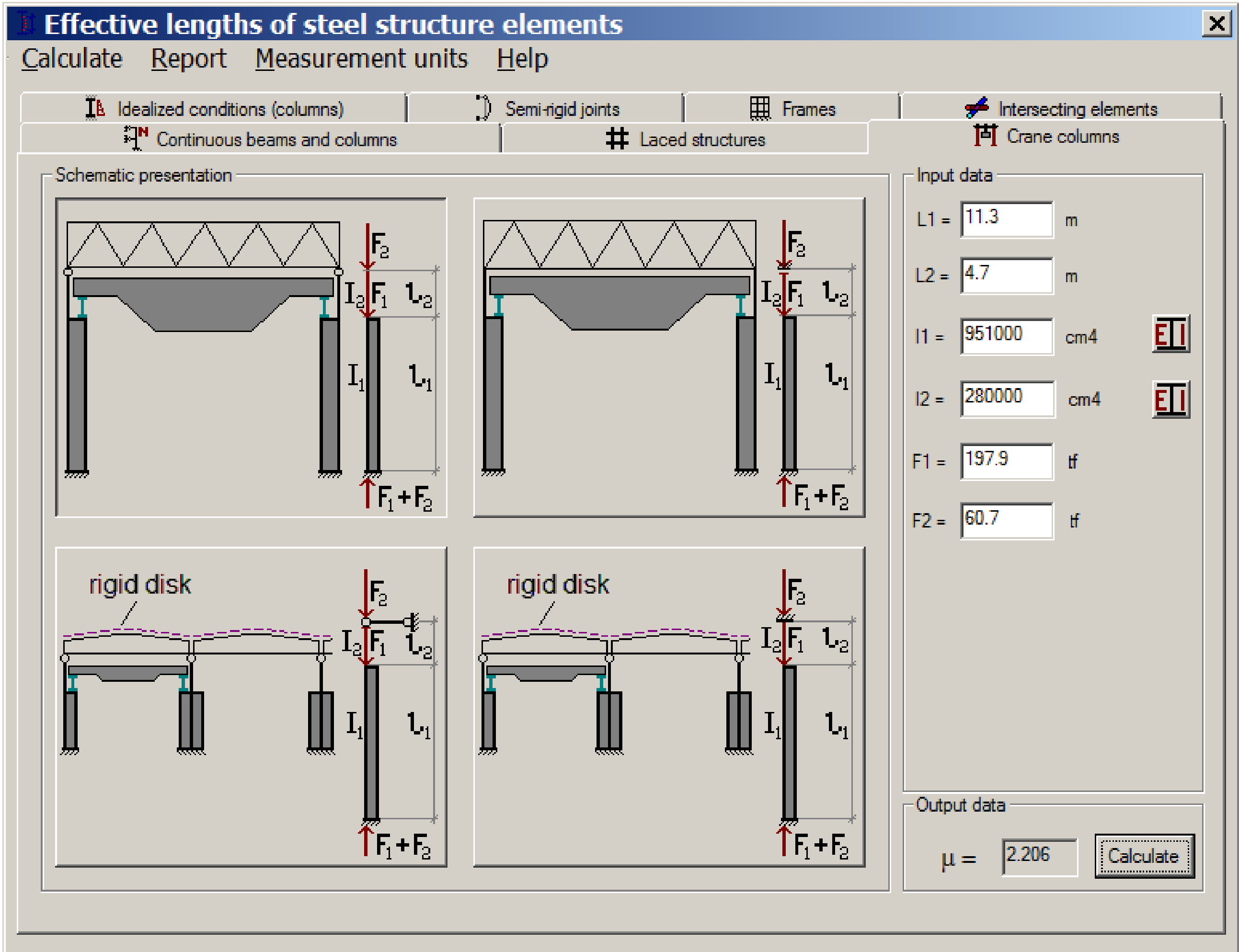


Figure 4.11

Input data

When you start the program, select appropriate tab with the short description of topology for computed elements of the structure (e.g. Crane columns).

In the dialog box, select with the pointer the element to be calculated or design model of the structure (except **Frames**, where frame parameters are described with number of defined storeys and spans).

Generally, it is necessary to define geometry for the element or the structure and stiffness properties. In some cases, define load on element or structure. The stiffness may be described either numerically or with program options in the **Steel cross-section** dialog box (button  - **Stiffness**). When you click the **Section** button in the **Steel cross-section** dialog box, it is possible to select different types of sections, for which stiffness is calculated automatically.

 www.liraland.com

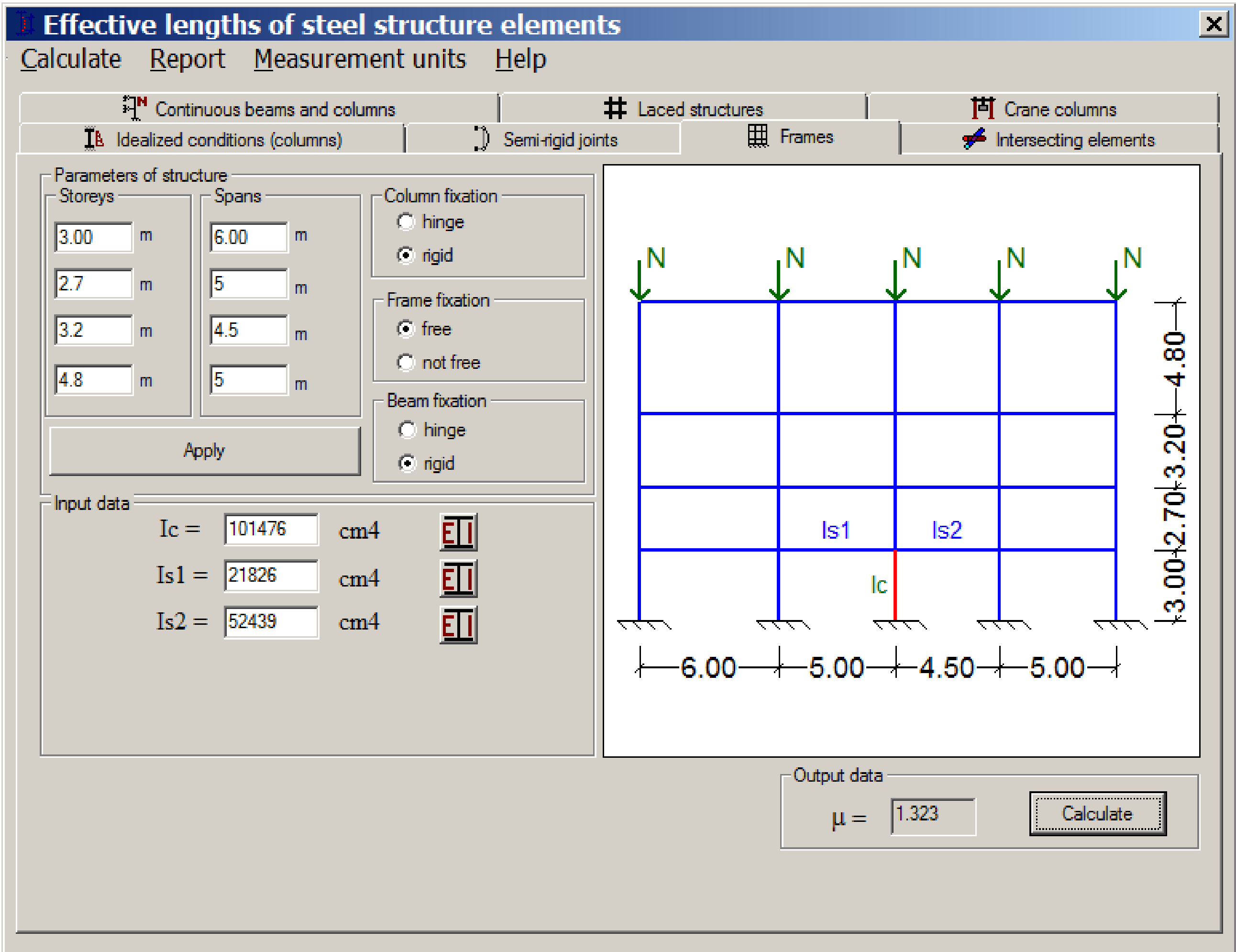
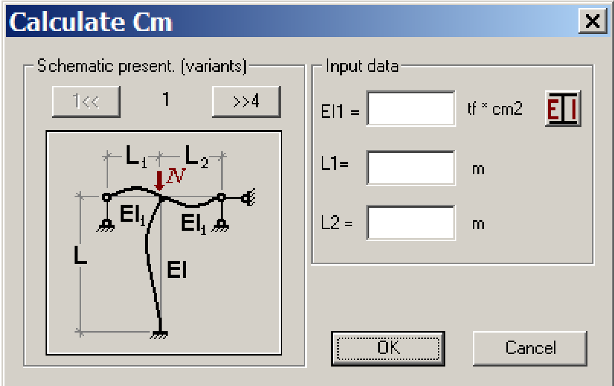


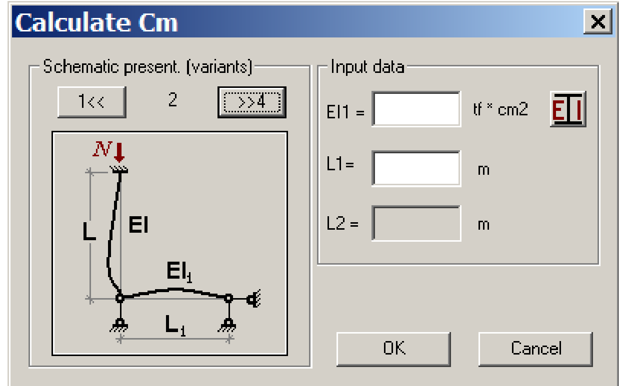
Figure 4.12

Elastic fixing at the ends on the **Semi-rigid joints** tab (elements of uniform section with semi-rigid joints) may be defined numerically or you could compute appropriate stiffness according to the variant of design model (button ).

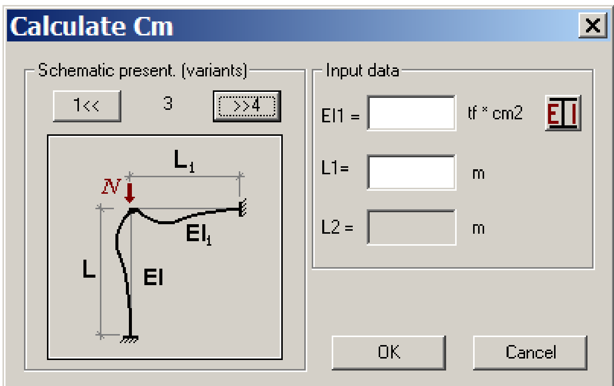
To start calculation, click **Calculate**.



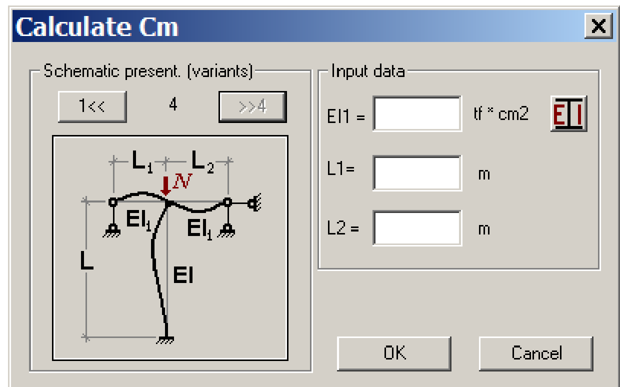
a)



b)



c)



d)

Figure 4.13

Output data

Output data contains coefficient (coefficients) **μ**. Effective length (lengths) may be obtained if geometric length of element (elements) of structure is multiplied by this coefficient (coefficients).
Output data may be presented as a report. To present a report document in HTML format, click **Report** on the menu bar.

Parametric joints of steel structures

This module represents STC-SAPR module adapted to work in ESPRI environment.
The module enables you to design and check joints of steel structures.
The program supports the following normative documents:
– SNIP II–23–81* 'Steel structures',
– Manual on design of steel structures (to SNIP II–23–81* 'Steel structures'),
– Provisions contained in [45, 46].

Input data

On the FILE menu, point to **New** and click **Joint**. In the **Select joint type** dialog box, select the joint prototype from the following menu: **Trusses, Beam-to-column joint, Beam-to-beam joint, Column-to-column joint, Column bases, Sway bracing, Rigid connections**.

Then select with the pointer the icon of appropriate joint and click **OK** or simply double-click appropriate icon.

In the **General** dialog box, define safety factor and service condition factor. In the **Group name for joint elements** list, select certain group of components: principal elements of joint, welds, plates, bolts, etc. List of groups depends on selected type of joint. When you click **Group properties**, the dialog box that corresponds to this group will be displayed on the screen.

For the principal elements of joint, from appropriate lists select the profile and its name, then steel table and steel name.

For other groups of elements, appropriate dialog boxes will appear. Necessary data is defined there by default. If necessary, correct the data.

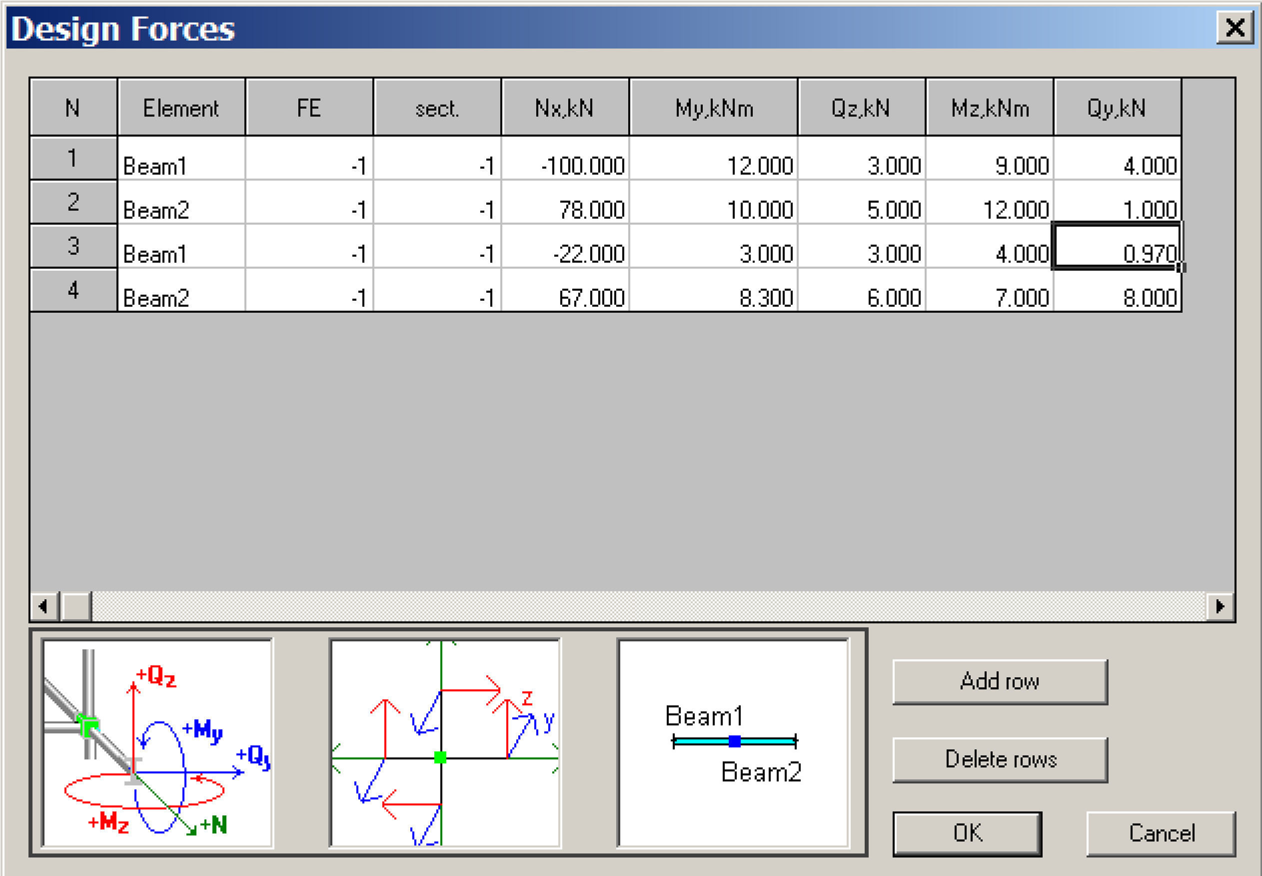
If it is necessary to input data about design resistances that differ from the ones mentioned in SNIP II–23–81*, then select the **Edit** check box for appropriate options and define the values manually.

For the joints of columns, in the **General** dialog box, define class of concrete for foundation.

When the input data is defined for every group (or for all groups), the **Apply specified properties to all elements of group** button becomes available. Click this button to apply properties.

Then click **OK**.

The **Design forces** dialog box will be displayed on the screen. There you should define forces at joint.



N	Element	FE	sect.	Nx,kN	My,kNm	Qz,kN	Mz,kNm	Qy,kN
1	Beam1	-1	-1	-100.000	12.000	3.000	9.000	4.000
2	Beam2	-1	-1	78.000	10.000	5.000	12.000	1.000
3	Beam1	-1	-1	-22.000	3.000	3.000	4.000	0.970
4	Beam2	-1	-1	67.000	8.300	6.000	7.000	8.000

Figure 4.14

When you click **OK**, analysis will be carried out.

The EDIT menu contains the following commands:
– **Forces** – to display the **Design forces** dialog box;
– **General properties** – to display the **General** dialog box;
– **Joint parameters** – to display the **Joint properties** dialog box with the joint type and defined parameters.

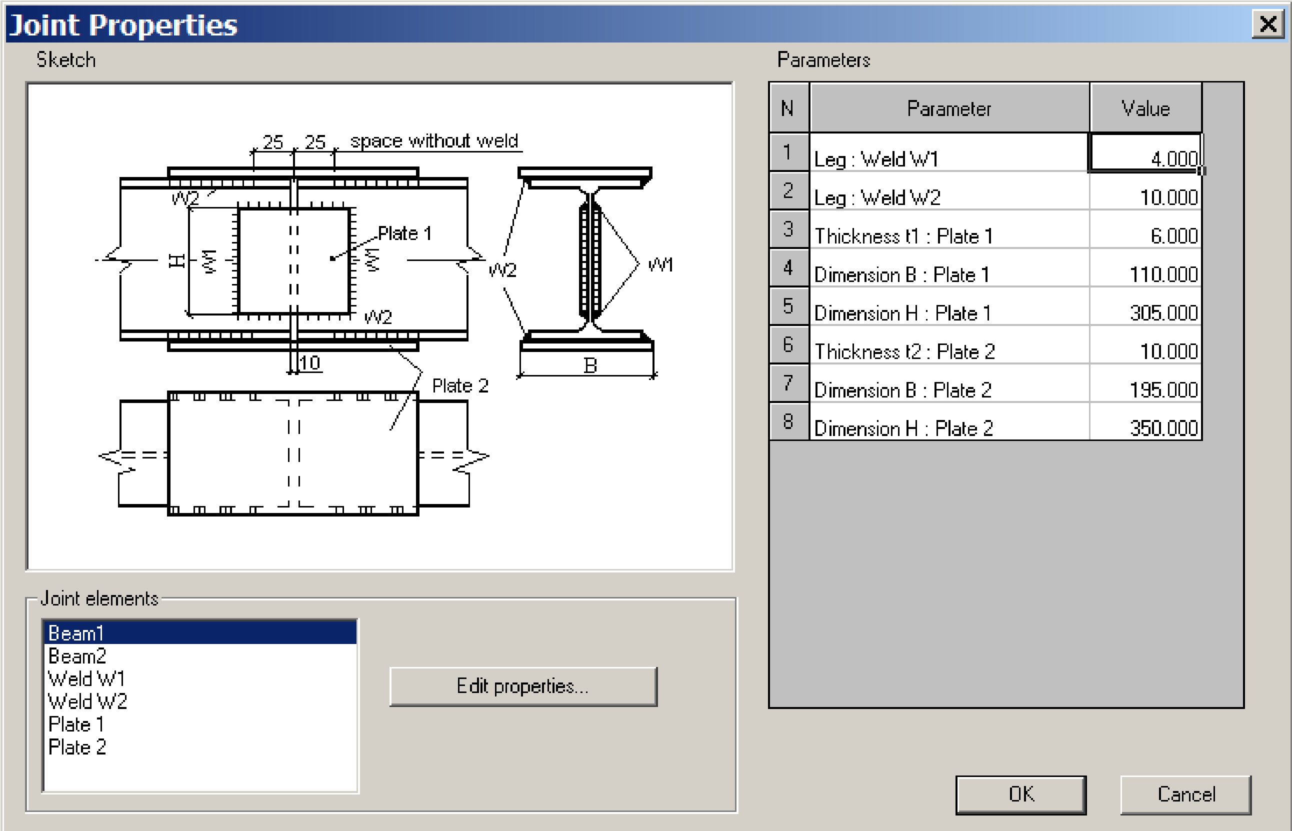


Figure 4.15

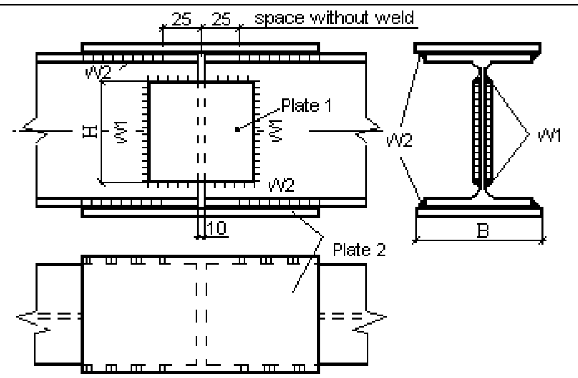
The ANALYSIS menu contains the following commands:
– **Check joint elements** – to check elements of joint according to selected prototype;
– **Select joint elements** – to select elements of joint according to selected prototype;
– **Tracing routine** – to generate file with tracing routine about analysis procedure. This file will be opened in web-browser right away.

Output data

Output data presents utilization ratio (ratio of bearing capacity of element in a joint to the real stress multiplied by 100%) by strength of elements available at joint as well as sketch of drawing.

STC-SAPR module [Steel joints parametric design] - [Joint1]

File Edit Analysis Options Window Help



Joint : Input Data

Joint Element	Property	Value	Units
Beam1	Shape	I35B1;GOST 26020 - 83	--
	Steel	09G2;	--
Beam2	Shape	I35B1;GOST 26020 - 83	--
	Steel	09G2;	--
Weld W1	Material	Wire mark: Св-08	--
Weld W2	Material	Wire mark: Св-08	--
Plate 1	Steel	ВСт3кп2	--
	Width	110.00	mm
	Length	305.00	mm
	Thickness	6.00	mm
Plate 2	Steel	ВСт3кп2	--
	Width	195.00	mm
	Length	350.00	mm
	Thickness	10.00	mm

Joint : Check Results (SNIP II-23-81*)

Parameter	Property	Value	Utilization ratio,%	Internal forces				
				N, kN	My, kNm	Qz, kN	Mz, kNm	Qy, kN
Weld W1	Leg	4.0 mm	10.4	-100.000*	12.000	3.000*	9.000	4.000
Weld W2	Leg	10.0 mm	105.8	78.000*	10.000*	5.000	12.000*	1.000*
Plate 1	Thickness t1	6.0 mm	5.4	67.000*	8.300	6.000*	7.000	8.000
	Dimension B	110.0 mm						
	Dimension H	305.0 mm						
Plate 2	Thickness t2	10.0 mm	14.4	-100.000*	12.000*	3.000	9.000*	4.000*
	Dimension B	195.0 mm						
	Dimension H	350.0 mm						

* - forces used in selection or check of the appropriate parameter

Done NUM

Figure 4.16

 www.liraland.com

Use the **Tracing routine** command to generate the tracing routine file for the check and selection of joint. This file will be opened in web-browser right away.

Analysis of welds

The program enables the user to analyse the following types of weld joints in steel structures:
— Lap joints
— T-joints, butt
— T-joints, flanged

The program supports the following building codes:
1. SNIP II-23-81* 'Steel structures. Design rules' — M.: TSITP Gosstroy USSR, 1990;
2. Manual on analysis and design of weld joints in steel structures (to SNIP II-23-81* 'Steel structures') / Moscow, 1984;
3. Manual on design of steel structures (to SNIP II-23-81*) / TSNIISK named Kucherenko Gosstroy USSR.

Analysis of welds in overlap joints is carried out according to sections 11.2*, 11.3*, 11.5, 12.8 SNIP II-23-81*.

Welding techniques are provided according to table 34* SNIP II-23-81*.

Min leg lengths are selected in **Select** option (or defined values are checked in **Check** option) according to table 38* SNIP II-23-81*.

The program works with steel tables that may be edited and created in the **Steel table** module (ESPRI program, **Steel structures** chapter).

The program supports the following building codes:

Mode	References to building codes
Steel	Table 50* SNIP II-23-81* Table 51* SNIP II-23-81* Table 51,б SNIP II-23-81*
Materials for welding	sect. 2.2*, 3.4 SNIP II-23-81* Table 55* SNIP II-23-81* Table 56 SNIP II-23-81*
Parameters of weld joints	sect. 11.2*-11.3*, 11.5, 12.8 SNIP II-23-81*
Partial safety factors	Table 6* SNIP II-23-81*

Input data

The program window contains four tabs where you could define input data for analysis of weld joint:

— Joint type
— Steel
— Parameters
— Welding materials

Input data for analysis of selected joint is defined in sequence tab by tab from left to right; at the same time the program checks whether assigned data is correct. Otherwise, the program displays the error message and it is not possible to select another tab in program window.

When defined data is approved as correct one, you will be able to select another tab and define appropriate data there. When you define data for selected template joint at any tab, you could return to any of the previous tabs to edit or review the previously defined data.

When you switch from one tab to another, all defined data is saved in both directions. If previously defined data do not correspond to current data, then the program opens the tab where the inconsistency is found and you could remove such inconsistency.

As you define input data, the program may display various warnings and tips for the user.

On the **Joint type** tab it is possible to define data about weld joint that should be analysed, its service conditions and parameters for welding.
In the **Select template joint** area, select their type: **Lap, T-joints (butt** and **flanged)**.
In the list of joint templates located below select appropriate template. For the check option, selected template will be displayed in the upper small window.
In the **Service conditions** and **Parameters of weld joint** areas, select in the drop-down lists and with option buttons all data necessary for analysis according to SNIP II–23–81* 'Steel structures'. Ratio of defined **Welding technique** and **Weld location** are checked for correspondence according to SNIP.

On the **Steel** tab it is possible to assign steel for every element in the joint (use appropriate option buttons to activate certain mode):
— **Profile (Strip)** and **Gusset** for the lap joint;
— **Profile (Rolled sheet)** and **End-plate** for the T-joint (butt);
— **Flange** and **Web** for the T-joint (flanged).

Depending on previously defined **Service conditions**, in the **Material for joined elements** box, for every element according to table 50* SNIP II-23-81* you will see lists of recommended steel or steel grades according to other building codes that may be used instead of steel recommended in GOST 27772-88.

With the help of option buttons you should define whether you will work either with classes or with grades of steel. In this case, when you select the class of steel in the **Steel recommended according to GOST 27772-88** list, in the special list **Steel grades by other GOST and TU** (not active at the moment) you will see steel grades that correspond to this class and vice versa.

In case of butt joint of pipes you will see the list of steel grades according to table 50* and table 51.a SNIP II-23-81* .
To compute flanged joint of built-up beam, it is possible to assign steels of different strength for flange and beam.
In the information window to the right you will see normative and design values of steel strength. These values depend on the thickness of steel profiles and sheets for selected class (grade) of steel in appropriate list.
The steel name and its mechanical properties for any element in the joint may be defined manually, if required. To do this, select appropriate check box. In this case, the input data is not checked but when you click the **Parameters** tab, defined value of thickness range is considered.

For steel profiles, in the **Designation** list for certain **Shape** you will see only shape in which flange thickness corresponds to the specified thickness range. If there are no such thickness values in the available shapes, then the **Designation** list remains empty and the program displays a warning that the thickness range should be modified.

Defined values of steel sheets are checked not so strictly. When you open the **Parameters** tab for the first time, min value for thickness range is displayed. Min value may be modified but within the range.

When you switch from the **Parameters** tab to the next tab **Welding materials**, the program checks correspondence between assigned thickness range for steel profiles and sheets of a certain element and the thickness value defined in the **Elements of joint** area. In case of any inconsistency, appropriate warning will be displayed and the thickness range should be modified.

Select with the pointer class or grade of steel for the element of weld joint and then click **Assign**. To do the same, it is also possible to double-click selected row. Information about defined steel for every element in the joint will be displayed in the information box to the right.
Class or grade of steel may be defined for four groups of structures and three climatic zones according to table 50* SNIP II-23-81*. If you select a steel for which there is a note in the table (about specific features of application), the program displays appropriate message. If the non-recommended steel is assigned for the gusset, you will see appropriate warning. But you could take a decision whether it is possible to apply such steel.

Important. The steel should be assigned separately for every element in the joint. To do this, use appropriate option buttons for every element in the joint and then select and assign steel for these elements.

On the **Parameters** tab it is possible to define data about dimensions of the joint and the forces that act on the joint. On schematic presentation of selected joint you will see necessary data about defined parameters and positive direction of applied forces.
Depending on the type of calculation (**Check** or **Select** options), either leg lengths and weld lengths are defined (**Check** option) and then their values are checked or min allowed values for leg lengths are displayed (**Select** option) as input ones. When you place the

pointer over appropriate boxes for leg lengths and weld lengths, you will see tooltips with information about allowed values for these parameters. When you select this tab for the first time, min allowed values for leg lengths and weld lengths are displayed by default.

In the **Elements of joint** area, select the **Shape** and **Designation** of the profile in the steel table or define height and thickness for the **Sheet**,as well as thickness for **Gusset** or **End plate**. For the butt joint of built-up I-section, in this area specify width and thickness of rolled sheets from which the I-section is consisted of and thickness of the end-plate.

For the T-joints (flanged) just as for the butt joint of built-up I-section, it is necessary to define geometric properties of rolled sheets that the built-up I-section of the beam is consisted of. If there is a local stress in this beam, you should define length (b) of application of concentrated load to flange of built-up beam.

To display additional information about main geometric properties of the sections, click the **Geometric properties** button.

In the **Type of analysis** area, select with option button either **Check** or **Select**.

If the **Check** option is selected, then weld lengths and leg lengths should be defined in appropriate boxes, schematic presentation is available. The check procedures (by SNIP) carried out in computation depend on the joint type and combination of loads that act on this joint. For this type of computation, nonzero forces acting on the joint are checked unlike the **Select** option where the same condition causes that parameters of welds are selected by design requirements.

If the **Select** option is selected, then min allowed leg lengths (for defined element thickness) are displayed while weld lengths remain as zero values. For certain types of joints, additional options for **Select** procedure are available. For L-shaped lap joints it is possible to define additional option **Leg lengths at the heel and toe are equal**, for the flanged joints - **One-sided flange weld**. For the T-joint (butt) of the sheet element, in both modes of analysis it is also possible to assign **One-sided weld and edge preparation in full penetration**.

In the **Forces** area, there are forces for which **Check** or **Select** procedures for weld parameters is carried out according to the sign convention displayed on schematic presentation (for the active forces equal to zero, parameters of weld will be selected according to design conditions).

In analysis of **T-joints (butt)**, the **check strength of joint across the thickness of rolled steel** option may be defined. In this case, when the tensile stresses arise in the main steel across the thickness of rolled steel, the program displays another dialog box where you should define **Pattern for edge preparation and type of penetration** and, if required, define depth for edge preparation for strength analysis of the end-plate across the thickness of rolled steel.

All linear dimensions are defined in mm.

Important. When you assign another **Shape** or thickness for any element in the joint from rolled sheet and then click at any box or at the **Leg length** boxes, in these boxes the program will display recomputed min allowed value for defined parameters of elements.

On the **Welding materials** tab it is possible to select and assign welding materials for the structure. Materials are selected according to table 55* SNIP II-23-81*. In the **Materials for welding** area, you will see the list of available electrode types or marks of welding wire; the content of the list depends on previously defined **Service conditions**.
With the mouse pointer, select the electrode type or mark of welding wire in the table.
In the lower left corner of the dialog box, the **Calculate** button becomes available.

Check of input data

When you move sequentially from one tab to another, the assigned data is checked for compliance with the requirements of SNIP II-23-81*. If the specified data does not satisfy the building code, one of the following messages is displayed:

 - information warning about non-compliance with recommendations SNIP; further work (defining input data and carrying out analysis) is possible as preferred by the user;

 - message about error in the input data, further work (defining input data and carrying out analysis) is not available until this data is modified.

When a steel for which specific features are specified in table 50* SNIP II-23-81* is assigned to elements of joint, the program displays the warning with the  icon.

Data specified on the Joint type tab is checked for mutual consistency according to SNIP provisions mentioned below.
Selected **Welding technique** and **Weld location** allow you to determine the coefficients β_f and β_z according to table 34* SNIP II-23-81*. When assigned weld location does not correspond to selected welding technique and you try to move to the next tab, the warning is displayed and you cannot move to the next tab.

In case of butt joint for the pipe for structures of *Group 1* (table 50* SNIP II-23-81*), the program also displays appropriate warning.

Defined value of **safety factor** for **unique object** for **class of responsibility** is also checked.

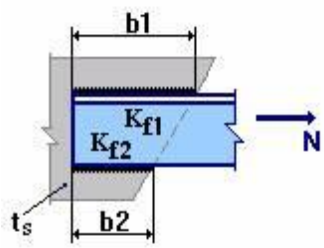
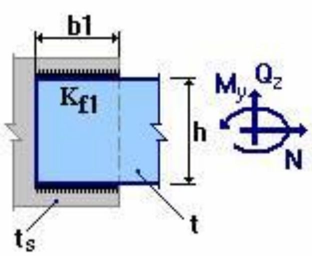
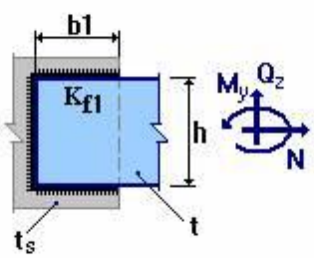
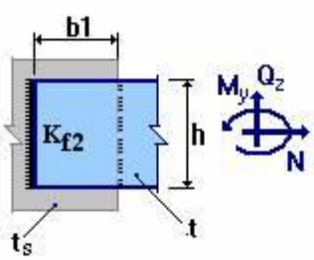
When you move from the Steel tab to the following Parameters tab, the program checks the steel assigned for the second element in the joint.

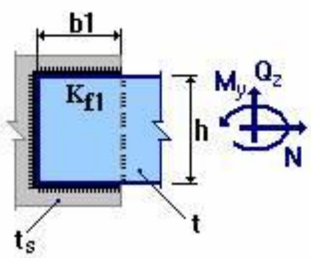
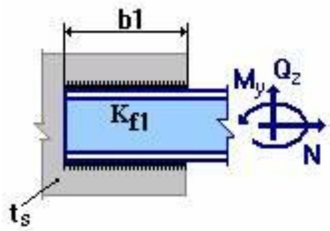
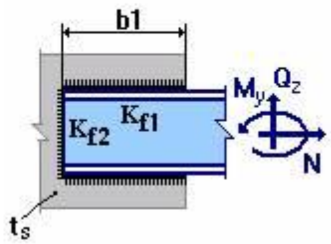
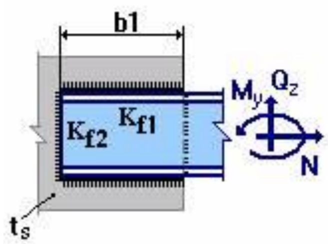
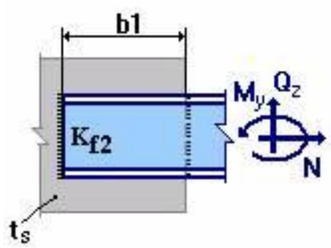
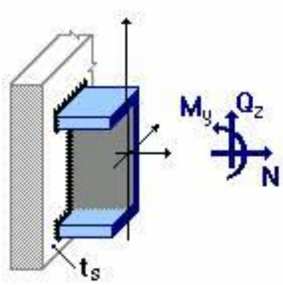
Values for **Parameters** of joints of all types are checked according to sect.11.2, in some cases - also sect.11.3, 11.5 SNIP II-23-81*. Defined values of element thickness are checked for:

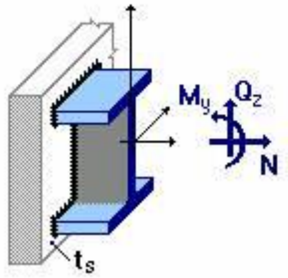
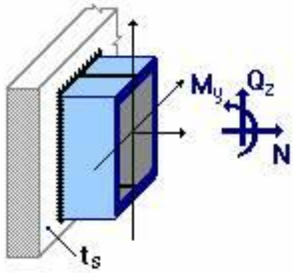
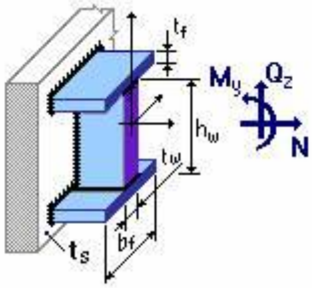
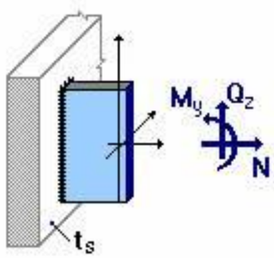
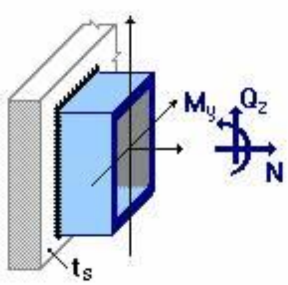
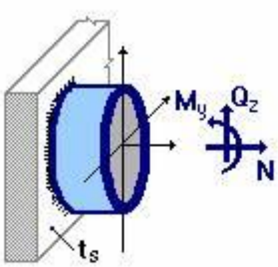
- whether it is allowed to weld (table 38* SNIP II-23-81*);
- whether steels defined for elements are allowed.

For lap joints of sheets, the program also checks whether defined height *h* is correct.

Here for the **Check** procedure, leg length and weld length are specified. When you move from one tab to another, the program checks whether these values correspond to requirements of sect.12.8 SNIP II-23-81*:

Joint	***Check of parameters***
	K_f min ≤ K_{f1} ≤ 1.2*min{1.2*t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} K_f min ≤ K_{f2} ≤ 1.2*min{0.9*t, t_s }, K_f min = Table 38* SNIP II-23-81*{min(0.9*t, t_s)} b1 ≥ max{ 40 mm, 4*$K_{f\,1}$} b2 ≥ max{ 40 mm, 4*$K_{f\,2}$}
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ max{ 40 mm, 4*K_{f1} } h ≥ 5*min{t, t_s }
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ max{ 40 mm, 4* K_{f1} } h ≥ max{ 40 mm, 4* K_{f1}, 5*min{t, t_s } }
	K_f min ≤ K_{f2} ≤ 1.2*min{t, ts}, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ 5*min{t, t_s } h ≥ max{ 40 mm, 4* K_{f2} }

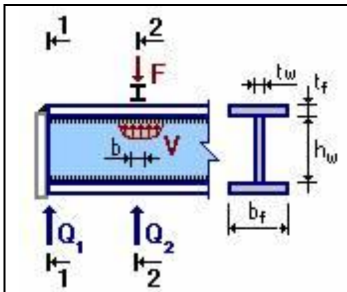
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ max{ 40 mm, 4* K_{f1}, 5*min{t, t_s } h ≥ max{ 40 mm, 4* K_{f1}, 5*min{t, t_s }
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ max{ 40 mm, 4* K_{f1} }
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – flange thickness of channel section K_f min ≤ K_{f2} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)},where t – web thickness of channel section b1 ≥ max{ 40 mm, 4* K_{f1} }
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – flange thickness of channel section K_f min ≤ K_{f2} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – web thickness of channel section b1 ≥ max{ 40 mm, 4* K_{f1}, 5*min{t, t_s }
	K_f min ≤ K_{f2} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} b1 ≥ 5*min{t, ts}
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – flange thickness of profile K_f min ≤ K_{f2} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – web thickness of profile

 	
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – flange thickness of built-up I-section K_f min ≤ K_{f2} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)}, where t – web thickness of built-up I-section
	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s}, K_f min = Table 38* SNIP II-23-81* { max(t, t_s)} h ≥ max{ 40 mm, 4* K_f }
 	K_f min ≤ K_{f1} ≤ 1.2*min{t, t_s }, K_f min = Table 38* SNIP II-23-81* {max(t, t_s)}

In some cases it is not possible to define min leg lengths correctly because of attempt to weld elements of inappropriate combinations of thicknesses or to use steel not mentioned in Table 38* SNIP II-23-81*. In this case, zero values of the leg lengths are displayed in the corresponding input boxes and the warning is issued. When you carry out the **Select** procedure, the values of min leg lengths are also checked to be not equal to zero. To continue calculation, data should be reset.

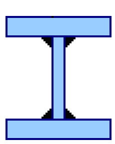
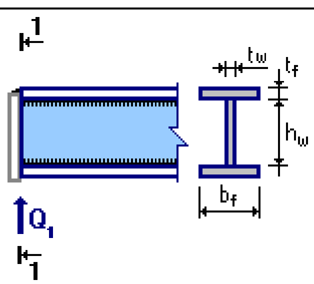
When you carry out the **Check** procedure, the program checks that at least one of the acting forces should be with non-zero value. For the **Select** procedure, zero forces in the joint are allowed and weld dimensions are selected according to design requirements.

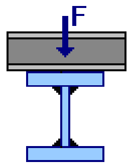
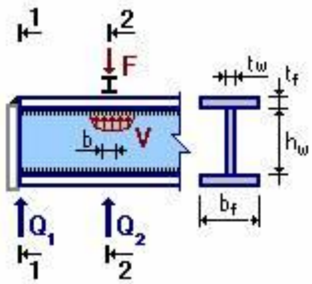
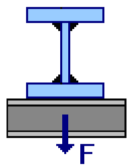
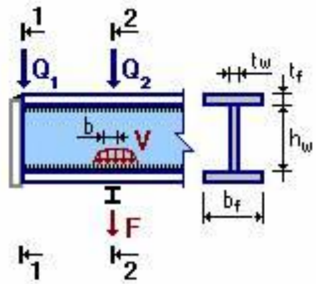
For lap joints of the L-shaped profile, if only longitudinal force is acting, the thickness of the gusset is checked for compliance with the recommended value depending on the acting force. In case of non-compliance, the warning is displayed.
Geometric properties of the section of the flanged T-joint are checked for compliance with the following requirements:

	$b_{f2} \geq b_f$, where b_f – flange width of built-up I-section in section 1-1, b_{f2} – flange width in section 2-2 $h_w \geq b_f$, $h_w > 40$ mm, $b_f > t_w$, $b \geq 50$ mm, if $F \neq 0$, $Q_1=0$ and $Q_2=0$ and $F=0$ (in this case, the weld is assigned according to design features) or $Q_1=0$ and $Q_2=0$ and $F=0$ or $Q_1 \neq 0$ and $Q_2 \neq 0$ and $F=0$ or $Q_1 \neq 0$ and $Q_2 \neq 0$ and $F \neq 0$

Weld T-joints (flanged)

In built-up beams, flanges are joined to webs by means of fillet welds.
For analysis of T-joints (flanged), the following types of flanged joints in built-up I-sections are available:

		Flanged joint of built-up I-section when there is no local stress ($\sigma_{loc} = 0$)

		Flanged joint of built-up I-section with local stress ($\sigma_{loc} \neq 0$)
		Flanged joint of built-up I-section with local stress ($\sigma_{loc} \neq 0$)

Load on flange of built-up beam may be transferred from steel purlin and from RC slab. Pressure V from design value of concentrated loads F should be computed by formula

$V = F/l_{ef}$,
where
$l_{ef} = b + 2t_f$ – effective length of load distribution,
t_f – thickness of appropriate flange of beam.
Important. When load F is applied to the bottom flange of beam, welds of the bottom flange are computed with account of force Q (regardless of the presence of stiffener in section where load F is applied to).

Output data

To carry out analysis, click the **Calculate** button (on the **Welding materials** tab).

Output data is presented on the **Parameters** tab. This tab is displayed automatically after analysis procedure. The **Report** button becomes available after computation.

After the **Check** of weld joint, the program computes max utilization ratio of design strength of weld joints either by weld metal or by metal of fusion weld boundary (ratio of real stress to effective strength of weld, expressed as %).

In the **Select** procedure, the program computes min allowed leg lengths and corresponding weld lengths (that do not exceed min allowed effective lengths) that provide bearing capacity of the joint. Computed values are displayed in the appropriate boxes.

For analysis of T-joint butt welds in case of tensile stresses in the main steel across the thickness of rolled steel, the program carries out the check of steel strength. The output data contains utilization ratio of steel strength across the thickness of rolled steel according to this check.

In case forces (that act on the joint of any type) are defined as equal to zero for the **Select** procedure, the leg length and length of weld are assigned according to design features (you will see appropriate warning).

For the flanged joint, the program carries out only **Selection** of leg length at the support of beam and in the plane of local stress (section 2-2 Fig. in the table). If computed leg length exceeds max allowed value, then the program displays the warning that it cannot determine weld for the specified parameters and forces.

After analysis procedure, it is possible to generate the **Report** that contains the input data and the output data from **Check** or **Select** procedures. To generate and open the report file in HTML format, click **Report**. It is possible to save the copy of the report with user-defined name and location (on the FILE menu, use the **Save as** command).

Bolted connections

The module enables the user to analyse and design lap bolted connections in steel structures of the following main types:
— connections of single and double angles to the gusset;
— connections of sheet elements with splices;
— connections of beam web (rolled or built-up) with splices.

The program supports the following building codes:
1. SNIP II-23-81* 'Steel structures. Design rules' — M.: TSITP Gosstroy USSR, 1990;
2. Manual on design of steel structures (to SNIP II-23-81*) / TSNIISK named Kucherenko Gosstroy USSR.

The program window contains four tabs where you could define input data for analysis of bolted connection:
Connection type
Steel of elements
Parameters of connection
Requirements to bolts

When input data is defined and analysis results are obtained, the output data is generated as report file that may be printed or saved, if necessary.

Set of checks by SNIP II-23-81* depends on type of connections and sections of its elements, type of bolts (without controlled tension or with controlled tension - friction-grip bolts) and all loads applied to connection. Below there is a list of checks for every type of connection.

Connections of angle sections:
· bolt in shear (without controlled tension) — sect.11.7;
· connection in shear (friction-grip bolts) — sect.11.13*;
· angle in local compression — sect.11.7;
· gusset in local compression — sect.11.7;

· strength in weakened section of angle — sect. 5.1.

Connections of sheet elements with splices:
· bolt in shear (without controlled tension) — sect.11.7;
· connection in shear (friction-grip bolts) — sect.11.13*;
· splice in local compression — sect.11.7;
· gusset in local compression — sect.11.7;
· strength in weakened section of splice — sect. 5.1.
· strength in weakened section of gusset (if its width is equal to width of splice) — sect. 5.1.

Connections of beam web:
· bolt in shear (without controlled tension) — sect.11.7;
· connection in shear (friction-grip bolts) — sect.11.13*;
· web of beam in local compression — sect.11.7;
· splice in local compression — sect.11.7;
· strength in weakened section of web in beam — sect. 5.1.
· strength in weakened section of splice — sect. 5.1.

For analysis of different connections of structural shapes, the steel table from ESPRI program '**Steel structures**' chapter is used. Geometric properties of rolled sheets are defined by the user.

The program supports the following building codes:

Mode	References to buildings codes
Steel of elements	Table 50* SNIP II-23-81* Table 51* SNIP II-23-81* Table 51,6 SNIP II-23-81*
Matchmarks that determine location of bolt holes in elements	GOST 24839-81 «Steel structures. Arrangement of holes in rolled sheets. Dimensions» sect. 12.19*, Table 39 SNIP II-23-81*
Requirements to bolts	sect. 2.4*, 2.7*, 3.5, 3.7 SNIP II-23-81* Table 57*, 58* and 59* SNIP II-23-81* (bolts without controlled tension) Table 36* SNIP II-23-81* (friction-grip bolts)
Partial safety factor	Table 6* SNIP II-23-81*
Partial safety factor for connection	Table 35* SNIP II-23-81*

Input data

Input data for analysis of selected connection is defined in sequence tab by tab from left to right; at the same time the program checks whether assigned data is correct. Otherwise, the program displays the error message and it is not possible to select another tab in program window.

When defined data is approved as correct one, you will be able to select another tab and define appropriate data there. When you define data for selected template connection at any tab, you could return to any of the previous tabs to edit or review the previously defined data.

When you switch from one tab to another, all defined data is saved in both directions. If previously defined data do not correspond to current data, then the program opens the tab where the inconsistency is found and you could remove such inconsistency.

As you define input data, the program may display various warnings and tips for the user.

On the **Connection type** tab it is possible to define data about connection that should be analysed, its service conditions, type of structure and type of bolts.
In the **Select template connection** area, select the template type that will be displayed in the upper small window.
In the **Service conditions** and **Parameters of bolted connection** areas, select in the drop-down lists and with option buttons all data necessary for analysis according to SNIP II–23–81* 'Steel structures'.
In the **Parameters of bolted connection** area, define the type of structure from one of the following:
OHTL - overhead transmission line;
OSG - outdoor switchgear;
CS - overhead control system.

On the **Steel** tab it is possible to assign steel for every element in the connection (use appropriate option buttons to activate certain mode):
— **Profile (Strip)** and **Gusset** for the lap connection of angle and sheet;
— **Profile** (rolled or built-up from sheets) and **Splice** for connection of built-up or rolled beam.

Depending on previously defined **Group of structures** and **Climatic zone according to SNIP II-23-81***, in the **Material for connected elements** box, for every element according to table 50* SNIP II-23-81* you will see lists of recommended steel or steel grades by other building codes that may be used instead of steel recommended in GOST 27772-88.

With the help of option buttons you should define whether you will work with steel classes or with steel grades. When you select the class of steel in the **Steel recommended according to GOST 27772-88** list, in the special list **Steel grades by other GOST and TU** (not active at the moment) you will see steel grades that correspond to this class and vice versa.

In the information window to the right you will see normative and design values of steel strength. These values depend on the thickness of steel profiles and sheets for selected class (grade) of steel in appropriate list.

The steel name and its mechanical properties for any element in the connection may be defined manually, if required. To do this, select appropriate check box. In this case, the input data is not checked but when you click the **Parameters** tab, defined value of thickness range is considered. If any property of user-defined steel was modified but not confirmed when you select this tab once more, then when you switch to another tab you will see appropriate warning.

For steel profiles, in the **Designation** list for certain **Profile** you will see only profiles in which flange thickness corresponds to the specified thickness range. If there are no such thickness values in the available profiles, then the **Designation** list remains empty and the program displays a warning that the thickness range should be modified.

Defined values of steel sheets are checked not so strictly. When you open the **Parameters** tab for the first time, min value for thickness range is displayed. Min value may be modified but within the range.

When you switch from the **Parameters** tab to the next tab **Requirements to bolts**, the program checks correspondence between assigned thickness range for steel profiles and sheets of a certain element and the thickness value defined in the **Elements of connection** area. In case of any inconsistency, appropriate warning will be displayed and the thickness range should be modified.

Select with the pointer class or grade of steel for the element of bolted connection and then click **Assign**. To do the same, it is also possible to double-click selected row. Information about defined steel for every element in the connection will be displayed in the information box to the right.

Class or grade of steel may be defined for four groups of structures and three climatic zones according to table 50* SNIP II-23-81*. If you select a steel for which there is a note in the table (about specific features of application), the program displays appropriate message.

Important. The steel should be assigned separately for every element in the connection. To do this, use appropriate option buttons for every element in the connection and then select and assign steel for these elements.

On the **Parameters** tab it is possible to define data about dimensions of the connection, forces that act on the connection, diameter and accuracy class of bolts, hole misalignment (difference in nominal diameter of holes and bolts) and location of centres of bolt holes.

On schematic presentation of selected connection you will see necessary data about defined parameters, matchmarks that determine location of bolt holes in connected elements, and about applied forces.

Depending on selected type of connection, matchmarks may be either completely defined by the user according to sect.12.19*, Table 39 SNIP II-23-81* or, for connections of angles, recommended ones (matchmarks stipulated in GOST 24839-81). In connections with

staggered arrangement of bolts, min distance between bolt centres is limited. In this case, the weakened section is checked and the area of weakened section is computed in vertical direction across the force.
When you place the pointer over appropriate boxes for matchmark values (distances from bolt centre to edge of element and between bolt centres in any direction) and number of bolt spacings in both directions, you will see tooltips with information about allowed values for these parameters. The program checks these parameters according to sect.12.19* SNIP II-23-81*.

When you open this tab for the first time, in the boxes with geometrical data the program displays min allowed values (thickness) for bolted connections and in boxes for matchmarks - their min allowed values are displayed as initial ones and may be modified.

In the **Elements of connection** area, select the **Profile** and its **Designation** in the steel table or define height and thickness for the **Splice**,as well as thickness for **Gusset**. For the connection with splice (splices)of built-up I-section, in this area specify width and thickness of rolled sheets from which the I-section is consisted of and thickness of the splice.

To display additional information about main geometric properties of the sections, click the **Geometric properties** button.

When you define **Forces** applied to connection, necessary **Accuracy class**, diameter of bolts $\mathbf{d_b}$ and **Hole misalignment**, dimensions for **Necessary contours of bolted field**, to confirm all modifications that determine allowed (min and max) values of matchmarks, click **Set matchmarks**. If the data is defined incorrectly or the program cannot select allowed value of matchmark, you will see appropriate warning with the list of errors, and in case of selection procedure, with recommendations about possible modification of this data.

All linear dimensions are defined in mm.

Important. When you assign another **Profile** or thickness for any element in the connection from rolled sheet, set of forces applied to the connection, values of matchmarks or other parameters of connection that determine matchmarks (diameter or accuracy class of bolts, hole misalignment), click **Set matchmarks**. The program will check whether current values of matchmarks are allowed.

On the **Requirements to bolts** tab it is possible to select and assign materials for the structure depending on type of bolts defined earlier. For bolts without controlled tension, in the **Requirements to bolts** area, the program displays the list of **Accuracy classes** for bolts stipulated depending on previously defined **Service conditions**. If temperature conditions for the connection are defined as below -40°C, the user will be able to select the mode of maintenance in heated space.

Select with the pointer accuracy class for the bolts. In the information boxes you will see appropriate values for design strength in shear R_{bs} and in tension R_{bt} for bolts of selected class. Below you will also see information about design strength in local compression for connected elements R_{bp} for the current class of accuracy defined on the **Parameters** tab.

In the lower left corner of the dialog box, click **Calculate**.

For the friction-grip bolts, in the **Connections on high strength bolts with controlled tension** area, define **Method of processing (cleaning) the surfaces to be joined** and information about mechanical properties of high strength bolts.

Select with the pointer certain method of processing the surfaces. In this case, in the information boxes you will see appropriate values of friction coefficient μ and load factor γ_h.

In the table, select the **Steel grade according to GOST 4543-71*** from the list. The contents of the list depend on previously defined nominal diameter of thread.

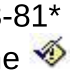
In the lower left corner of the dialog box, click **Calculate**.

Check of input data

When you move sequentially from one tab to another, the assigned data is checked for compliance with the requirements of SNIP II-23-81*. If the specified data does not satisfy the building code, one of the following messages is displayed:

 - information warning about non-compliance with recommendations SNIP; further work (defining input data and carrying out analysis) is possible as preferred by the user;

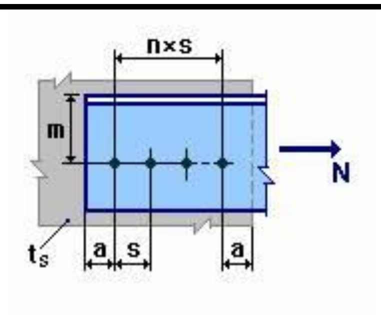
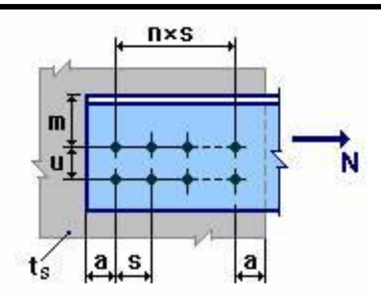
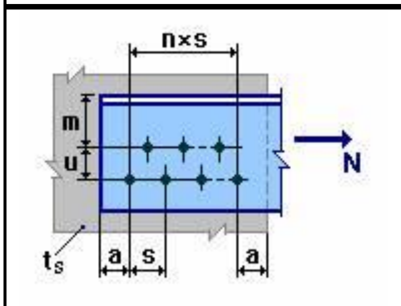
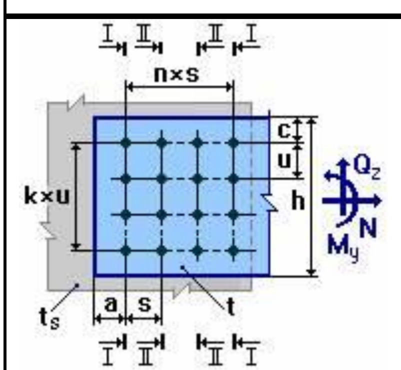
 - message about error in the input data, further work (defining input data and carrying out analysis) is not available until this data is modified.

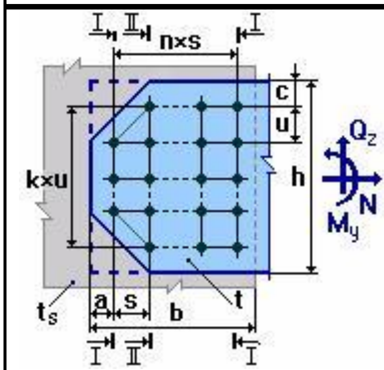
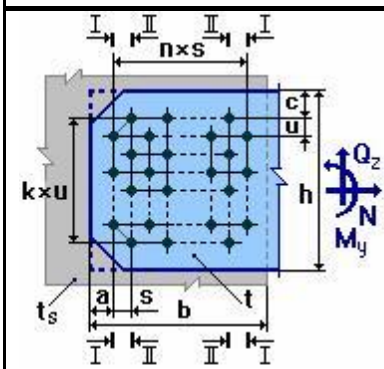
When a steel for which specific features are stipulated in table 50* SNIP II-23-81* is assigned to elements of connection, the program displays the warning with the  icon.

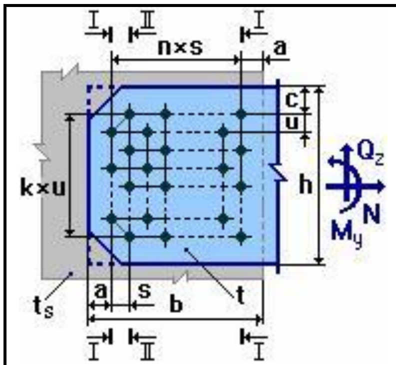
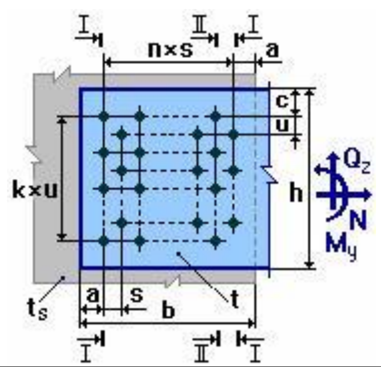
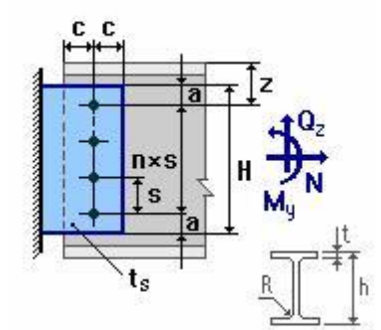
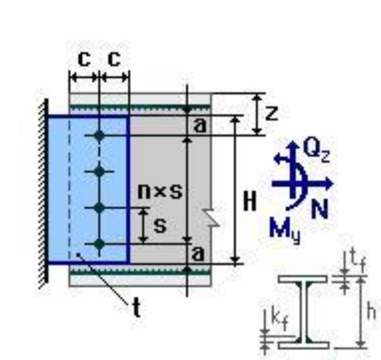
When you move from the Steel tab to the following Parameters tab, the program checks the steel assigned for the second element in the connection.

Values for **Parameters** of connections of all types are checked for mutual correspondence for further correct arrangement of bolts in connection according to SNIP II-23-81*. Defined values of element thickness are checked:
- whether it is allowed to connect with bolts of selected diameter (table 39 SNIP II-23-81*);
- whether steels defined for elements are allowed.

When you move to **Requirements to bolts** tab or click **Calculate**, the program checks whether defined values of matchmarks correspond to requirements of sect.12.19* SNIP II-23-81*:

Connection	***Check of parameters (Table 39 SNIP II-23-81*)***
	m – defined according to GOST 24839-81 $a_{min} \le a \le \min(4d,8T)$, $T=\min(t, t_s)$ – for connection of single angle to gusset T=t – for connection of double angles to gusset a_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.3d – for high strength bolts with controlled tension (friction-grip bolts) $s_{min} \le s \le \min(8d,12T)$ smin = 2d (Ryn≤ 375MPa) or 3d (Ryn>375MPa) n ≥ 0
	m, u – defined according to GOST 24839-81 $a_{min} \le a \le \min(4d,8T)$, $T=\min(t, t_s)$ – for connection of single angle to gusset T=t – for connection of double angles to gusset a_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.3d – for high strength bolts with controlled tension (friction-grip bolts) $s_{min} \le s \le \min(8d,12T)$ s_{min} = 2d (Ryn≤ 375MPa) or 3d (Ryn>375MPa) n ≥ 0
	m, u – defined according to GOST 24839-81 $a_{min} \le a \le \min(4d,8T)$, $T=\min(t, t_s)$ – for connection of single angle to gusset T=t – for connection of double angles to gusset a_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.3d – for high strength bolts with controlled tension (friction-grip bolts) $s_{min} \le s \le \min(8d,12T)$ s_{min} = 2(u +1.5d) n≥1
	a_{min} (c_{min}) ≤ a (c) ≤ min(4d,8T), $T=\min(t, t_s)$ – for connection of single splice or T= t_s – for similar connection with double splices a_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa) c_{min} = 1.5d – for sheared edges c_{min} = 1.2d – for rolled edges if N ≠0; Q_z, M_y = 0 c_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.5d – for sheared edges a_{min} = 1.2d – for rolled edges if Q_z≠0; N, M_y = 0 c_{min} = a_{min} = 1.5d ($R_{yn} \le$ 375MPa) or 2.5d (R_{yn}>375MPa), for all other combinations of forces a_{min} = c_{min} = 1.3d – for friction-grip bolts s_{min} (u_{min}) ≤ s (u) ≤ min(8d,12T)

	s_{min} (u_{min}) = 2d (R_{yn}≤ 375MPa) or 3d (R_{yn}>375MPa) n≥0 k≥0
	a_{min} (c_{min}) ≤ a (c) ≤ min(4d,8T), T=min(t, t_s) – for connection of single splice or T= t_s – for similar connection with double splices a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) c_{min} = 1.5d – for sheared edges c_{min} = 1.2d – for rolled edges if N ≠0; Q_z, M_y = 0 c_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.5d – for sheared edges a_{min} = 1.2d – for rolled edges if Q_z≠0; N, M_y = 0 c_{min} = a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa), for all other combinations of forces a_{min} = c_{min} = 1.3d – for friction-grip bolts s_{min} (u_{min}) ≤ s (u) ≤ min(8d,12T) s_{min} (u_{min}) = 2d (R_{yn}≤ 375MPa) or 3d (R_{yn}>375MPa) n≥1 k ≥ 2
	a_{min} (c_{min}) ≤ a (c) ≤ min(4d,8T), T=min(t, t_s) – for connection of single splice or T= t_s – for similar connection with double splices a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) c_{min} = 1.5d – for sheared edges c_{min} = 1.2d – for rolled edges if N ≠0; Q_z, M_y = 0 c_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.5d – for sheared edges a_{min} = 1.2d – for rolled edges if Q_z≠0; N, M_y = 0 c_{min} = a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa), for all other combinations of forces a_{min} = c_{min} = 1.3d – for friction-grip bolts u_{min} ≤ u ≤ min(8d,12T) u_{min} = d (R_{yn}≤ 375MPa) or 1.5d (R_{yn}>375MPa) s_{min} ≤ s ≤ min(8d,12T) s_{min} = u +1.5d n≥2, always even k≥2, always even

 	a_{min} (c_{min}) ≤ a (c) ≤ min(4d,8T), T=min(t, t_s) – for connection of single splice or T= t_s – for similar connection with double splices a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) c_{min} = 1.5d – for sheared edges c_{min} = 1.2d – for rolled edges — if N ≠0; Q_z, M_y = 0 c_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.5d – for sheared edges a_{min} = 1.2d – for rolled edges — if Q_z≠0; N, M_y = 0 c_{min} = a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa), for all other combinations of forces a_{min} = c_{min} = 1.3d – for friction-grip bolts u_{min} ≤ u ≤ min(8d,12T) u_{min} = d (R_{yn}≤ 375MPa) or 1.5d (R_{yn}>375MPa) s_{min} ≤ s ≤ min(8d,12T) s_{min} = u +1.5d n≥1, always odd k≥2, always even
 	a_{min} (c_{min}) ≤ a (c) ≤ min(4d,8T), T=min(t, t_s) – for connection of beam web with single splice or T= t_s – for similar connection with double splices a_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) c_{min} = 1.5d – for sheared edges c_{min} = 1.2d – for rolled edges — if Q_z≠0; N, M_y = 0 c_{min} = 1.5d (R_{yn}≤ 375MPa) or 2.5d (R_{yn}>375MPa) a_{min} = 1.5d – for sheared edges a_{min} = 1.2d – for rolled edges — if N ≠0; Q_z, M_y = 0 a_{min} = c_{min} = 1.5d (R_{yn}<=375MPa) or 2.5d (R_{yn}>375MPa), for all other combinations of forces a_{min} = c_{min} = 1.3d – for friction-grip bolts s_{min} ≤ s ≤ min(8d,12T) s_{min} = 2d (R_{yn}≤ 375MPa) or 3d (R_{yn}>375MPa) z – a >= t + R +2 – for rolled beam z – a >= t_f + k_f +2 – for built-up beam h – n*s = 2z n ≥ 0 k_f ≥ 3

In some cases it is not possible to define correctly min values of matchmarks because of attempt to connect elements with unallowable thickness combinations for the specified steel for the current diameter of hole. It means that for these parameters, min allowed value of matchmark exceeds max value that is determined according to Table 39 SNIP II-23-81*.

When you open the **Parameters of connection** tab for the first time for the current connection or after significant changes to input parameters considered as a new problem, the min allowed values of the element thickness and matchmarks are displayed in the corresponding boxes. When you modify any parameters presented on this tab and then confirm them with the **Set matchmarks** button, a warning is displayed that assigned matchmarks do not correspond to the tolerance limits. The min allowed value of T (see above) is indicated. In this case, to perform further calculations, the data must be modified properly.

Output data

To carry out analysis, click the **Calculate** button.

Output data is presented on the **Parameters** tab. This tab is displayed automatically after analysis procedure. The **Report** button becomes available after computation.

After the **Check** of bolted connection, the program computes max utilization ratio of bearing capacity of weakened sections of elements in connections. Location of weakened sections is also provided.

After analysis procedure, it is possible to generate the **Report** that contains the input data and the output data from check procedures. To generate and open the report file in HTML format, click **Report**. It is possible to save the copy of the report with user-defined name and location (on the FILE menu, use the **Save as** command).

Cold-formed shapes

The module enables you to check and select sections of structures made of cold-formed steel shapes according to *Guidelines on design, manufacturing and assemblage of framework structures in low-rise buildings and attics of cold-formed galvanized shapes made at 'Balt-Profil' Ltd.* (Edited by E. Ajrumyan).

The work with this program is similar to the work with 'Analysis of steel elements' module. Dialog boxes to define the input data are presented in Figures 4.24 – 4.27.

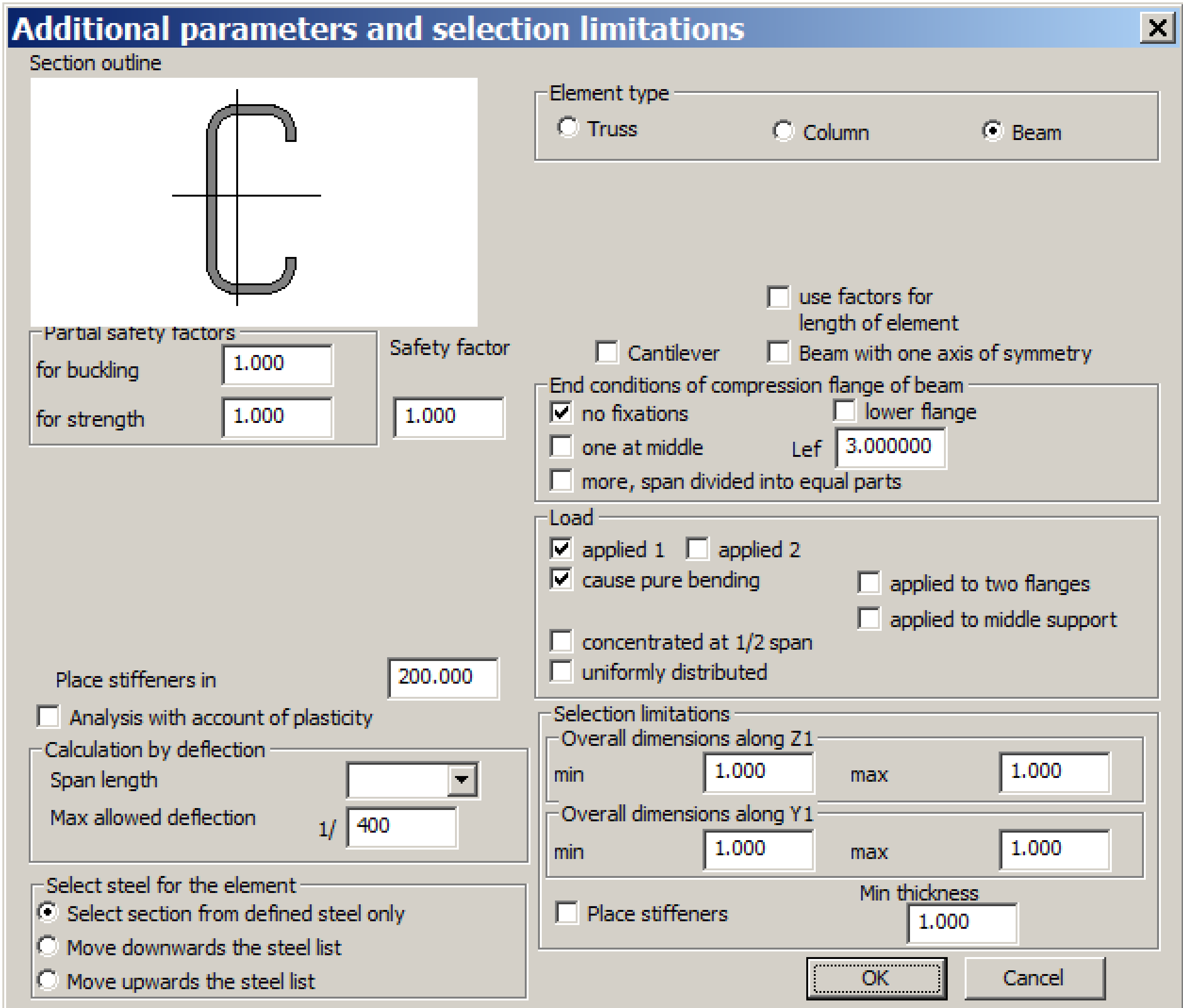

Additional parameters and selection limitations
Section outline
Element type
Truss
Column
Beam
use factors for
length of element
Partial safety factors
for buckling
1.000
for strength
1.000
Safety factor
1.000
Cantilever
Beam with one axis of symmetry
End conditions of compression flange of beam
no fixations
lower flange
one at middle
Lef
3.000000
more, span divided into equal parts
Load
applied 1
applied 2
cause pure bending
applied to two flanges
applied to middle support
concentrated at 1/2 span
uniformly distributed
Place stiffeners in
200.000
Analysis with account of plasticity
Calculation by deflection
Span length
Max allowed deflection
1/
400
Selection limitations
Overall dimensions along Z1
min
1.000
max
1.000
Overall dimensions along Y1
min
1.000
max
1.000
Select steel for the element
Select section from defined steel only
Move downwards the steel list
Move upwards the steel list
Place stiffeners
Min thickness
1.000
OK
Cancel

Figure 4.24

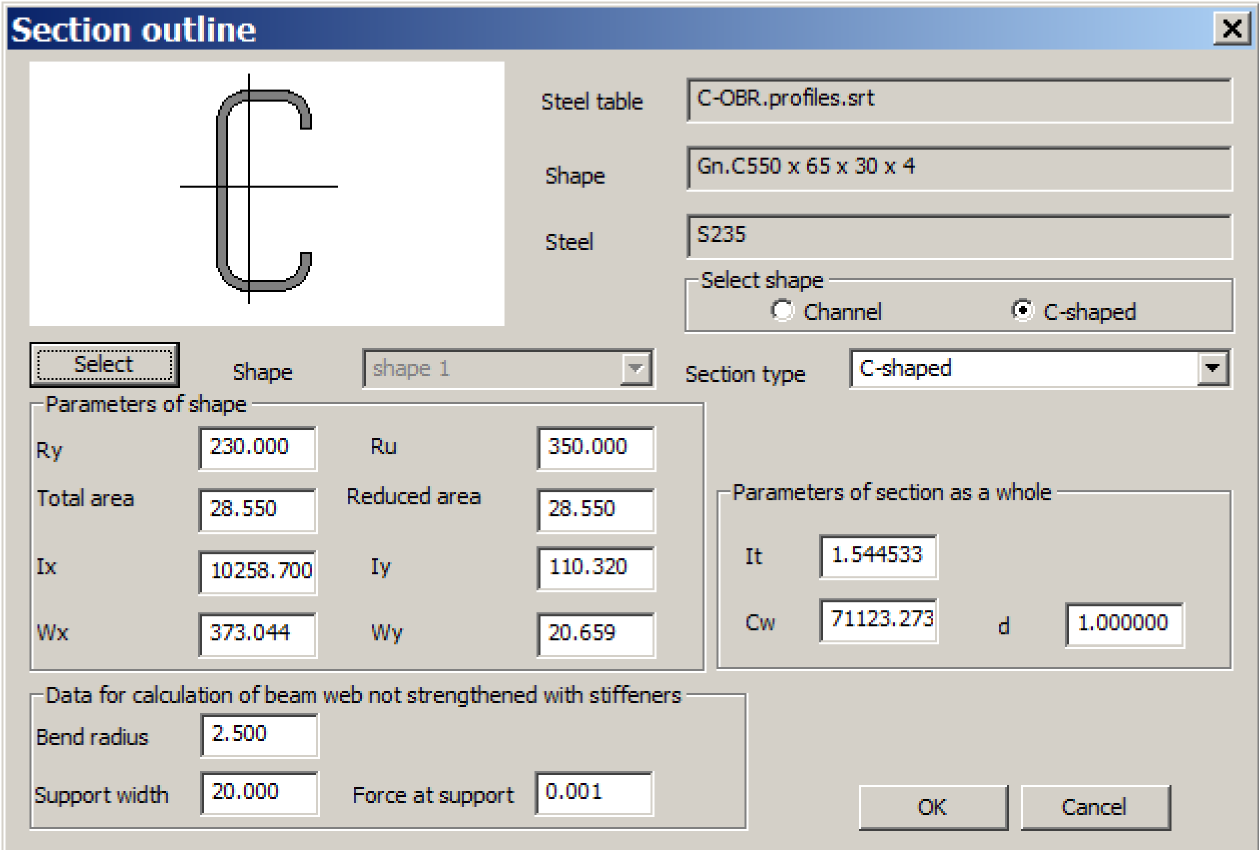

Section outline
Steel table
C-OBR.profiles.srt
Shape
Gn.C550 x 65 x 30 x 4
Steel
S235
Select shape
Channel
C-shaped
Select
Shape
shape 1
Section type
C-shaped
Parameters of shape
Ry
230.000
Ru
350.000
Total area
28.550
Reduced area
28.550
Ix
10258.700
Iy
110.320
Wx
373.044
Wy
20.659
Parameters of section as a whole
It
1.544533
Cw
71123.273
d
1.000000
Data for calculation of beam web not strengthened with stiffeners
Bend radius
2.500
Support width
20.000
Force at support
0.001
OK
Cancel

Figure 4.25

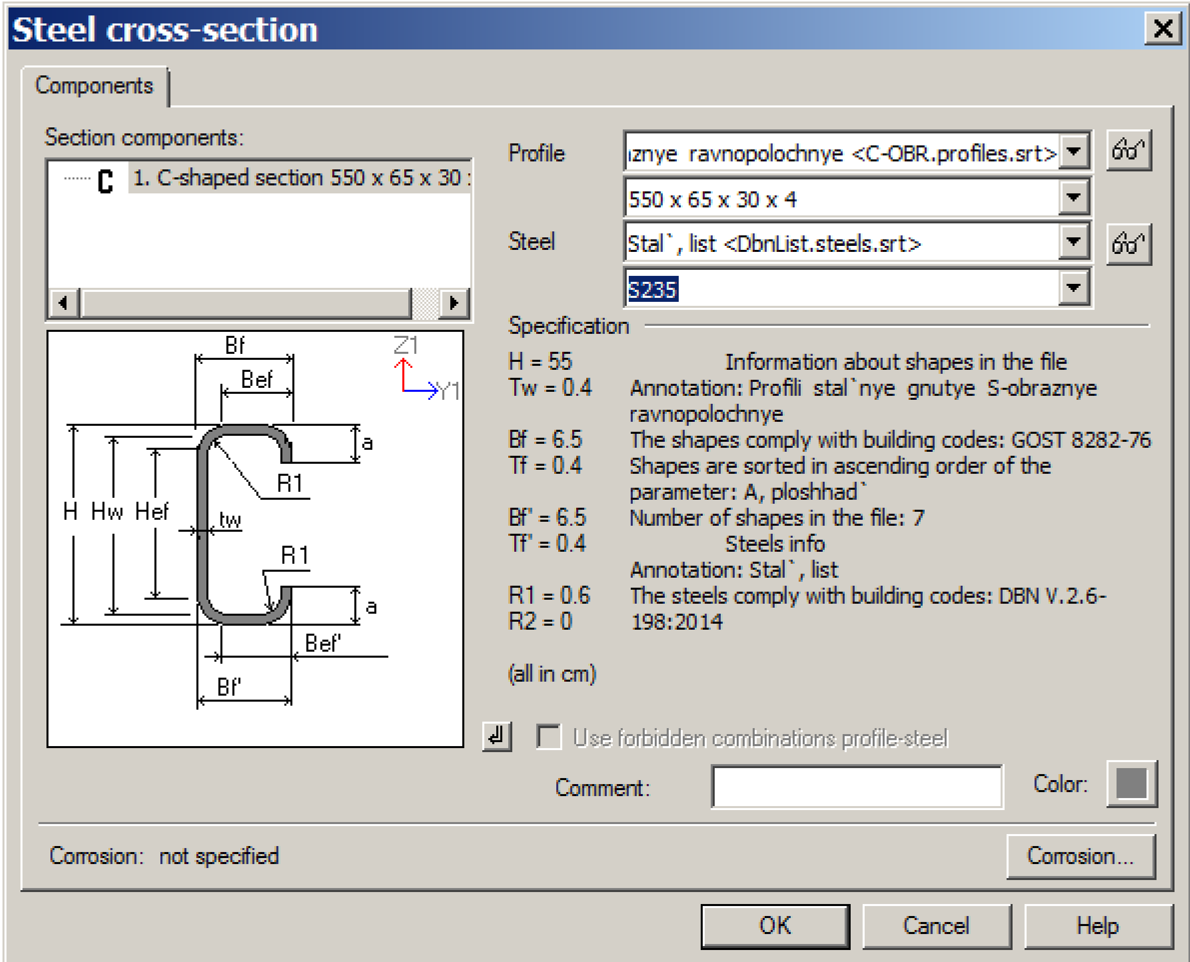

Steel cross-section
Components
Section components:
1. C-shaped section 550 x 65 x 30 :
Bf
Bef
Z1
Y1
a
R1
H Hw Hef
tw
R1
a
Bef'
Bf'
Profile
ıznye ravnopolochnye <C-OBR.profiles.srt>
550 x 65 x 30 x 4
Steel
Stal`, list <DbnList.steels.srt>
S235
Specification
H = 55
Tw = 0.4
Bf = 6.5
Tf = 0.4
Bf' = 6.5
Tf' = 0.4
R1 = 0.6
R2 = 0
(all in cm)
Information about shapes in the file
Annotation: Profili stal`nye gnutye S-obraznye ravnopolochnye
The shapes comply with building codes: GOST 8282-76
Shapes are sorted in ascending order of the parameter: A, ploshhad`
Number of shapes in the file: 7
Steels info
Annotation: Stal`, list
The steels comply with building codes: DBN V.2.6-198:2014
Use forbidden combinations profile-steel
Comment:
Color:
Corrosion: not specified
Corrosion...
OK
Cancel
Help

Figure 4.26

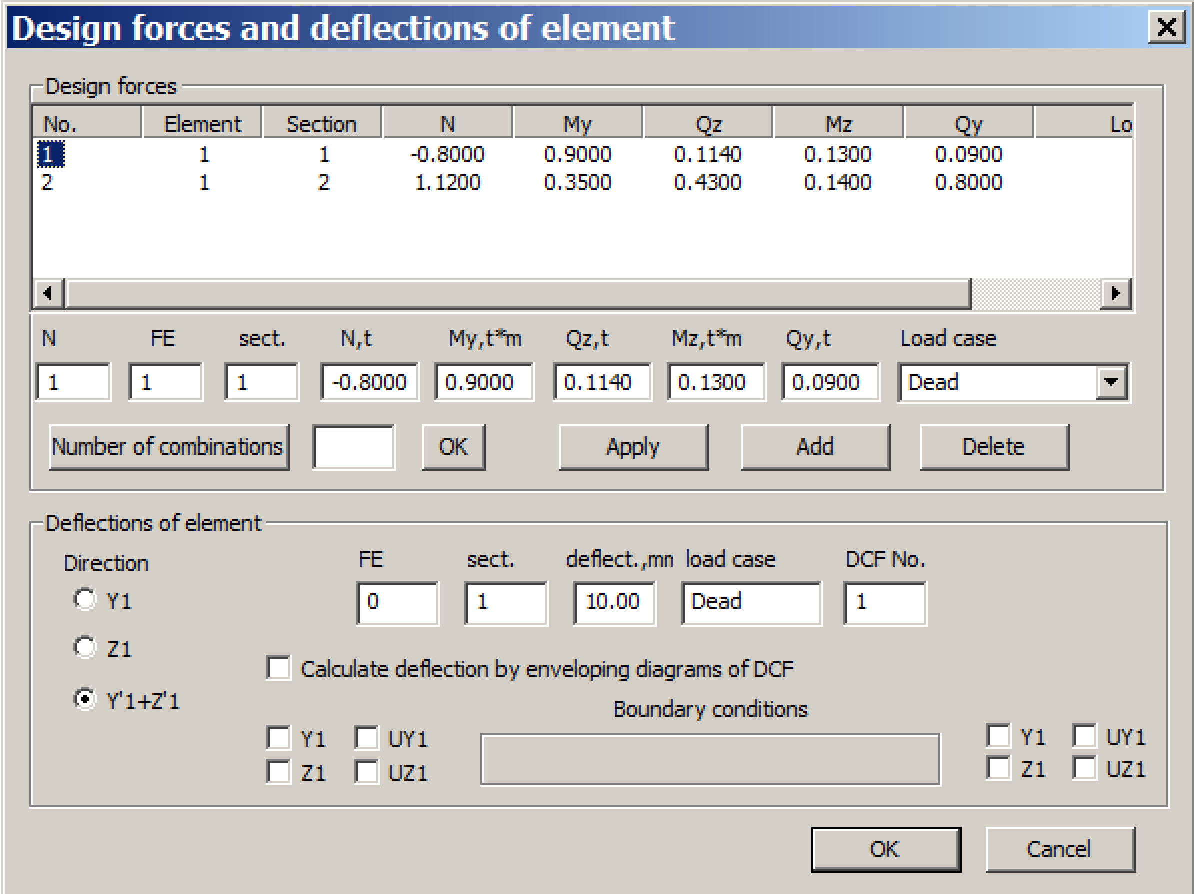


Figure 4.27

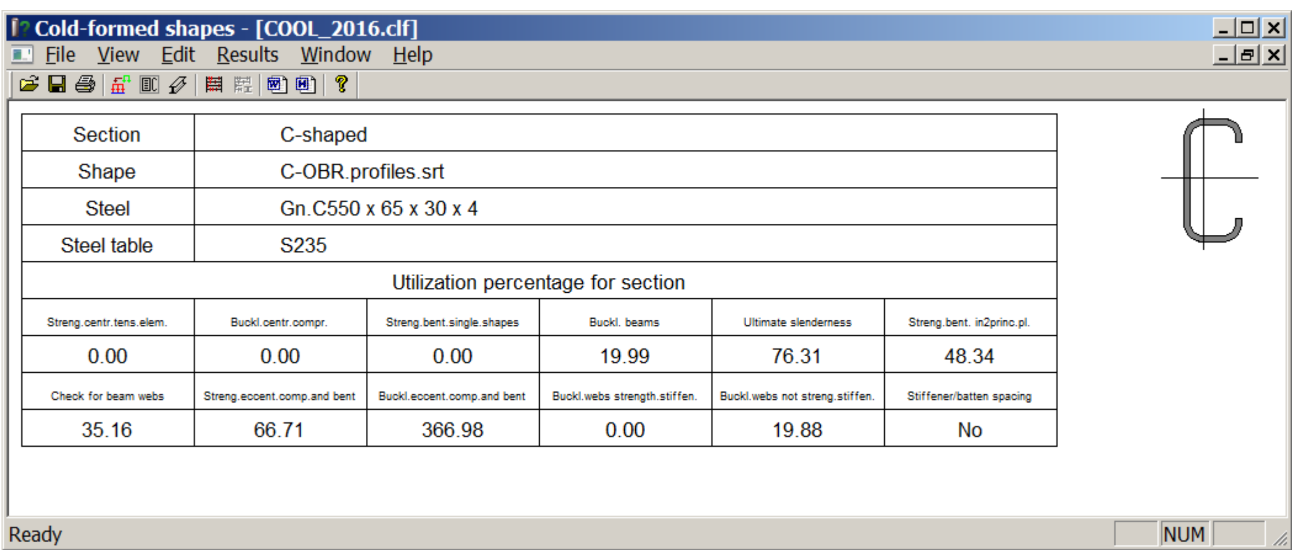


Figure 4.28

Cold-formed shapes	www.liraland.com
ESPRI 2016. Engineering Assistance Package. Version: 5.0.	
LIRA SAPR (Kiev, Ukraine)	international@liraland.com.ua

Element type: C-shaped

Steel table: C-OBR.profiles.srt

Shape: Gn.C550 x 65 x 30 x 4

Steel: S235

Geometric properties of section

F	F_red	Ix	Iy	Wx	Wy	It	Cw
28.55	28.55	10258.7	110.32	373.044	20.659	1.545	71123.273

Combinations of forces by which calculation is made

Combination No.	N	My	Qz	Mz	Qy
1	-0.8	0.9	0.114	0.13	0.09
2	1.12	0.35	0.43	0.14	0.8

Utilization percentage for section by combinations

Combination No.	Sh. 1	Sh. 2	Sh. 3	Sh. 4	Sh. 5	Sh. 6	Sh. 7	Sh. 8	Sh. 9	Sh. 10	Sh. 11
1	0.00	0.00	0.00	19.99	76.31	48.34	27.46	66.71	366.98	0.00	19.88
2	0.00	0.00	0.00	7.77	38.15	43.03	35.16	0.00	0.00	0.00	19.88

Total utilization percentage for section

Sh. 1	Sh. 2	Sh. 3	Sh. 4	Sh. 5	Sh. 6	Sh. 7	Sh. 8	Sh. 9	Sh. 10	Sh. 11
0	0	0	19.99 (1)	76.31 (1)	48.34 (1)	35.16 (2)	66.71 (1)	366.98 (1)	0	19.88 (1)

Figure 4.29

Utilization percentage for the section is computed for every check (see Fig.4.28).
Output data is presented as the report file either in HTML or RTF format (see Fig.4.29).

Reinforced concrete (RC) structures

Reinforced concrete (RC) structures

This chapter contains modules for analysis of different types of reinforced concrete structures, such as bars, beams, slabs, shells.

Properties of concrete;
Table of reinforcement;
Anchorage of reinforcement by DSTU 3760-07;
Sections of RC elements;
Reinforced concrete shell;
Reinforced concrete wall-beam;
Reinforced concrete slab;
Principal and equivalent stresses in reinforced concrete structures;
Strengthening with composite materials;
Strength of reinforced concrete butt joint in shear;
Concrete sections with fibre-reinforced polymer (FRP) bars;
Concrete pipe sections;
Composite steel and concrete columns;
Column strengthened with composed materials;
Composite steel and concrete slabs.

Figure 5.1

The programs of this chapter are based on the following regulations:
SNIP 2.03.01–84* 'Concrete and reinforced concrete structures';

Manual on design of concrete and reinforced concrete structures from heavyweight and lightweight concrete without prestressing of reinforcement (to SNIP 2.03.01–84);
SNIP 51–01–2003 'Concrete and reinforced concrete structures';
SP 52–101–2003 'Concrete and reinforced concrete structures without prestressing of reinforcement';
SP 63.13330.2012 'Concrete and reinforced concrete structures. General rules';
Manual on design of concrete and reinforced concrete structures from heavyweight concrete without prestressing of reinforcement (to SP 52–101–2003);
Eurocode 2. Design of concrete structures;
DSTU 3760–98. Rolled rebars for reinforced concrete structures;
DSTU 3760–2006. Rolled rebars for reinforced concrete structures;
TSN 102–00*. Reinforced concrete structures with reinforcement of class A500C and A400C.

Properties of concrete

The module provides reference materials about normative and design strength of concrete for ultimate and serviceability limit states according to SNIP 2.03.01–84* ''Concrete and reinforced concrete structures'.

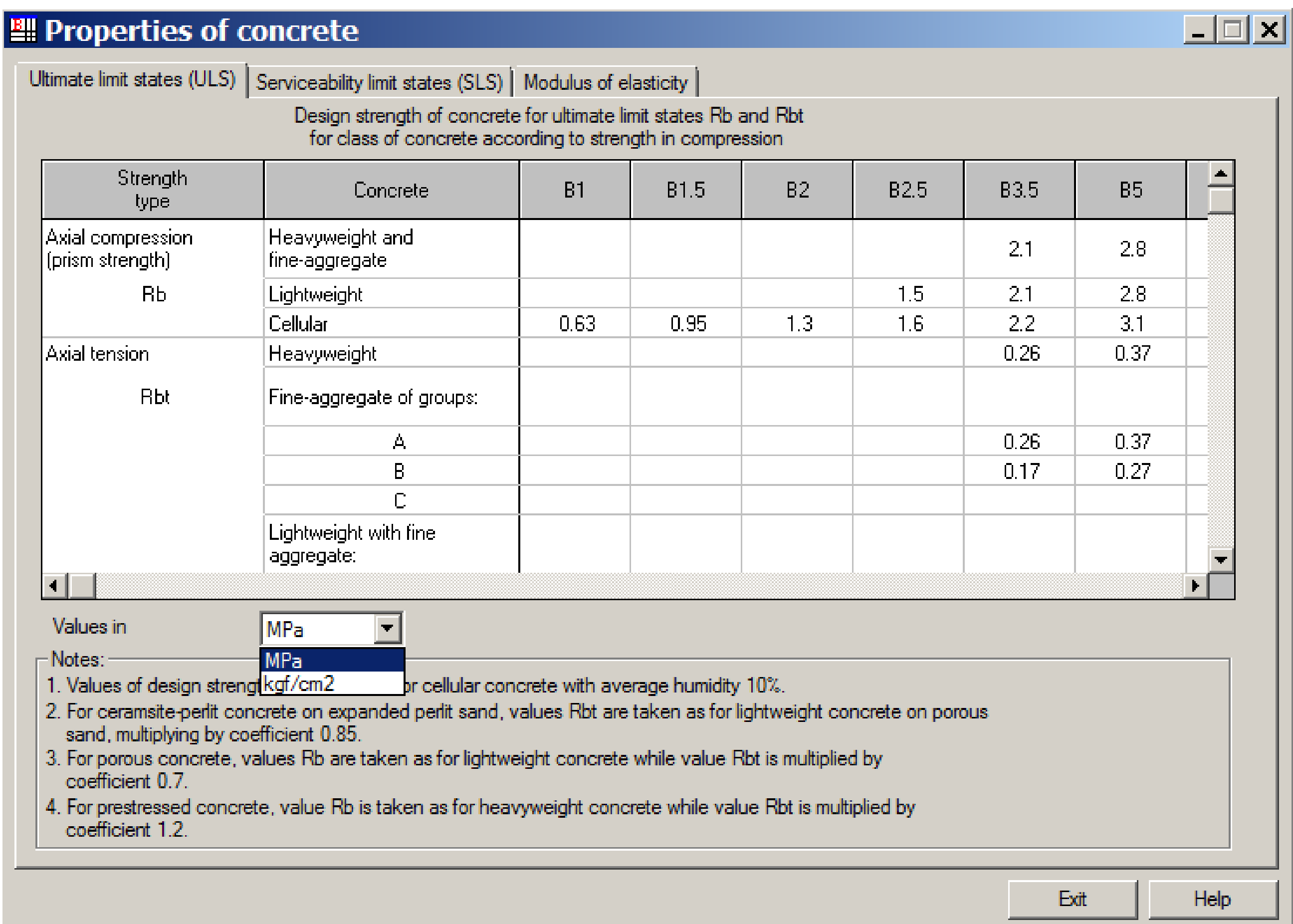
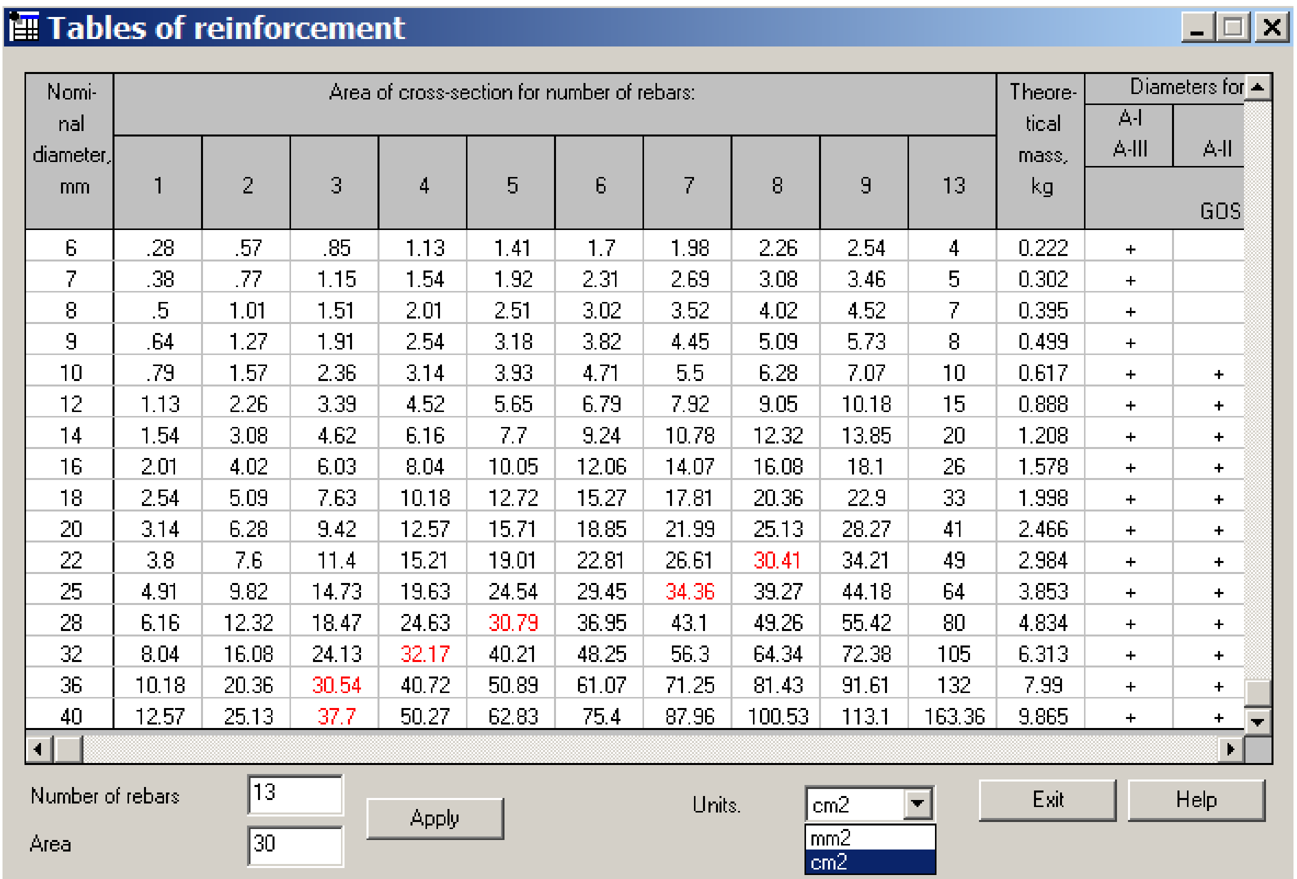


Figure 5.2

To preview properties of concrete by ultimate or serviceability limit states as well as elasticity moduli, click appropriate tabs in the dialog box.
It is possible to preview concrete properties either in MPa or in kgf/cm^2.

Table of reinforcement

The module provides reference materials about design area and theoretical mass per running metre of reinforcement depending on number and diameter of rebars according to SNIP 2.03.01–84* "Concrete and reinforced concrete structures'.
Program provides several computation tools. You could determine number and diameters of rebars for the specified area and preview area of cross-section for rebars depending on defined number of rebars.



Nominal diameter, mm	Area of cross-section for number of rebars: 1	2	3	4	5	6	7	8	9	13	Theoretical mass, kg	Diameters for A-I A-III	A-II GOS
6	.28	.57	.85	1.13	1.41	1.7	1.98	2.26	2.54	4	0.222	+	
7	.38	.77	1.15	1.54	1.92	2.31	2.69	3.08	3.46	5	0.302	+	
8	.5	1.01	1.51	2.01	2.51	3.02	3.52	4.02	4.52	7	0.395	+	
9	.64	1.27	1.91	2.54	3.18	3.82	4.45	5.09	5.73	8	0.499	+	
10	.79	1.57	2.36	3.14	3.93	4.71	5.5	6.28	7.07	10	0.617	+	+
12	1.13	2.26	3.39	4.52	5.65	6.79	7.92	9.05	10.18	15	0.888	+	+
14	1.54	3.08	4.62	6.16	7.7	9.24	10.78	12.32	13.85	20	1.208	+	+
16	2.01	4.02	6.03	8.04	10.05	12.06	14.07	16.08	18.1	26	1.578	+	+
18	2.54	5.09	7.63	10.18	12.72	15.27	17.81	20.36	22.9	33	1.998	+	+
20	3.14	6.28	9.42	12.57	15.71	18.85	21.99	25.13	28.27	41	2.466	+	+
22	3.8	7.6	11.4	15.21	19.01	22.81	26.61	30.41	34.21	49	2.984	+	+
25	4.91	9.82	14.73	19.63	24.54	29.45	34.36	39.27	44.18	64	3.853	+	+
28	6.16	12.32	18.47	24.63	30.79	36.95	43.1	49.26	55.42	80	4.834	+	+
32	8.04	16.08	24.13	32.17	40.21	48.25	56.3	64.34	72.38	105	6.313	+	+
36	10.18	20.36	30.54	40.72	50.89	61.07	71.25	81.43	91.61	132	7.99	+	+
40	12.57	25.13	37.7	50.27	62.83	75.4	87.96	100.53	113.1	163.36	9.865	+	+

Figure 5.3

To preview the **Area of cross-section for number of rebars** that differ from 1-9, in the **Number of rebars** box, define any whole number.
To preview number of rebars of various diameters that satisfy certain area, in the **Area** box, define appropriate value. Click **Apply**.
Then in the **Area of cross-section for number of rebars** columns, some values will be coloured red. Necessary number of rebars for the certain diameter if the area of appropriate diameters of rebars is greater than predefined value.

It is possible to preview concrete properties either in mm^2 or cm^2.

Anchorage of reinforcement by DSTU 3760-07

This module enables you to determine anchorage length of rebars (length of rebar outside the section where it should take forces acting in the section).

Anchorage of reinforcement is carried out according to DSTU 3760–07 'Reinforcing bars for RC structures'.

Length of anchorage in reinforcing bars is computed for ***horizontal beams and slabs***.

Length of anchorage is computed for separate and for double rebars. Rebars in which clear spacing is less than their diameter are considered as *double rebars*.

Concrete cover for the corner rebar is selected from the two values: vertical and horizontal, ***whichever is less***. For intermediate rebars, concrete cover is specified up to the side at which it is located.

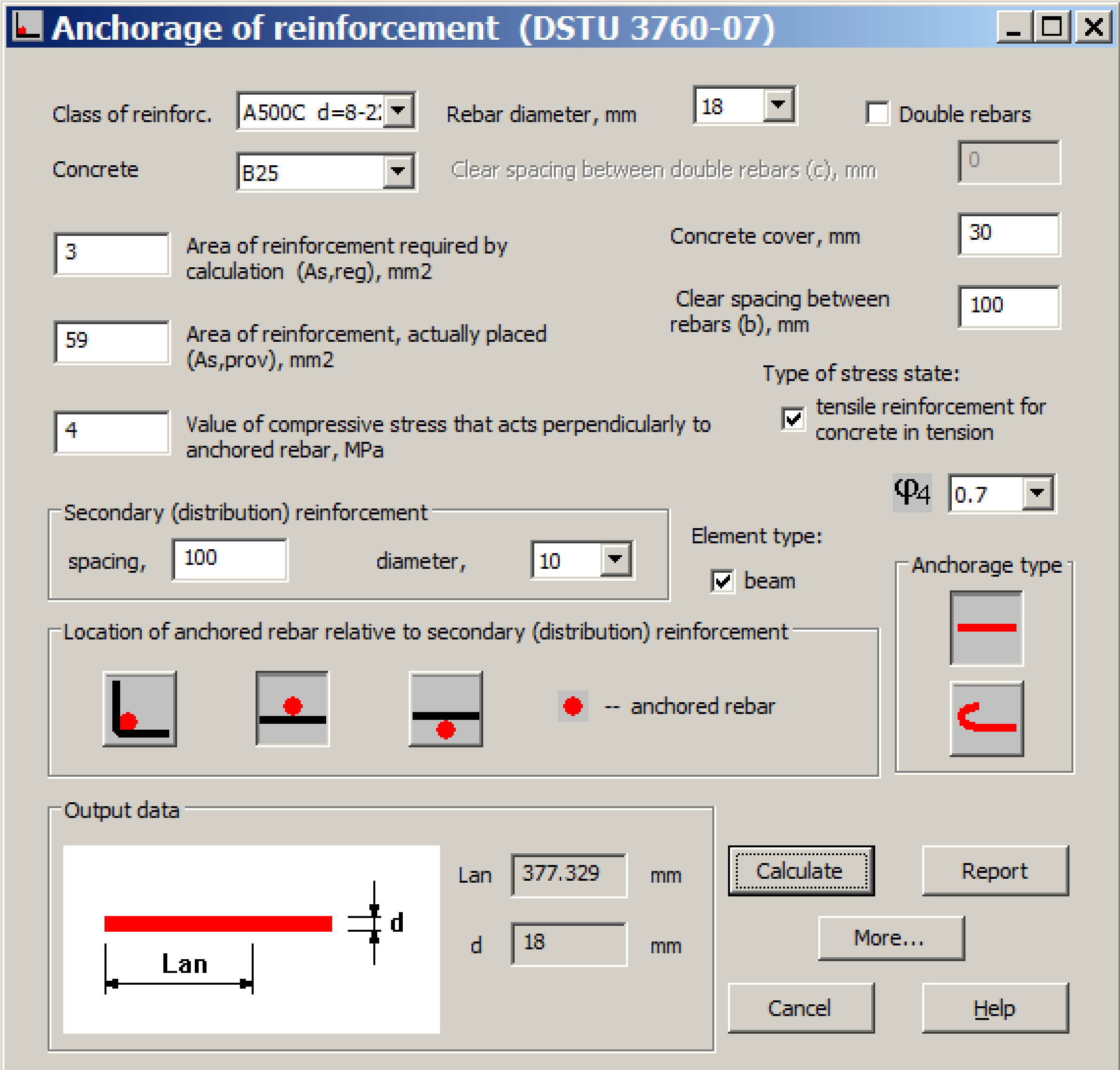


Figure 5.4

Area of distribution reinforcement ($\sum A_{st}$) located along the length l_{an} is calculated with computer program by the successive approximations method based on the spacing and diameter of distribution reinforcement.

Value of compressive stress in concrete (σ_b) that acts perpendicularly to anchored rebar is obtained by dividing support reaction into support area of an element.

Coefficients φ_1 and φ_2 are computed depending on the type of anchorage and ratio of anchored rebar diameter to distribution rebar diameter.

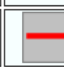
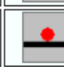
Coefficient φ_4 takes into account location of reinforcement during placing of concrete; coefficient is determined according to the table 12 (Recommendations).

Input data should be defined in the dialog box. then click **Calculate**.
To display the output data together with temporary computation data, click **More**.
To display the report file (with input & output data) in HTML format, click **Report**.
The report is presented below.

Input data

Code	DSTU 3760-07

Materials	
Diameter of anchoring rebar, mm	18
Grade of concrete	B25
Class of reinforcement	A500C d=8-22

Concrete cover, mm	30
Clear distance between rebars, mm	100
Element type: beam	
Tensile reinforcement in tensioned concrete	
Area of reinforcement required by calculation, mm2	3
Area of reinforcement actually arranged, mm2	59
Value of compressive stress that acts perpendicularly to anchoring rebar, MPa	4
Spacing of secondary (distribution) reinforcement, mm	100
Diameter of secondary (distribution) reinforcement, mm	10
Anchorage type	
Location of anchoring rebar relative to secondary (distribution) reinforcement	

Output data

lb	Basic length of anchorage, mm	1257.76
lan,min	Min length of anchorage, mm	377.329
Lan_	Effective length of anchorage, mm	53.5639
Lan	Length of anchorage, mm	377.329

Sections of RC elements

The module enables you to carry out analysis of reinforcement in sections of RC elements by ultimate (ULS) and serviceability limit states (SLS) according to the following building codes.

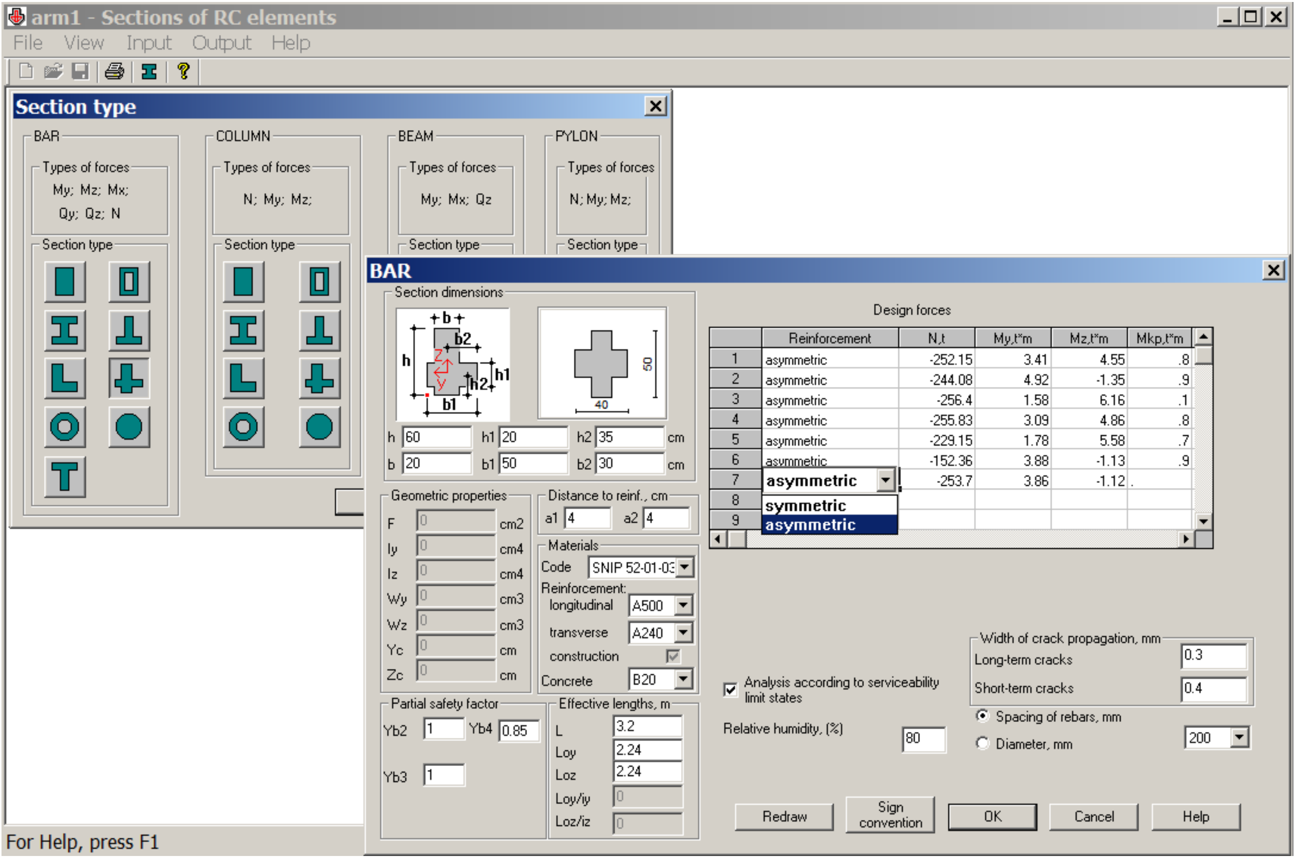
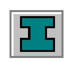


Figure 5.5

Input data

On the INPUT menu, click **Input data and calculation** (button  on the toolbar).
In the **Section type** dialog box, select appropriate **type of section** by type of forces and click **OK**.
In the **Bar**, **Column**, **Beam** or **Pylon** dialog box (depending on defined **Type of forces**), in the **Materials** area, define the building code by which computation should be carried out.
In the dialog box define appropriate section dimensions; distances from the gravity centre of reinforcement to the bottom (top) face (a1) and to the side face (a2); define all necessary properties according to the building code.

When you click **Redraw**, schematic presentation of the section will be updated according to appropriate dimensions and in the **Geometric properties** area you will see geometric properties of the section (they are computed automatically).

To find out schematic presentation of positive directions for forces in element section, click **Sign convention**.

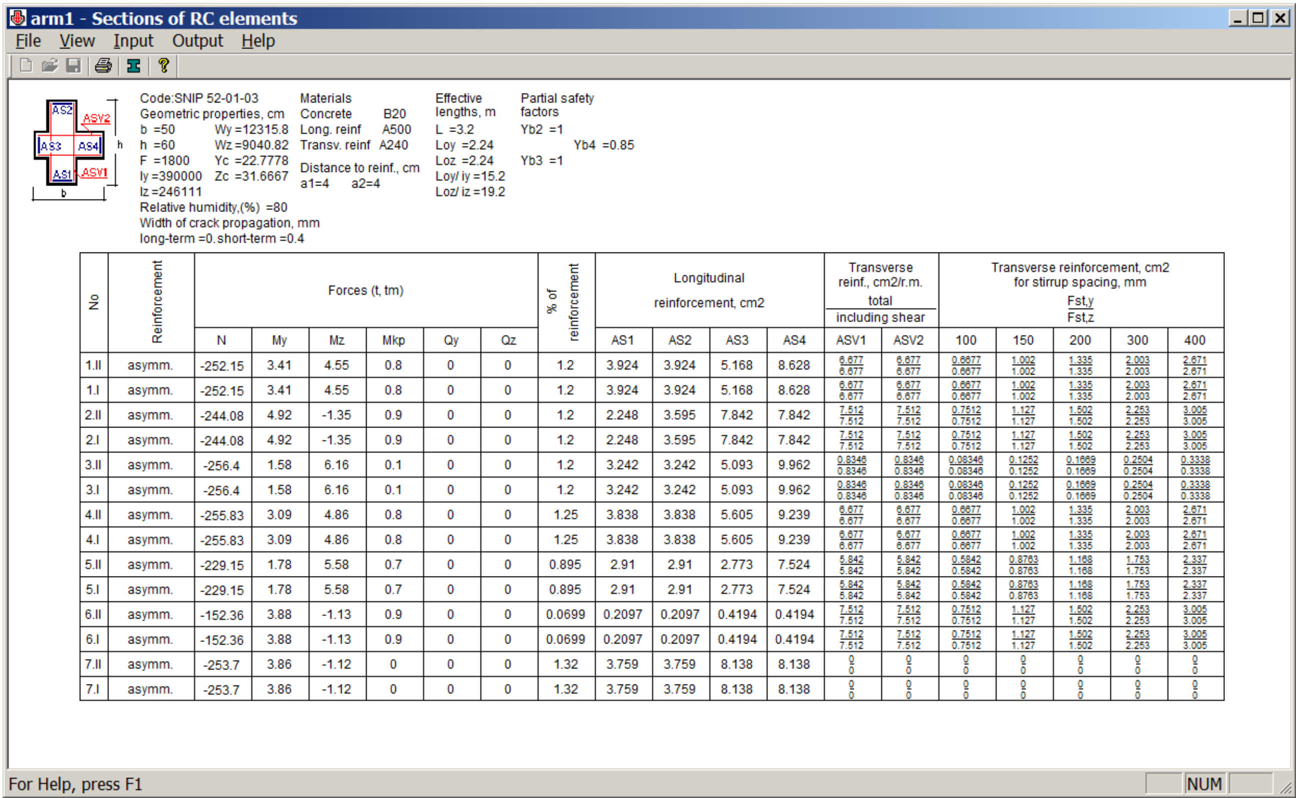
Define appropriate data for **Design forces**.

To start calculation of reinforcement, click **OK**.

 www.liraland.com

Output data

To present analysis results as a report (input & output data for the problem) file in HTML format, on the OUTPUT menu, click **Report**.



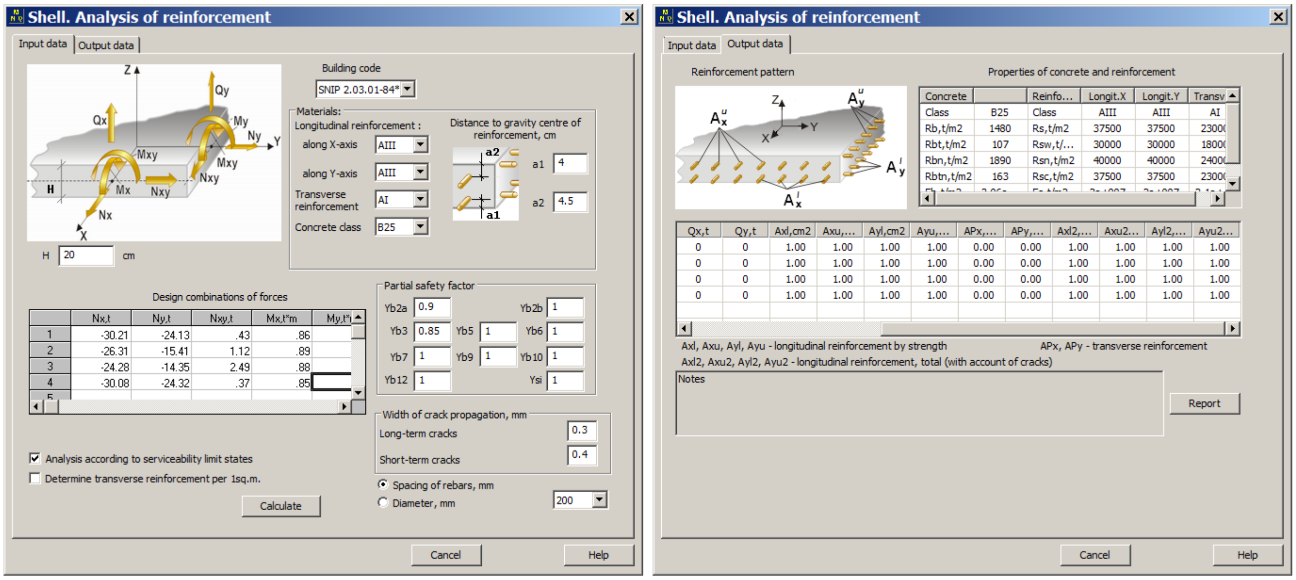
No	Reinforcement	Forces (t, tm)						% of reinforcement	Longitudinal reinforcement, cm2				Transverse reinf., cm2/r.m. total / including shear		Transverse reinforcement, cm2 for stirrup spacing, mm Fst,y / Fst,z				
		N	My	Mz	Mkp	Qy	Qz		AS1	AS2	AS3	AS4	ASV1	ASV2	100	150	200	300	400
1.II	asymm.	-252.15	3.41	4.55	0.8	0	0	1.2	3.924	3.924	5.168	8.628	6.677 6.677	6.677 6.677	0.6677 0.6677	1.002 1.002	1.335 1.335	2.003 2.003	2.671 2.671
1.I	asymm.	-252.15	3.41	4.55	0.8	0	0	1.2	3.924	3.924	5.168	8.628	6.677 6.677	6.677 6.677	0.6677 0.6677	1.002 1.002	1.335 1.335	2.003 2.003	2.671 2.671
2.II	asymm.	-244.08	4.92	-1.35	0.9	0	0	1.2	2.248	3.595	7.842	7.842	7.512 7.512	7.512 7.512	0.7512 0.7512	1.127 1.127	1.502 1.502	2.253 2.253	3.005 3.005
2.I	asymm.	-244.08	4.92	-1.35	0.9	0	0	1.2	2.248	3.595	7.842	7.842	7.512 7.512	7.512 7.512	0.7512 0.7512	1.127 1.127	1.502 1.502	2.253 2.253	3.005 3.005
3.II	asymm.	-256.4	1.58	6.16	0.1	0	0	1.2	3.242	3.242	5.093	9.962	0.8346 0.8346	0.8346 0.8346	0.08346 0.08346	0.1252 0.1252	0.1669 0.1669	0.2504 0.2504	0.3338 0.3338
3.I	asymm.	-256.4	1.58	6.16	0.1	0	0	1.2	3.242	3.242	5.093	9.962	0.8346 0.8346	0.8346 0.8346	0.08346 0.08346	0.1252 0.1252	0.1669 0.1669	0.2504 0.2504	0.3338 0.3338
4.II	asymm.	-255.83	3.09	4.86	0.8	0	0	1.25	3.838	3.838	5.605	9.239	6.677 6.677	6.677 6.677	0.6677 0.6677	1.002 1.002	1.335 1.335	2.003 2.003	2.671 2.671
4.I	asymm.	-255.83	3.09	4.86	0.8	0	0	1.25	3.838	3.838	5.605	9.239	6.677 6.677	6.677 6.677	0.6677 0.6677	1.002 1.002	1.335 1.335	2.003 2.003	2.671 2.671
5.II	asymm.	-229.15	1.78	5.58	0.7	0	0	0.895	2.91	2.91	2.773	7.524	5.842 5.842	5.842 5.842	0.5842 0.5842	0.8763 0.8763	1.168 1.168	1.753 1.753	2.337 2.337
5.I	asymm.	-229.15	1.78	5.58	0.7	0	0	0.895	2.91	2.91	2.773	7.524	5.842 5.842	5.842 5.842	0.5842 0.5842	0.8763 0.8763	1.168 1.168	1.753 1.753	2.337 2.337
6.II	asymm.	-152.36	3.88	-1.13	0.9	0	0	0.0699	0.2097	0.2097	0.4194	0.4194	7.512 7.512	7.512 7.512	0.7512 0.7512	1.127 1.127	1.502 1.502	2.253 2.253	3.005 3.005
6.I	asymm.	-152.36	3.88	-1.13	0.9	0	0	0.0699	0.2097	0.2097	0.4194	0.4194	7.512 7.512	7.512 7.512	0.7512 0.7512	1.127 1.127	1.502 1.502	2.253 2.253	3.005 3.005
7.II	asymm.	-253.7	3.86	-1.12	0	0	0	1.32	3.759	3.759	8.138	8.138	0 0	0 0	0 0	0 0	0 0	0 0	0 0
7.I	asymm.	-253.7	3.86	-1.12	0	0	0	1.32	3.759	3.759	8.138	8.138	0 0	0 0	0 0	0 0	0 0	0 0	0 0

Figure 5.6

To print the output data directly from the program, on the FILE menu, click **Print**.

Reinforced concrete shell

This module enables you to analyse reinforcement in plate RC structures with complex stress state (shells) by ultimate and serviceability limit states according to different building codes.



a)

b)

Figure 5.7

Input data

To define input data, follow these steps:

– In the **Shell. Analysis of reinforcement** dialog box, select the building code for calculation from appropriate list.

– In the **Materials** area, select classes of reinforcement along certain axes indicated on the model; define distances from extreme lower (*a1*) and extreme upper (*a2*) fiber of section to gravity centres of reinforcement; define **class of concrete**.

– In the **Partial safety factors** area, define required parameters according to selected building code (additional data is different and depends on the selected building code).

– Define thickness for the plate element in the **H** box.

– When the **Design combinations of forces** table if filled in, click **Calculate**.

Output data

When calculation is complete, on the **Output data** tab you will see determined reinforcement as well as input data.

To present a report document in HTML format, click **Report**.

Reinforced concrete wall-beam

This module enables you to analyse reinforcement in plate RC structures with complex stress state (wall-beam) by ultimate and serviceability limit states according to different building codes.

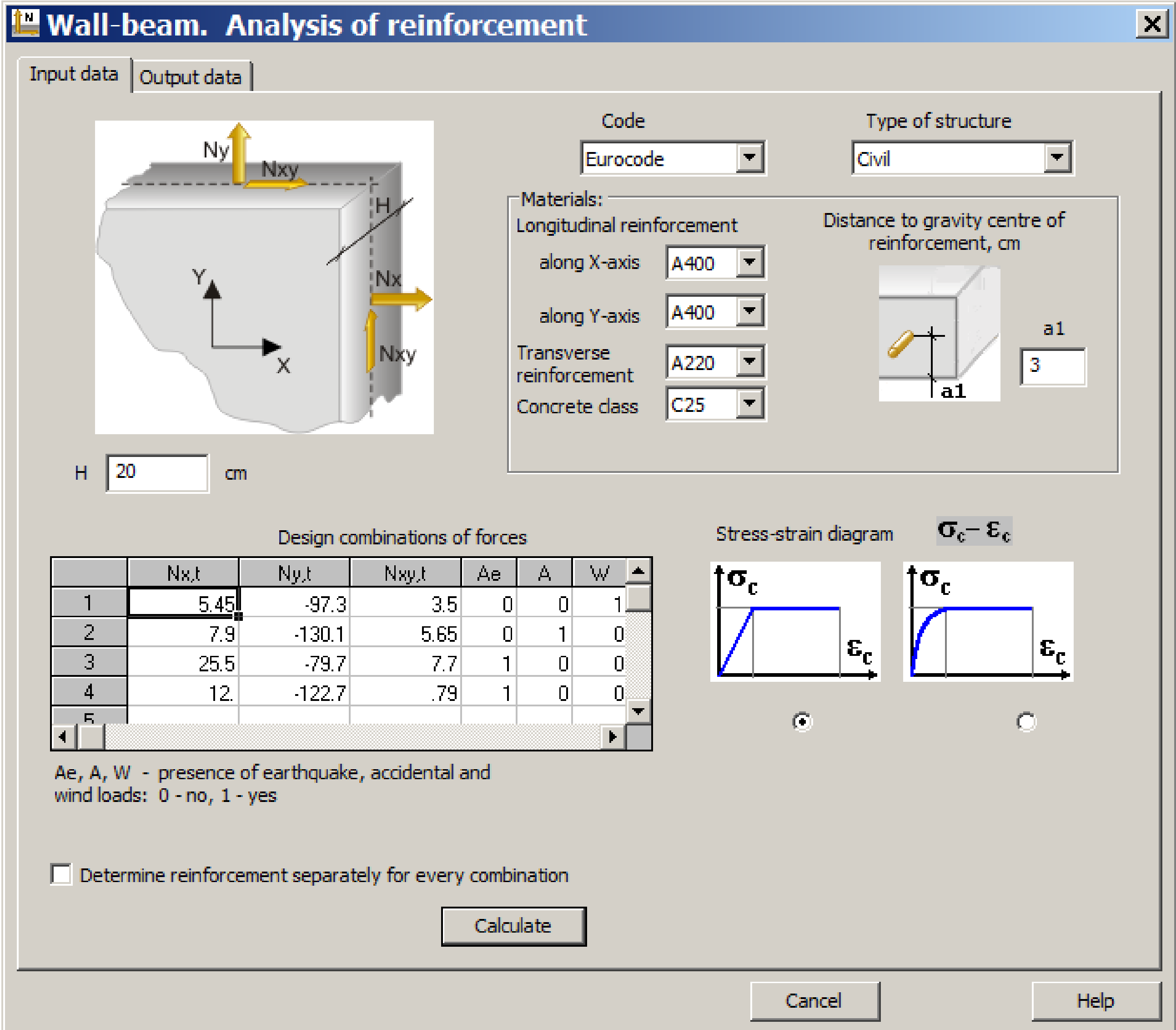

Wall-beam. Analysis of reinforcement
Input data
Output data
Ny
Nxy
H
Nx
Nxy
Y
X
Code
Eurocode
Type of structure
Civil
Materials:
Longitudinal reinforcement
along X-axis
A400
along Y-axis
A400
Transverse reinforcement
A220
Concrete class
C25
Distance to gravity centre of reinforcement, cm
a1
3
H
20
cm
Design combinations of forces
Nx,t
Ny,t
Nxy,t
Ae
A
W
1 5.45 -97.3 3.5 0 0 1
2 7.9 -130.1 5.65 0 1 0
3 25.5 -79.7 7.7 1 0 0
4 12. -122.7 .79 1 0 0
Ae, A, W - presence of earthquake, accidental and wind loads: 0 - no, 1 - yes
Stress-strain diagram
σc– εc
Determine reinforcement separately for every combination
Calculate
Cancel
Help

Figure 5.8 a)

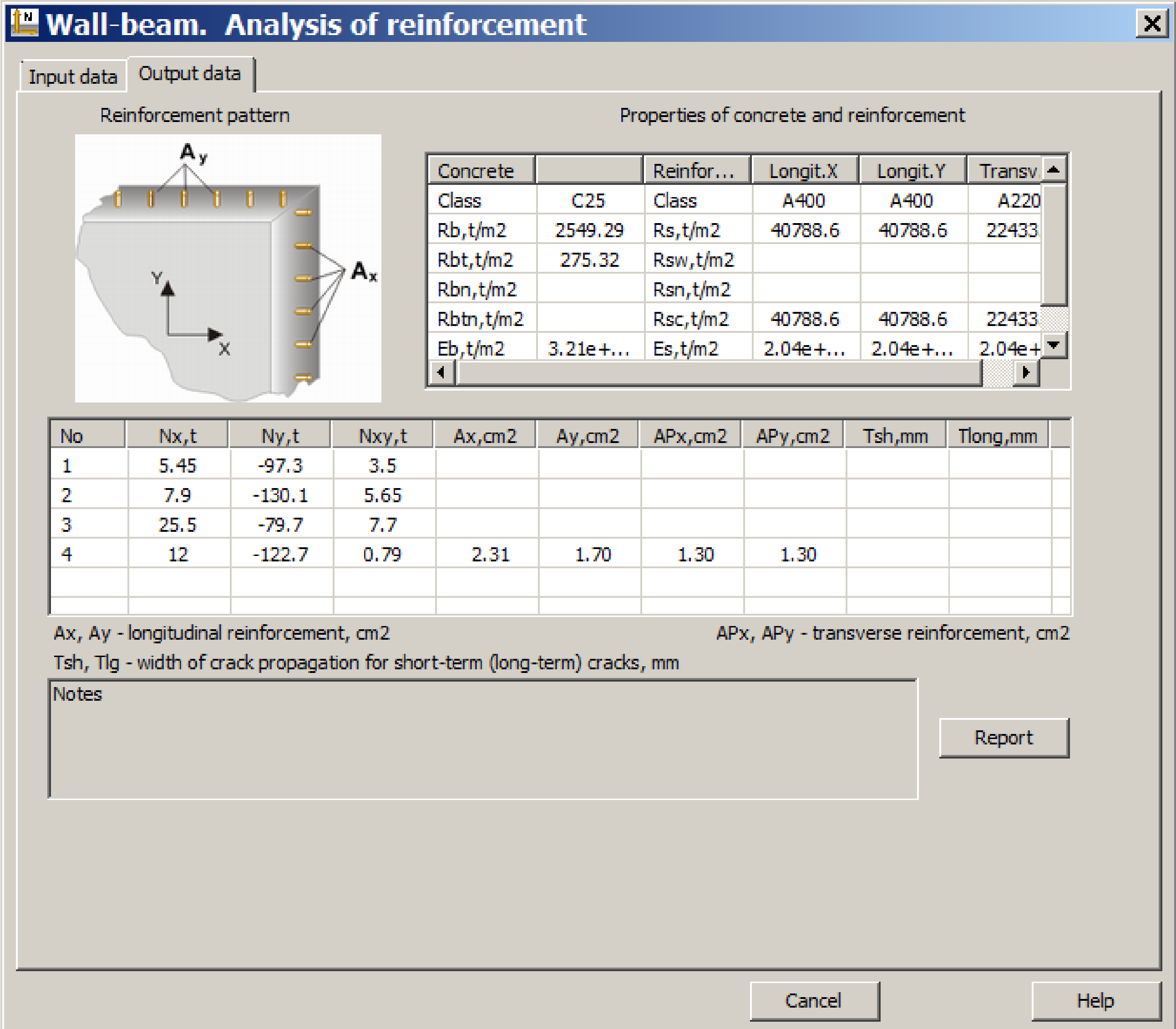


Figure 5.8 b)

Input data

To define input data, follow these steps:

– In the **Wall-beam. Analysis of reinforcement** dialog box, select the building code for calculation from appropriate list.

– In the **Materials** area, select classes of reinforcement along certain axes indicated on the model; define distances from extreme lower (*a1*) fiber of section to gravity centres of reinforcement; define **class of concrete**.

– In the **Partial safety factors** area, define required parameters according to selected building code (additional data is different and depends on the selected building code).

– Define thickness for the plate element in the **H** box.

– When the **Design combinations of forces** table if filled in, click **Calculate**.

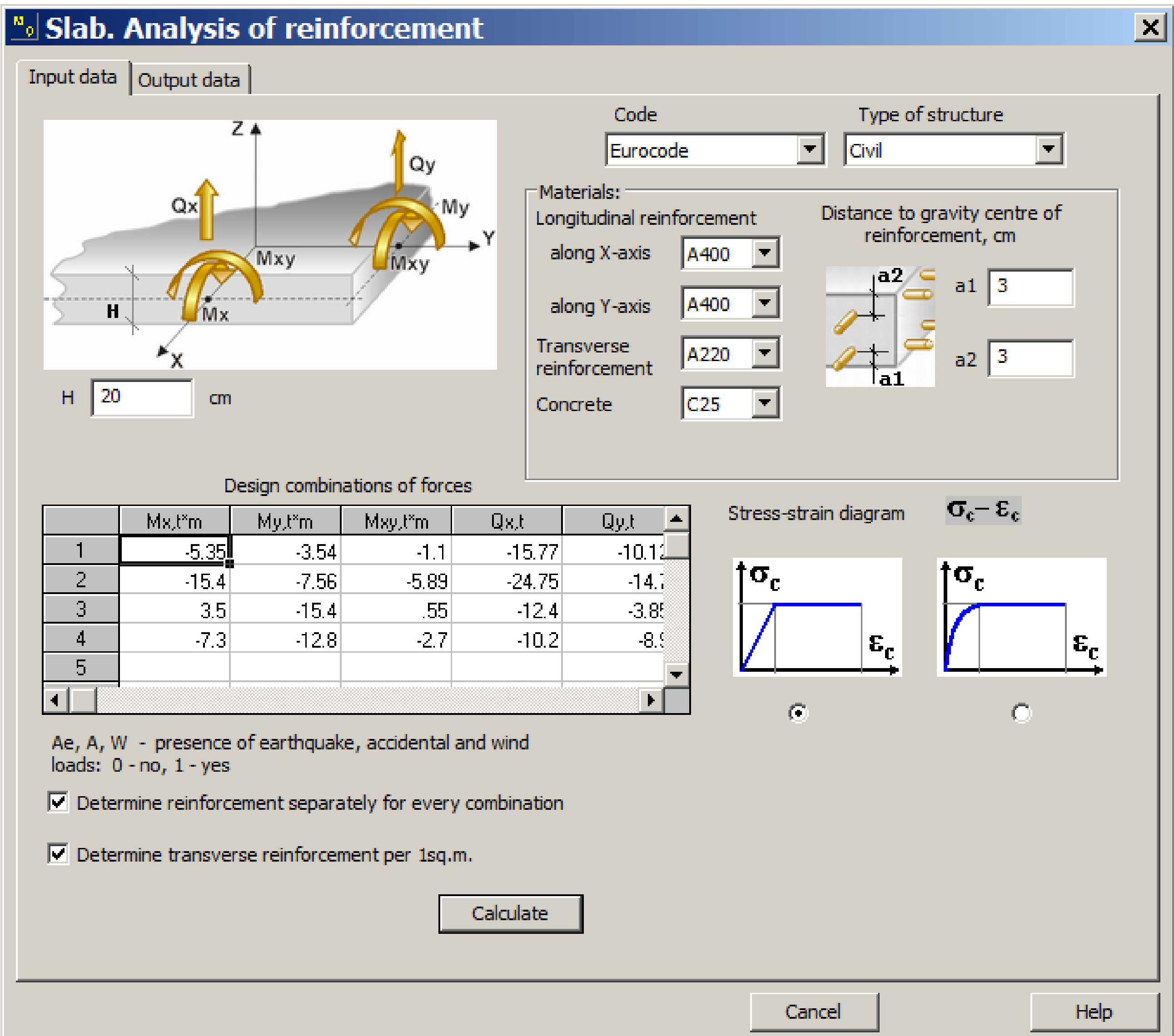
 www.liraland.com

Output data

When calculation is complete, on the **Output data** tab you will see determined reinforcement as well as input data.
To present a report document in HTML format, click **Report**.

Reinforced concrete slab

This module enables you to analyse reinforcement in plate RC structures with stress state of flexural elements by ultimate and serviceability limit states according to different building codes.



	Mx,t*m	My,t*m	Mxy,t*m	Qx,t	Qy,t
1	-5.35	-3.54	-1.1	-15.77	-10.1
2	-15.4	-7.56	-5.89	-24.75	-14.
3	3.5	-15.4	.55	-12.4	-3.8
4	-7.3	-12.8	-2.7	-10.2	-8.
5					

Figure 5.9 a)

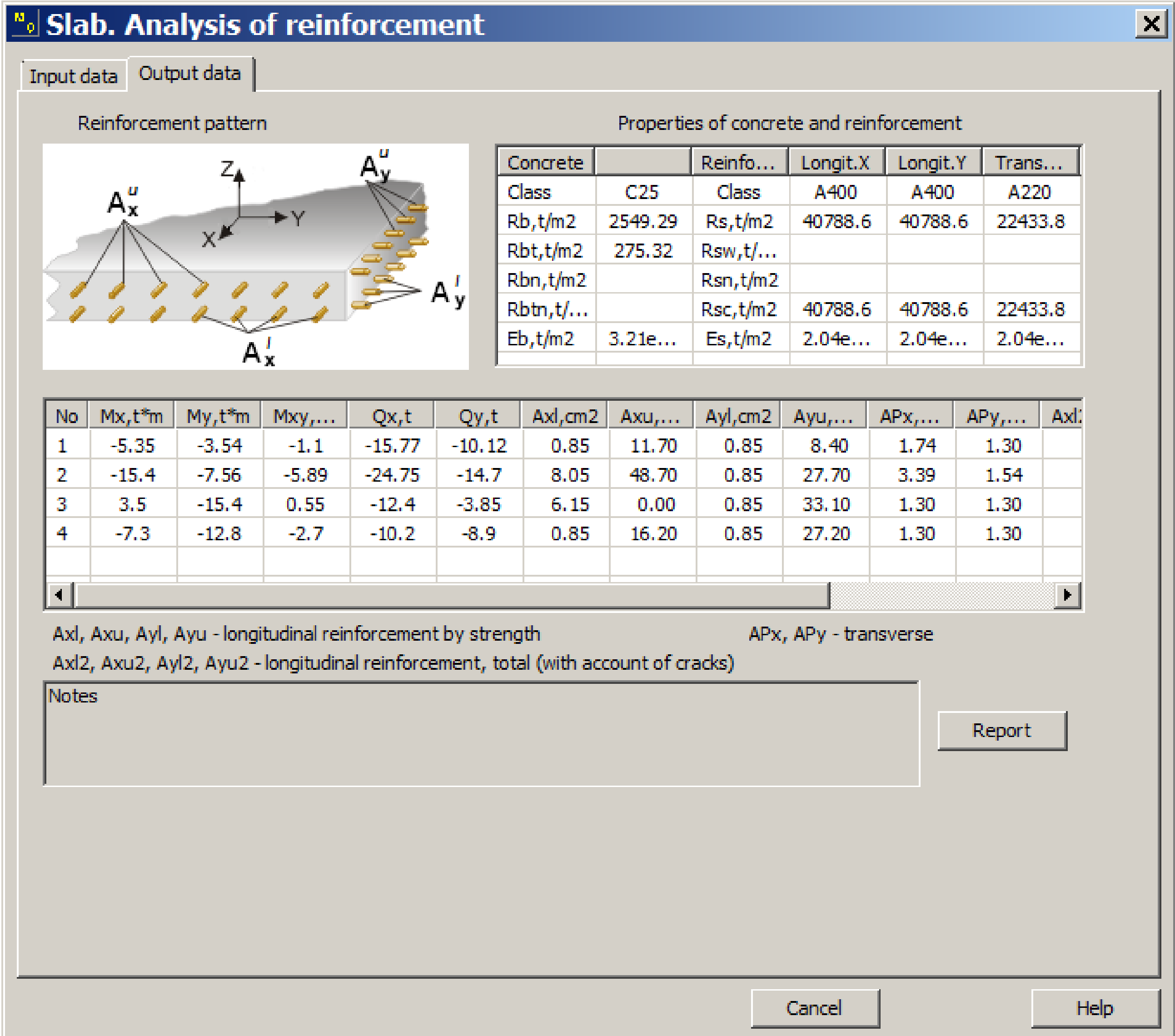


Figure 5.9 b)

Input data

To define input data, follow these steps:

– In the **Slab. Analysis of reinforcement** dialog box, select the building code for calculation from appropriate list.

– In the **Materials** area, select classes of reinforcement along certain axes indicated on the model; define distances from extreme lower (*a1*) and extreme upper (*a2*) fiber of section to gravity centres of reinforcement; define **class of concrete**.

– In the **Partial safety factors** area, define required parameters according to selected building code (additional data is different and depends on the selected building code).

– Define thickness for the plate element in the **H** box.

– When the **Design combinations of forces** table if filled in, click **Calculate**.

 www.liraland.com

Output data

When calculation is complete, on the **Output data** tab you will see determined reinforcement as well as input data.
To present a report document in HTML format, click **Report**.

Principal and equivalent stresses in reinforced concrete structures

The module enables you to calculate principal stresses σ_1, σ_2, σ_3 by the certain stress tensor {σ_x, σ_y, σ_z, τ_{xy}, τ_{xz}, τ_{yz}}.
To find out roots of principal stresses, cubic equation is solved based on stress matrix [σ] in the form:

$$[\sigma]=\begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

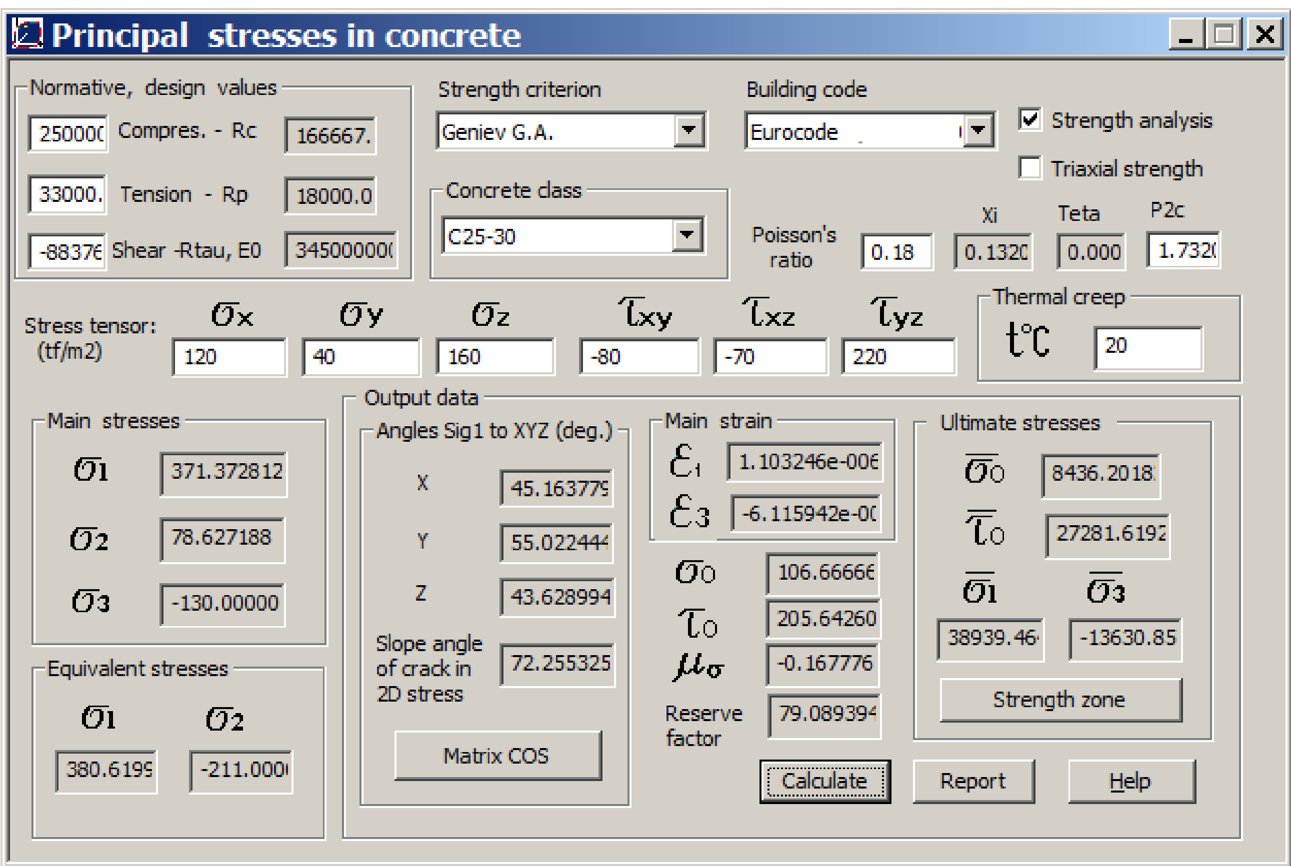


Рис. 5.10

In this program you could also check strength by the certain stress tensor.

Input data

For concrete it is necessary to define class and type of concrete according to SNIP 2.03.01-84*, for example:

– B25 HA – concrete class 25, heavyweight of groups A,B,C (fine-aggregate of groups A,B,C with thermal treatment B25 FAT), etc.;

– B3.5 L1400 – concrete class 3.5 lightweight, with unit weight 800-1800 kg/m^3;

– B2.5 C800 – concrete class 2.5 cellular, with unit weight 500-900 kg/m^3;

– strength parameters: strength in compression – **Rc**, strength in tension – **Rp**, are taken automatically from the database;

– to define arbitrary strength parameters for concrete: in the **Concrete class** box, select **Nonstandard concrete**; and then define strength parameters for concrete: **Rc > Rr** (tf/m^2).

Output data

When the **Strength analysis** check box is selected, after strength analysis, the following data is displayed:

σ_1, σ_3 – ultimate allowed values of principal stresses;

σ_0, τ_0 – ultimate axial and shear octahedral stresses;

– reserve factor (analysis by serviceability limit state is supposed, that is, normative strength parameters of material are considered in analysis);

– Lode-Nadai parameter is computed as: $\mu_\sigma = \dfrac{(2\sigma_2 - \sigma_1 - \sigma_3)}{(\sigma_1 - \sigma_3)}$;

To generate report file in HTML format, click **Report**. The report file contains the following data:

– principal stresses as: σ1 > σ2 > σ3;

– matrix of directional cosines MC(3,3) – orientation of principal stresses in the space relative to the X,Y,Z axes:

$$MC = \begin{bmatrix} l1 & m1 & n1 \\ l2 & m2 & n2 \\ l3 & m3 & n3 \end{bmatrix}$$

– slope angles AL1, AL2, AL3 (l1, l2, l3) of principal stress σ_1 to the X, Y, Z-axes; angles define orientation of the crack plane in the space;

– slope angle α3 of principal stress σ_3 to the X-axis; angle defines orientation of the crack plane relative to the X-axis ;

– Euler angles TETA, PSI, FI (θ, ψ, φ) that define location of principal stresses σ_1, σ_2, σ_3 relative to the X, Y, Z-axes of global coordinate system;

– additional strength parameters:

  – strength in biaxial uniform compression – R2c = Rc*P2c,

  – strength in biaxial uniform tension – R2p.

– whether cracks are present TB=1, 2, 3 – one, two, three; TB=10 (20) – destruction of element in compression. TB=0 – no cracks.

Strengthening with composite materials

This module enables you to check strength of reinforced concrete sections in case the structures are strengthened with composite materials.

Input data and calculation

In the dialog box select the section type (rectangle, T-section with flange at the bottom, T-section with flange at the top, I-section) and define its dimensions. To check T-section or I-section with trapezoidal flanges, define the average values for flange overhang.
Select appropriate composite material or define arbitrary material (if **User-defined** option is selected in the list). You should also define design forces.
The following types of composite materials are available:
ASLAN, FibARM®, ITECWRAP®, Mapei®, S&P®, Sika®, Tyfo®, УОЛ.
It is also possible to define non-standard data.

The following regulations are realized in the program: *'Manual on strengthening of RC structures with composite materials'* and *SP 52-101-2003 'Concrete and reinforced concrete structures without prestressing of reinforcement'*.
The following classes of concrete are available: B10, B15, B20, B25, B30, B35, B40, B45, B50, B55, B60.
The following classes of reinforcement are available: A240, A300, A400, A500, B500.
Max diameter of reinforcement - 40 mm.
Distance to reinforcement - distance from gravity centre of reinforcement group up to the nearest side (lower reinforcement to the lower side, upper reinforcement to the upper one).

Physical and mechanical properties of suggested reinforcement types FRP (composite materials) are taken from Appendix 5 *'Manual on strengthening of RC structures with composite materials'*.

Define thickness of monolayer, modulus of elasticity for composite material, strength in tension and ultimate tensile strength.
Note. It is recommended that you should contact the manufacturer to find out parameters of materials you selected.

Partial safety factor for FRP (fiber-reinforced polymer) is defined according to the table below.

Environmental conditions	Material	Laminates	Fabric
Inner premises	Carbon	0.95	0.9
	Glass	0.75	0.7
	Aramid	0.85	0.8
Outer structures	Carbon	0.85	0.8
	Glass	0.65	0.6

	Aramid	0.75	0.7
Corrosive medium	Carbon	0.85	0.8
	Glass	0.5	0.5
	Aramid	0.7	0.6

The dialog box where you define input data is presented below.

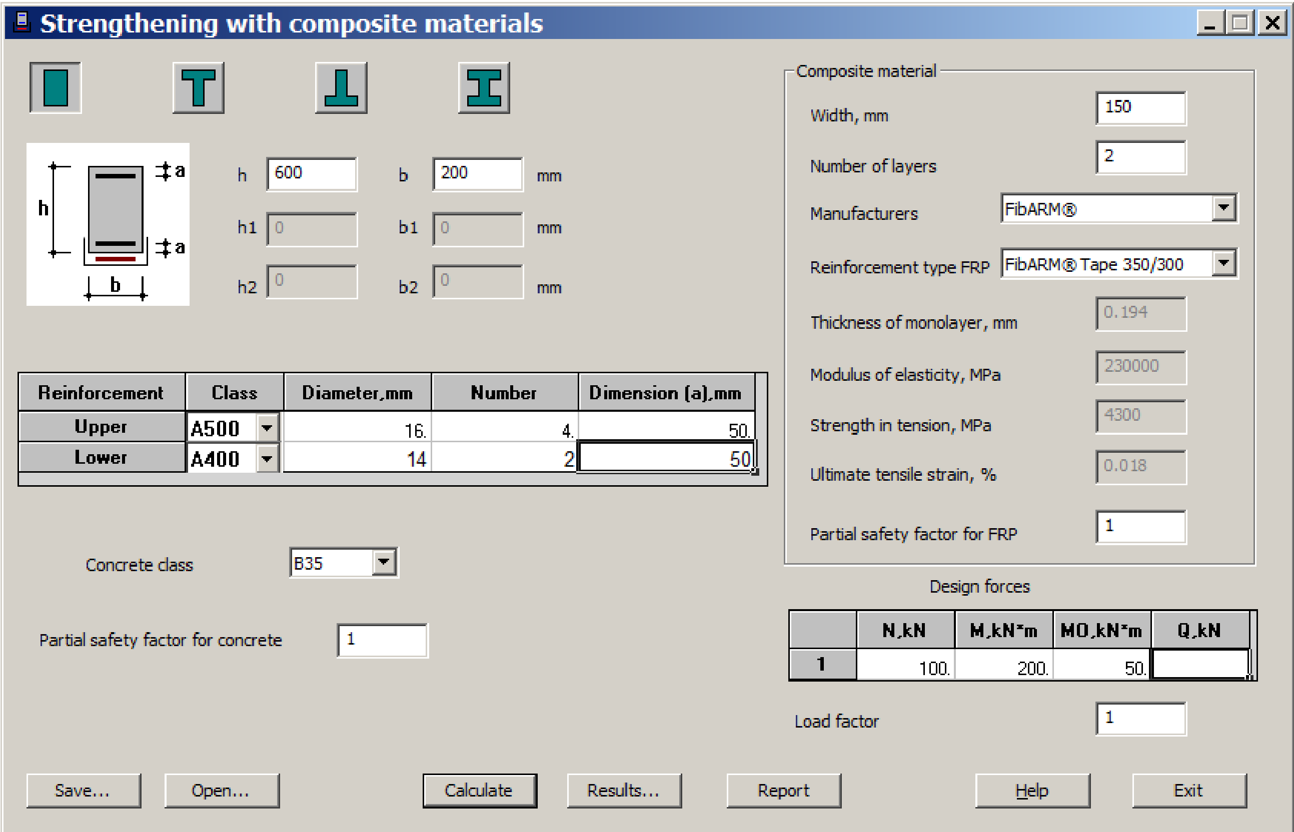


Figure 5.14

Define **design forces** in the appropriate table. To convert design values to normative values, the load factor γ_f will be applied.

N – axial force applied at the gravity centre, kN;

M – expected moment from external forces, kN*m;

M0 – moment from external forces before strengthening, kN*m;

Q – shear force, kN.

Sign convention: positive moment corresponds to tension in lower fiber (according to schematic presentation of section), positive axial force corresponds to compression.

Partial safety factor for concrete γ_{b2} is defined depending on the load pattern (by default, it is taken as equal to 1).

To make calculation, click **Calculate**.

Output data

To display **Output data** for the calculation, click **Results**.

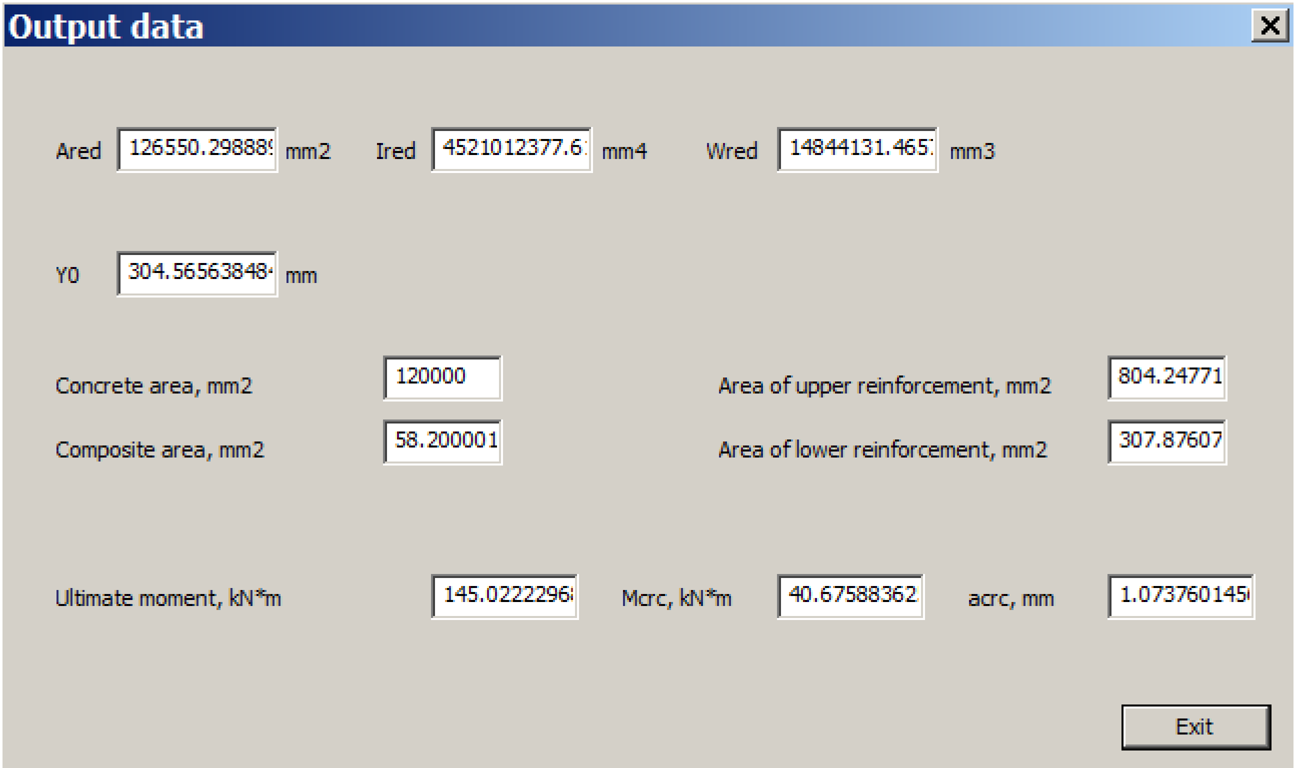


Figure 5.15

To save the model in **.cmp* format, click **Save**.
To open the input data of the saved model, click **Open**. The calculation should be made once again.
To present a report document in HTML format, click **Report**.

Notation in output data

Ared - area of equivalent section;
Ired - moment of inertia of equivalent section;
Wred - section modulus of equivalent section;
y0 - distance from lower fibre up to gravity centre of the section;
x - height of compression zone of concrete;
Mult - ultimate moment for the section;
Mcrc - moment in crack formation;
acrc - max width of crack propagation.

Important. If the **Mult** value in output data *is less than* defined value **M0** before strengthening *or equal to zero*, it is necessary to check defined parameters for concrete and reinforcement.

Concrete and reinforcement				
Reinforcement	Класс	Diameter,mm	Number	Dimension,mm
Upper	A400	18	4	50
Lower	A400	22	2	50

Concrete class	Eb, MPa	Rb, MPa	Rb,ser, MPa	Rbt,ser, MPa
B30	34500	17	22	1.75
Partial safety factor for concrete	1			

Composite material	
thickness, mm	0.111
width, mm	400
number of layers	2
type of reinforcement	FibARM® Tape 200/300
modulus of elasticity,MPa	230000
strength in tension,MPa	4300
max tensile strain,%	0.018
partial safety factor for FRP	1

Design combinations			
N,kN	M,kN*m	M0,Kn*m	Q,kN
500	220	50	0
Load factor			1

Output data

Area of effective section,mm2	Ared=250900
Moment of inertia of effective section,mm4	Ired=8.85101e+009
Section modulus of effective section,mm3	Wred=2.94269e+007
Centre of mass of section before strengthening,mm	Y0=300.78
Area of concrete,mm2	240000
Area of lower reinforcement,mm2	760.265
Area of upper reinforcement,mm2	1017.88
Area of composite,mm2	88.8
Ultimate moment, kN*m	Mult=295.308
Moment while crack formation, kN*m	Mcrc=110.14
Width of crack propagation,mm	acrc=0.483375

Figure 5.16

Strength of reinforced concrete butt joint in shear

The module enables you to check strength in shear for butt joints of precast and cast-in-situ RC structures as well as structures of cast-in-situ RC with additional layer for joint grouting.

The dialog box of the program is presented in Figure 5.11.

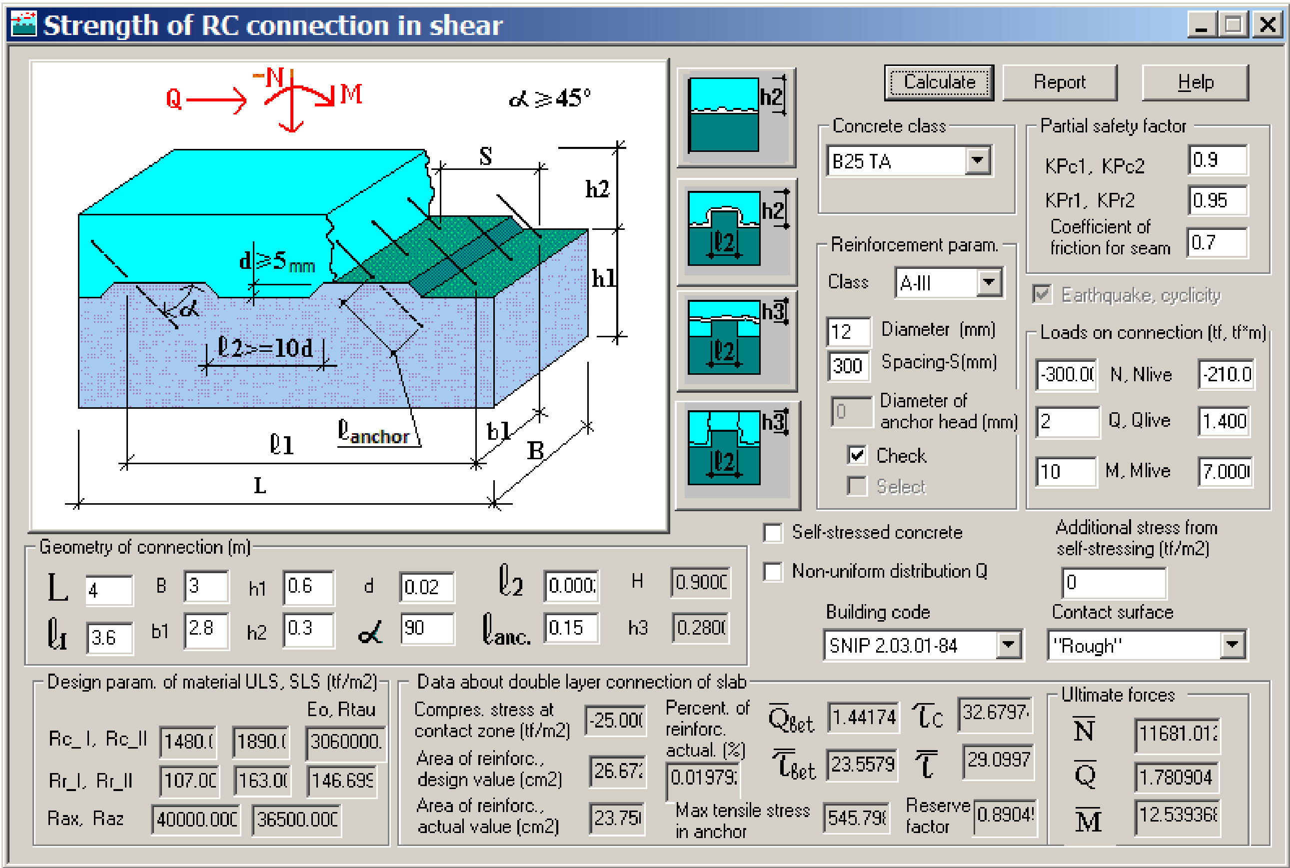


Figure 5.11

Calculation is made according to the following building codes:
1. EN 1992-1 Eurocode 2. 'Design of concrete structures. Part 1. General rules and rules for buildings'. 1992.
2. DSTU-N EN 1998-1:2011. 'Design in earthquake-prone regions. Basis of design.' (EN 1998-1:2004 Eurocode 8), 2004.
3. DSTU B V.2.6-156: 2010. 'Concrete and reinforced concrete structures from heavyweight concrete', 2010.

Output data and notation

$$B = L; \quad h = H$$

$$I_2 = b_{j1}; \quad h_{j2} = h_2$$

$$V_{Sd} = Q; \quad M_{Sd} = M; \quad N_{Sd} = N$$

$$f_{ck} = Rc\_II; \quad f_{cd} = Rc\_I$$

$$f_{yd} = Raz$$

$\bar{\tau}$ – ultimate shear stress taken by the section;

$\bar{Q}_{bet}$ – ultimate shear force taken by concrete section;

$\bar{N}$ – ultimate axial force taken by reinforced concrete section;

$\bar{Q}$ – ultimate shear force taken by reinforced concrete section;

$\bar{M}$ – ultimate moment taken by reinforced concrete section.

To present a report document in HTML format, click **Report**.

Concrete sections with fibre-reinforced polymer (FRP) bars

The module is meant for strength analysis of concrete sections with fibre-reinforced polymer (FRP) bars or with several types of reinforcement including steel one, by two theories - ultimate states and deformation model.

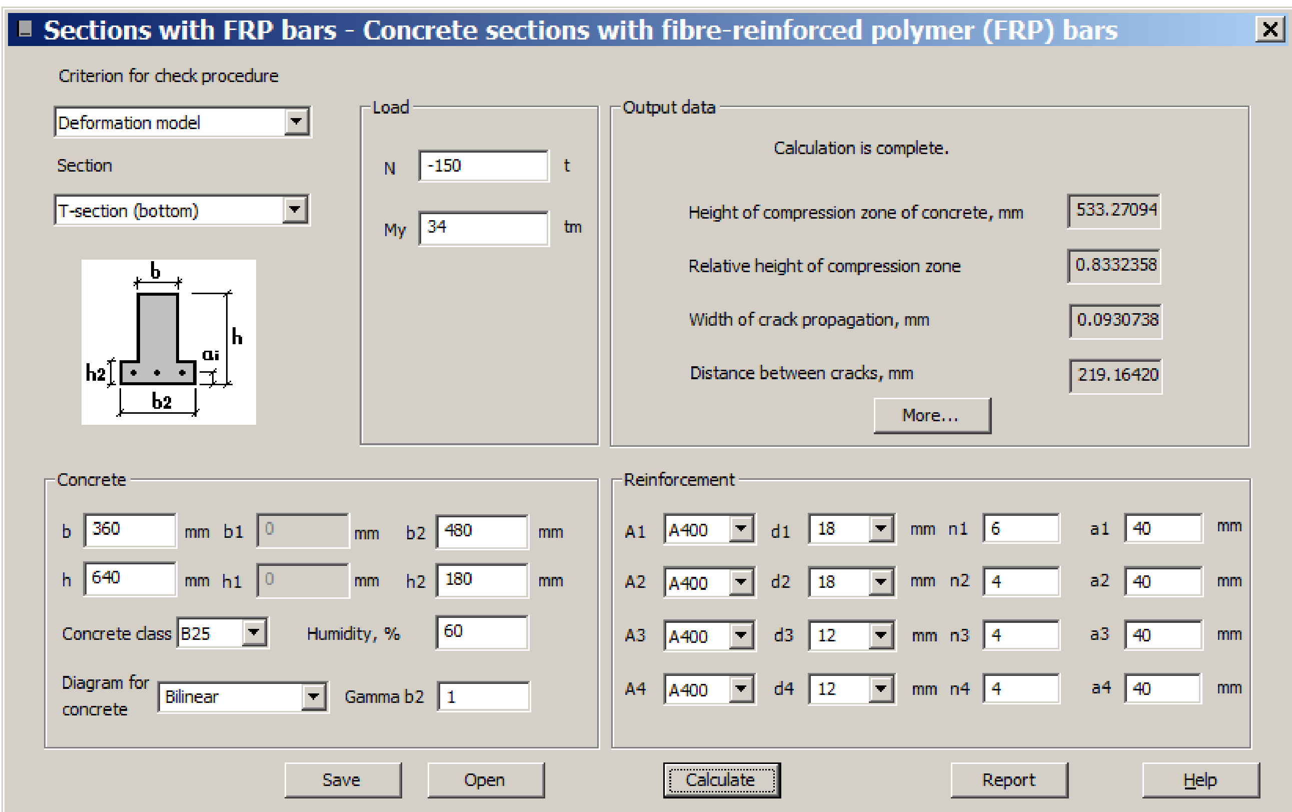


Figure 5.17

Building codes

The following building codes are supported in the module:

— SP 63.13330.2012 Concrete and reinforced concrete structures. General rules.

— SP ....2013 (first draft) Structures from concrete with fibre-reinforced polymer bars. Design requirements.

Materials

Concrete: heavy-weight concrete of classes B15, B20, B25, B30, B35, B40, B45, B50, B55, B60, B70, B80, B90, B100.
For calculation by deformation model, diagrams for concrete state are realized - bilinear, trilinear and nonlinear (Appendix G. SP 63.13330.2012).

Steel reinforcement: A240, A400, A500, A600, A800, A1000, B500, Bp500, Bp1200, Bp1300, Bp1400, Bp1500, Bp1600.

Non-steel reinforcement:
— glass-reinforced plastics (ANK-S);
— basalt-plastic (ANK-B);
— carbon fibre-reinforced plastic (ANK-U);
— aramid fibre (ANK-A);
— hybrid (ANK-H).

Section types
The following types of concrete sections are realized in the program:
— rectangular;
— T-section with table at the bottom;
— T-section with table at the top;
— I-section.

Input data

Concrete: section dimensions (**b, h, b1, h1, b2, h2**), class of concrete, relative humidity, partial safety factor for concrete Yb2, diagram for concrete state (for calculation by deformed shape).

Reinforcement: It is possible to define 4 types of reinforcement with different location relative to bottom edge of the section. For every type of reinforcement, the following data is defined: class of reinforcement (composite and steel), diameter **di**, number of rebars **ni** and distance from the bottom edge **ai**.

Load:
N – axial force (compression is denoted with sign «-»),
M – bending moment («+» if bottom edge of section is in tension, «-» if top edge of section is in tension).

To work with binary file of the program data, sue the **Save** and **Open** buttons. To start calculation, click **Calculate**.

Output data

When calculation is complete, you will obtain the following values: height of compression zone of concrete, relative height of compression zone of concrete, width of crack propagation and distance between cracks.

To view additional results, click **More**. In another dialog box you will see the following data:
– **x** – height of compression zone of concrete;
– **ksi** – relative height of compression zone of concrete;
– **% arm** – percentage of reinforcement for section;
– **l_crc** – distance between cracks;
– **a_crc** – width of crack propagation;
– **Abet** – area of concrete;
– **A1, A2, A3, A4** – areas of reinforcement;
– **Rs1, Rs2, Rs3, Rs4** – design strengths corresponding to them;
– **h_crc** – depth of crack (calculated only by deformation model).

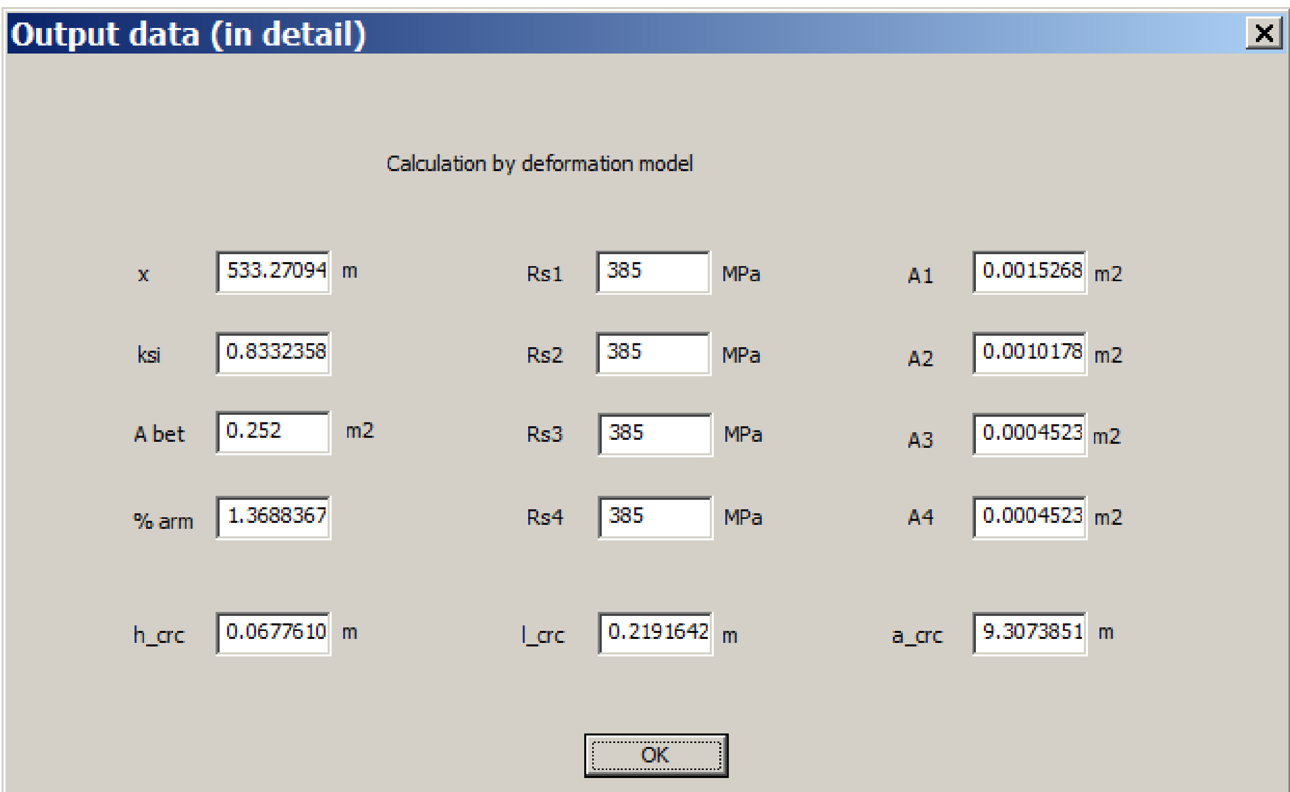


Figure 5.18

Report

To present a report document in HTML format, click **Report**.

The report contains material stress-strain properties as well as the output data:
Percentage of reinforcement for section, %
Relative height of compression zone of concrete - **ksi**
Height of compression zone of concrete, mm - **x**

Width of crack propagation, mm - **acrc**
Distance between cracks, m - **lcrc**
Equivalent area of the section, mm^2 - **Ared**
Equivalent moment of inertia for the section, mm^4 - **Ired**
Stress in extreme tensile or min compressed reinforcement, MPa - **Sigma_kr**

For calculation by ultimate states:

Equivalent section modulus for the section, mm^3 - **Wred**
Ultimate moment, tm - **Mult**
Cracking bending moment, tm - **Mcrc**

For calculation by deformation model:

Max depth of cracks, mm - **hcrc**
Relative strain in extreme compressed fibre of concrete - **EpsBMax**.

Concrete pipe sections

The module is meant for strength analysis of concrete pipe sections of columns based on deformation model.

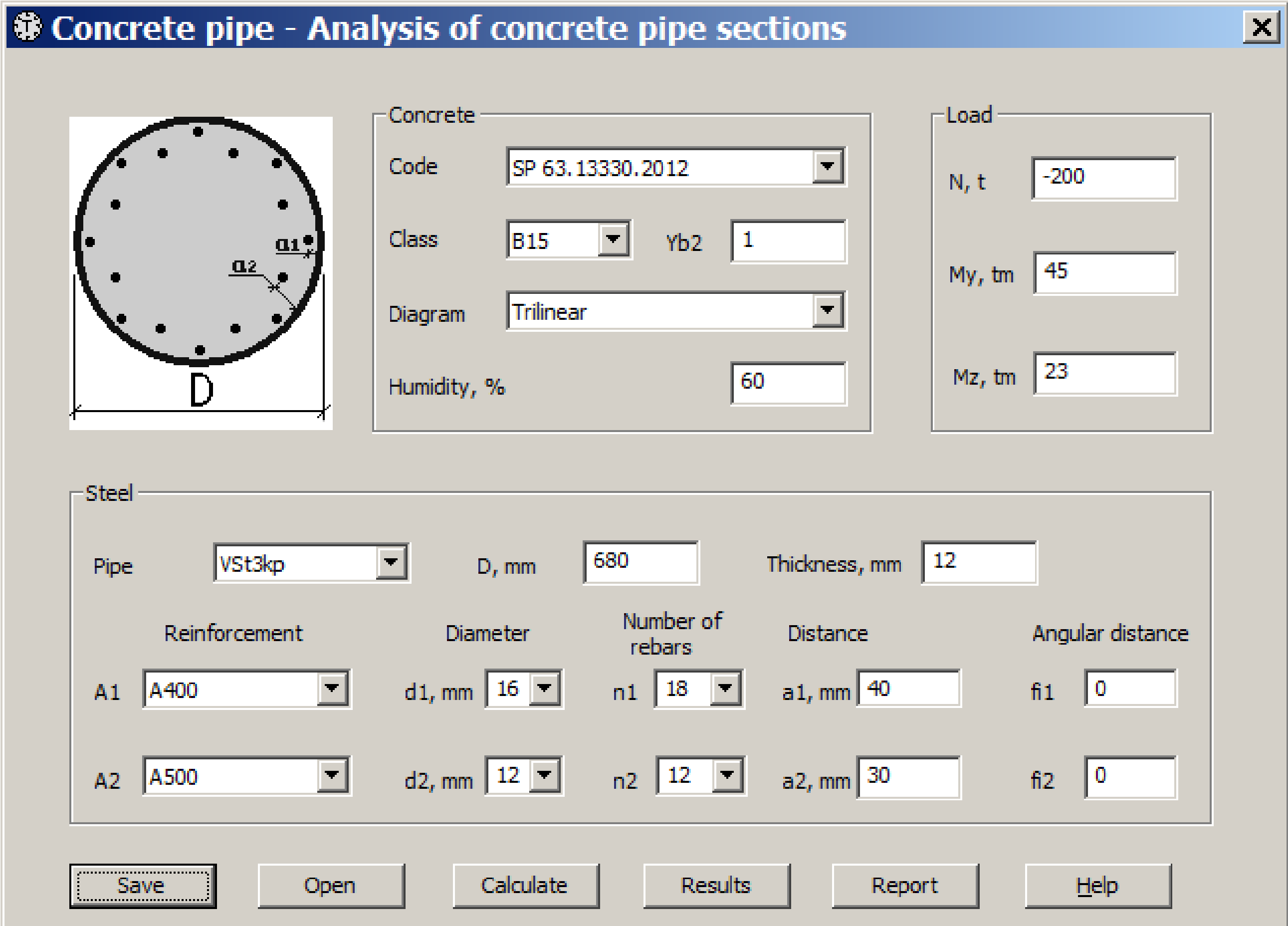


Figure 5.19

The following building codes are supported in the module:	
Russian Federation	**Ukraine**
– SP 63.13330.2012 - Concrete and reinforced concrete structures. General rules. – SP 16.13330.2011 - Steel structures. – SP ___ 2013 - Structures from concrete with fibre-reinforced polymer bars. Design requirements.	– DBN V.2.6-160:2010 - Steel-concrete composite structures. Basic provisions. – DBN V.2.6-98:2009 - Concrete and reinforced concrete structures. Basic provisions. – DBN V.2.6-135(163):2010 - Steel structures. Requirements for design, manufacturing and assembly. – DSTU-N B V.2.6-XXXX:2011 - Manual on design and manufacturing of concrete structures with non-steel composite reinforcement based on basalt and glass rovings.
Materials	

Concrete - heavyweight concrete of classes:	
B15, B20, B25, B30, B35, B40, B45, B50, B55, B60, B70, B80, B90, B100 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized (Apendix G. SP 63.13330.2012).	C12/15, C16/20, C20/25, C25/30, C30/35, C32/40, C35/45, C40/50, C45/55, C50/60 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized.
Steel reinforcement:	
A240, A400, A500, A600, A800, A1000, B500, Bp500, Bp1200, Bp1300, Bp1400, Bp1500, Bp1600	A240, A400, A500, B500
Non-steel reinforcement:	
glass-reinforced plastics (ANK-S); basalt-plastic (ANK-B); carbon fibre-reinforced plastic (ANK-U); aramid fibre (ANK-A); hybrid (ANK-H).	Basalt TU U V.2.7-25.2-34323267-001:2009 Basalt TU U V.2.7-25.2-21191464-:20 Glass TU U V.2.7-25.2-21191464-:20
Steel pipe:	
VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20	VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20 16G2AF 09G2S

Input data

Concrete: Class of concrete, relative humidity, partial safety factor for concrete Yb2, diagram of concrete state.

Steel pipe: Steel, diameter **D**, web thickness.

Reinforcement: It is possible to define 2 types of reinforcement with different location. For every type of reinforcement, the following data is defined: class of reinforcement (composite and steel), diameter **di**, number of rebars **ni**, distance from the bottom edge of pipe **ai** and angular distance relative to the Y-axis.

Load:
N – axial force (compression is denoted with sign «-»),
My – bending moment relative to Y-axis («+» if bottom edge of section is in tension, «-» if top edge of section is in tension),
Mz – bending moment relative to Z-axis («+» if right edge of section is in tension, «-» if left edge of section is in tension).

To work with binary file of the program data, sue the **Save** and **Open** buttons. To start calculation, click **Calculate**.

Output data

When calculation is complete, to obtain results, click **Results**. In another dialog box you will see the following data:
– **x** – height of compression zone of concrete;
– **Abet** – area of concrete;
– **Rb*/Rb** – coefficient for strengthening of compressed concrete,
– **a_crc** – width of crack propagation;
– **A, A1** and **A2** – areas of pipe and reinforcement;
– **% arm** – percentage of reinforcement for section.

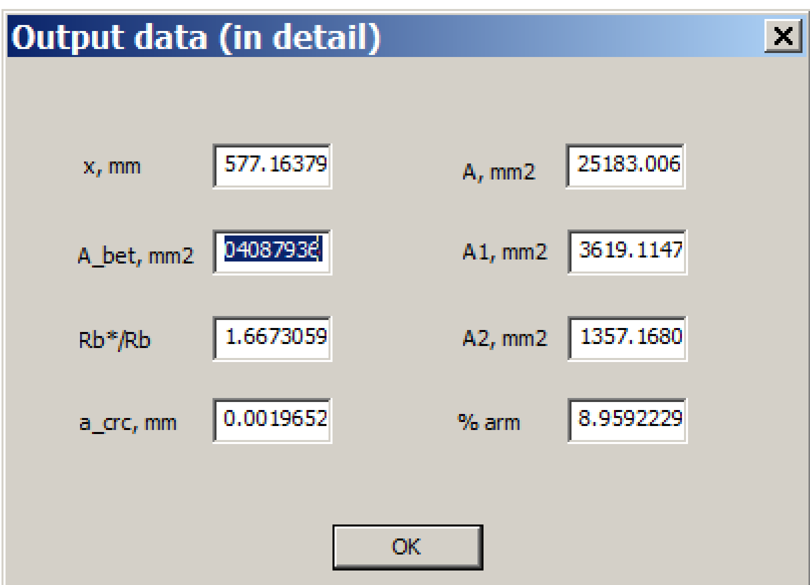


Figure 5.20

Report

To present a report document in HTML format, click **Report**.

The report contains physical and mechanical properties of materials as well as the output data:
Percentage of reinforcement for section, **%**
Relative height of compression zone of concrete – **ksi**
Height of compression zone of concrete, mm – **x**
Width of crack propagation, mm – **acrc**
Max depth of cracks, mm – **hcrc**
Distance between cracks, m - **lcrc**
Coordinates of gravity centre – **Y, Z**
Relative strain in extreme compressed fibre of concrete - **EpsBMax**
Coefficient for strengthening of compressed concrete – **Rb*/Rb**.

Composite steel and concrete columns

The module is meant for strength analysis of sections in composite steel and concrete columns based on deformation model.

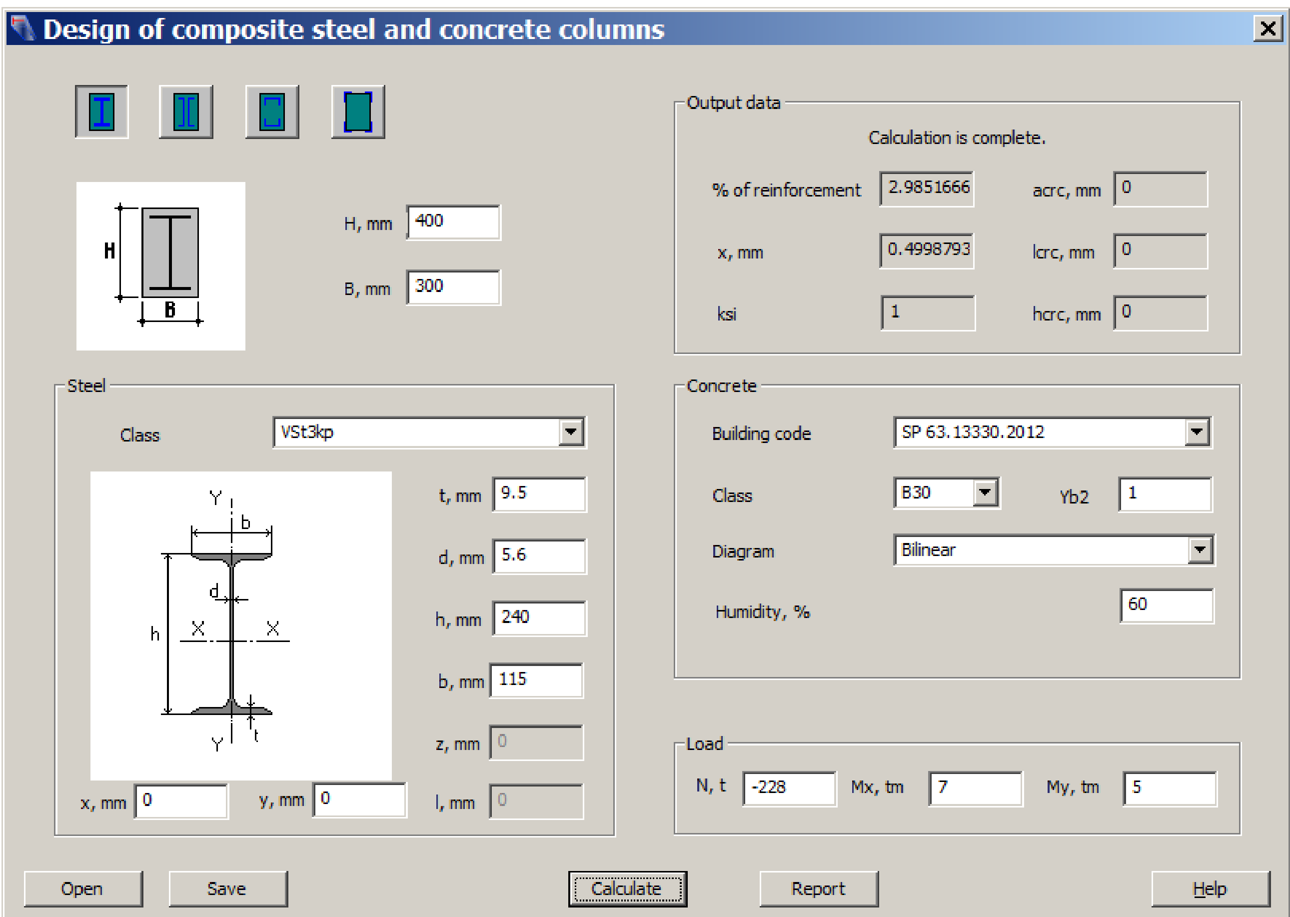


Figure 5.21

The following building codes are supported in the module:	
Russian Federation	**Ukraine**
– SP 63.13330.2012 - Concrete and reinforced concrete structures. General rules. – SP 16.13330.2011 - Steel structures.	– DBN V.2.6-160:2010 - Steel-concrete composite structures. Basic provisions. – DBN V.2.6-98:2009 - Concrete and reinforced concrete structures. Basic provisions. – DBN V.2.6-135(163):2010 - Steel structures. Requirements for design, manufacturing and assembly.
Materials	
Concrete - heavyweight concrete of classes:	
B15, B20, B25, B30, B35, B40, B45, B50, B55, B60, B70, B80, B90, B100 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized (Apendix G. SP 63.13330.2012).	C12/15, C16/20, C20/25, C25/30, C30/35, C32/40, C35/45, C40/50, C45/55, C50/60 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized.
Steel reinforcement:	
A240, A400, A500, A600, A800, A1000, B500, Bp500, Bp1200, Bp1300, Bp1400, Bp1500, Bp1600	A240, A400, A500, B500
Steel pipe:	
VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20	VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20 16G2AF 09G2S

Input data

Concrete: Class of concrete, relative humidity, partial safety factor for concrete Yb2, diagram of concrete state.

Steel: Steel and dimensions of steel shape **h, b, d, t, z**, distance between gravity centres (for channel sections) **l**.

Load:
N – axial force (compression is denoted with sign «-»),
My – bending moment relative to Y-axis («+» if bottom edge of section is in tension, «-» if top edge of section is in tension),
Mz – bending moment relative to Z-axis («+» if right edge of section is in tension, «-» if left edge of section is in tension).

To work with binary file of the program data, sue the **Save** and **Open** buttons. To start calculation, click **Calculate**.

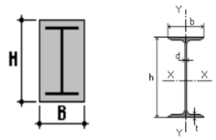
Output data

When calculation is complete, in appropriate section of the dialog box you will see results:
x – height of compression zone of concrete,
% arm – percentage of reinforcement for section,
ksi – relative height of compression zone of concrete,
a_crc – width of crack propagation,
h_crc – depth of crack,
l_crc – distance between cracks.

Report

To present a report document in HTML format, click **Report**.

Input data



Building codes
SP 63.13330.2012
SP 16.13330.2011

Concrete					
Class of concrete	Eb, GPa	Rb, MPa	Rbt, MPa	Epsb2	Epsbt2
B30	34.5	17	1.15	0.0048	0.00031
Partial safety factor for concrete	1	Humidity, %	60	Strain diagram	Bilinear
Section					
H, mm		B, mm		Area, mm^2	
400		300		120000	
Steel					
Class	Es, GPa	Ry, MPa	Ru, MPa	Ryn	Run
VSt3kp	206	215	350	225	370
hs, mm	bs, mm	ds, mm	ts, mm	---	---
240	115	5.6	9.5	---	---

Design forces

N, t	-228	My, tm	7	Mz, tm	5

Output data

Percentage of reinforcement in section, %	2.98517
Relative height of compression zone of concrete	ksi=1
Height of compression zone of concrete, mm	x=0.499879
Width of crack propagation, mm	acrc=0
Max depth of cracks, mm	hcrc=0
Distance between cracks, m	lcrc=0
Coordinates of gravity centre of the section	Y=0.00276343 m Z=-0.0145168 m
Equivalent stiffnesses for the section	EAred=100237 t EIx=1250.61 tm2 EIy=522.601 tm2

Figure 5.22

The report contains physical and mechanical properties of materials as well as the output data:
Percentage of reinforcement for section, **%**
Relative height of compression zone of concrete – **ksi**
Height of compression zone of concrete, mm – **x**
Width of crack propagation, mm – **acrc**
Max depth of cracks, mm – **hcrc**
Distance between cracks, m - **lcrc**
Coordinates of gravity centre – **Y, Z**
Equivalent stiffnesses for the section – **EAred, EIy, EIz**.

Column strengthened with composed materials

This module enables you to check strength of reinforced concrete sections in case the columns are strengthened with composite materials.

The following regulations are supported in the program:
SP 164.1325800.2014 'Strengthening of reinforced concrete structures with composite materials';
SP 63.13330.2012 'Concrete and reinforced concrete structures. General rules.'

 www.liraland.com

Input data

In the dialog box define section dimensions, properties of concrete, reinforcement and composite material as well as design force.

The following classes of concrete are allowed: B10, B15, B20, B25, B30, B35, B40, B45, B50, B55, B60.

The following classes of reinforcement are allowed: A240, A400, A500, B600, B800.

Max diameter of reinforcement bars - 40 mm.

Location of reinforcement (distance, a) - distance from gravity centre of reinforcement up to the nearest edge (bottom reinforcement to bottom edge, top reinforcement to top edge).

Physical and mechanical properties of suggested FRP reinforcement types (composite materials) are taken from Appendix 5 'Manual on strengthening of RC structures with composite materials'. It is strongly recommended that you should revise properties of selected materials from the manufacturer. If it is necessary to define properties of composite material that is not available in the **Manufacturers** and **Reinforcement type FRP** lists, then these properties may be defined manually in appropriate boxes - thickness of monolayer, its modulus of elasticity and tensile strength.

The dialog box of the program is presented in the figure below.

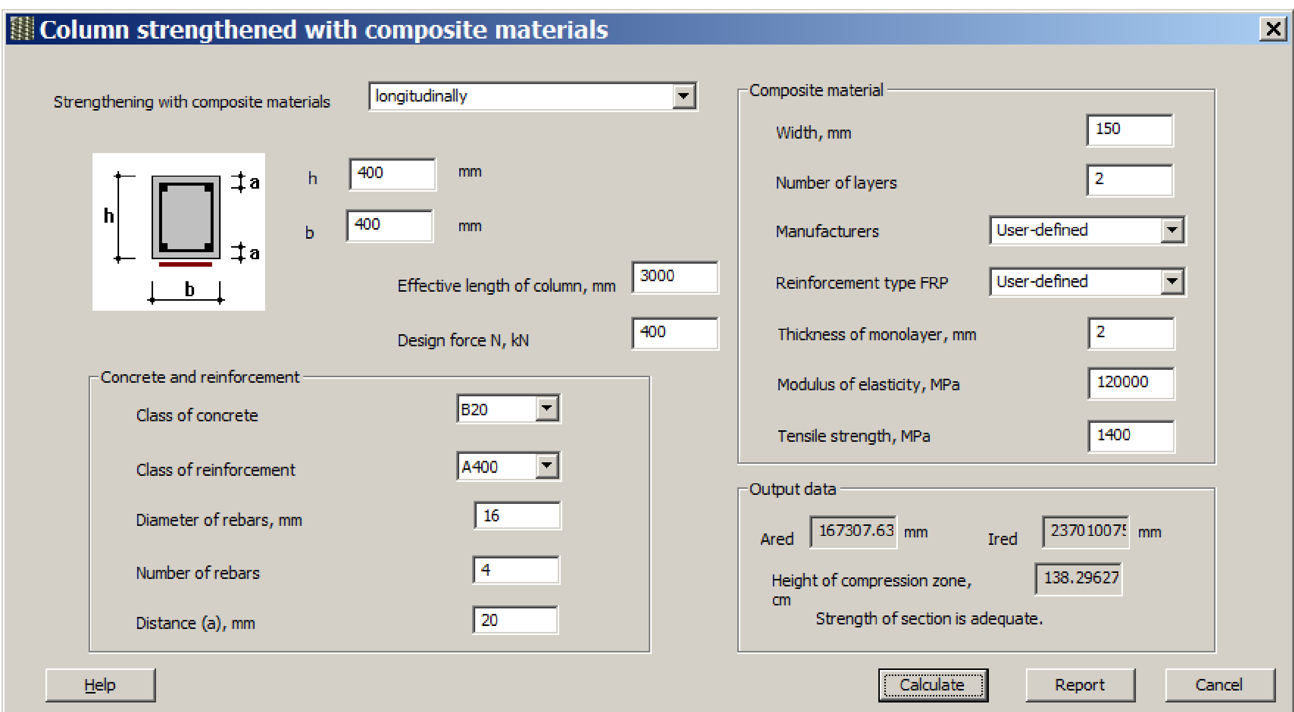


Figure 5.23

In the dialog box define the type of column strengthening with composite material - longitudinally or transversely.

You should also define effective length of column, section parameters and axial force N applied to the gravity centre.

To define composite material that is not available in the **Manufacturers** and **Reinforcement type FRP** lists, simply define properties in the appropriate boxes: thickness of monolayer, its modulus of elasticity and tensile strength.

To start calculation, click appropriate button.

Output data

To present a report document in HTML format, click **Report**.

Notation in output data

Ared - area of equivalent section;
Ired - moment of inertia of equivalent section;
x - height of compression zone of concrete.

Column strengthened with composite materials

Input data

Strengthening with composite materials:longitudinally	
Section height, mm	400
Section width, mm	400
Design force N, kN	400
Effective length of column, mm	3000
Concrete and reinforcement	
Class of concrete	B20
Class of reinforcement	A400
Diameter of reinforcement, mm	16
Number of rebars	4
Distance, mm	20
Composite material	
Width, mm	150
Number of layers	2
Manufacturer	User-defined
Reinforcement type FRP	User-defined
Thickness of monolayer, mm	2
Modulus of elasticity, MPa	120000
Tensile strength, MPa	1400

Output data

Ared, mm2	167308
Ired, mm3	2.3701e+009
Height of compression zone, cm	138.296

Figure 5.24

Composite steel and concrete slabs

The module is meant for strength analysis of sections in composite steel and concrete ribbed slabs based on deformation model.

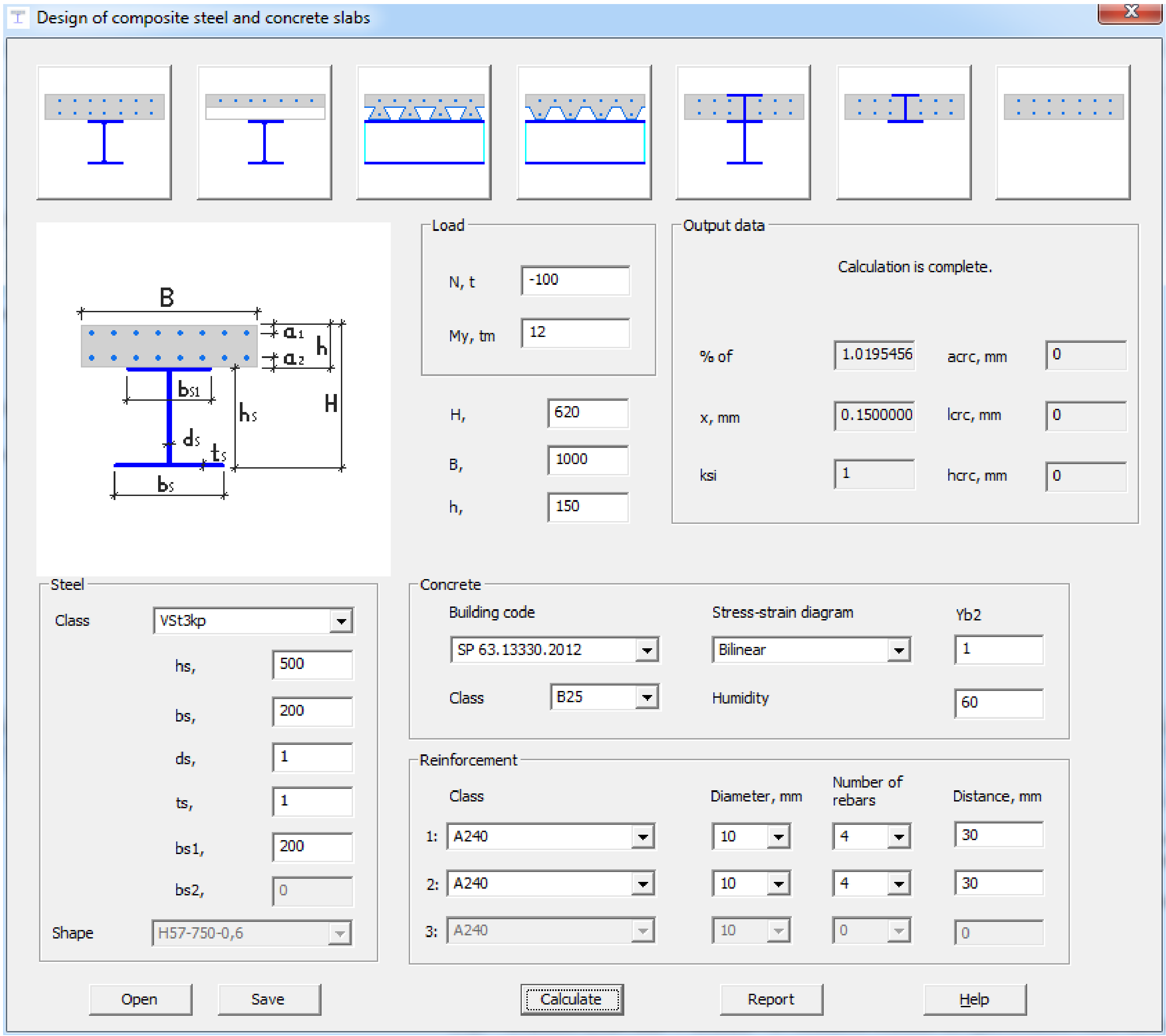


Figure 5.25

The following building codes are supported in the module:	
Russian Federation	**Ukraine**
– SP 63.13330.2012 - Concrete and reinforced concrete structures. General rules. Updated edition of SNIP 52-01-2003 (with amendment No.1) – SP 16.13330.2011 - Steel structures.	– DBN V.2.6-160:2010 - Steel-concrete composite structures. Basic provisions. – DBN V.2.6-98:2009 - Concrete and reinforced concrete structures. Basic provisions.

– GOST 24045-94 - Steel roll-formed trapezoidally corrugated sheets for construction.	– DBN V.2.6-135(163):2010 - Steel structures. Requirements for design, manufacturing and assembly. – DSTU-N B V.2.6-XXXX:2011 - Manual on design and manufacturing of concrete products and structures with non-metallic composite reinforcement based on basalt and glass rovings – DSTU B V.6-9:2008 Steel roll-formed trapezoidally corrugated sheets for construction. Technical requirements.
Materials	
Concrete - heavyweight concrete of classes:	
B15, B20, B25, B30, B35, B40, B45, B50, B55, B60, B70, B80, B90, B100 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized (Appendix G. SP 63.13330.2012).	C12/15, C16/20, C20/25, C25/30, C30/35, C32/40, C35/45, C40/50, C45/55, C50/60 For calculation by deformation model, the bilinear and trilinear diagrams of concrete state are realized.
Reinforcement (steel):	
A240, A400, A500, A600, A800, A1000, B500, Bp500, Bp1200, Bp1300, Bp1400, Bp1500, Bp1600	A240, A400, A500, B500
Reinforcement (other):	
fiberglass (ANK-S); basalt-plastic (ANK-B); carbon-fibre (ANK-U); aramid fibre (ANK-A); hybrid (ANK-H).	Basalt TU U V.2.7-25.2-34323267-001:2009 Basalt TU U V.2.7-25.2-21191464-:20 Glass TU U V.2.7-25.2-21191464-:20
Steel pipe:	
VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20	VSt3kp VSt3ps VSt3sp VSt3ps4 VSt3sp4 20 16G2AF 09G2S

Corrugated sheet:	
H57-750-0,6 H57-750-0,7 H57-750-0,8 H60-845-0,7 H60-845-0,8 H60-845-0,9 H75-750-0,7 H75-750-0,8 H75-750-0,9 H114-600-0,8 H114-600-0,9 H114-600-1,0 H114-750-0,8 H114-750-0,9 H114-750-1,0 HC35-1000-0,6 HC35-1000-0,7 HC35-1000-0,8 HC44-1000-0,7 HC44-1000-0,8 C10-899-0,6 C10-899-0,7 C10-1000-0,6 C10-1000-0,7 C18-1000-0,6 C18-1000-0,7	H57-750-0,6 H57-750-0,7 H57-750-0,8 H60-845-0,7 H60-845-0,8 H60-845-0,9 H75-750-0,7 H75-750-0,8 H75-750-0,9 H114-600-0,8 H114-600-0,9 H114-600-1,0 H114-750-0,8 H114-750-0,9 H114-750-1,0 HC35-1000-0,6 HC35-1000-0,7 HC35-1000-0,8 HC44-1000-0,7 HC44-1000-0,8 C10-899-0,6 C10-899-0,7 C10-1000-0,6 C10-1000-0,7 C18-1000-0,6 C18-1000-0,7 C15-800-0,6 C15-800-0,7 C15-800-0,6 C15-800-0,7 C21-1000-0,6 C21-1000-0,7 C44-1000-0,7

Input data

Concrete: Class of concrete, relative humidity, partial safety factor for concrete Yb2, diagram of concrete state.

Steel: Steel and dimensions of steel shape **h, b, d, t, z**.

Reinforcement: It is possible to define 2 (in some sections 3) types of reinforcement with different location. For every type of reinforcement, the following data is defined: class of reinforcement (steel and composite non-metallic), diameter of rebars **di**, number of rebars **ni**, distance from the bottom edge of pipe **ai**.

Shape: Range of corrugated sheets.

Load:
N – axial force (compression is denoted with sign «-»),
My – bending moment relative to Y-axis («+» if bottom edge of section is in tension, «-» if top edge of section is in tension).

To work with the file of program data, use the **Save** and **Open** buttons.
To start calculation, click **Calculate**.

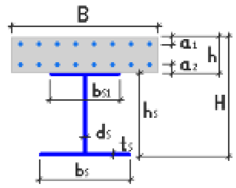
Output data

The report contains physical and mechanical properties of materials as well as the output data:
Percentage of reinforcement for section, **%**
Relative height of compression zone of concrete – **ksi**
Height of compression zone of concrete, mm – **x**
Width of crack propagation, mm – **acrc**
Max depth of cracks, mm – **hcrc**
Distance between cracks, m - **lcrc**
Coordinates of gravity centre – **Y, Z**
Equivalent stiffnesses for the section – **EAred, EIy, EIz**.

To present a report document in HTML format, click **Report**.

Design of composite steel and concrete slabs

Input data



Building codes
SP 63.13330.2012
SP 16.13330.2011
SP ___ 2013

Concrete					
Concrete class	Eb, GPa	Rb, Mpa	Rbt, MPa	Epsb2	Epsbt2
B25	30	14.5	1.05	0.0048	0.00031
Partial safety factor for concrete	1	Humidity, %	60	Stress-strain diagram	Bilinear
Section					
H, mm		B, mm		Area, mm^2	
150		1000		150000	
Steel					
Class	Es, GPa	Ry, MPa	Ru, MPa	Ryn	Run
VSt3kp	206	215	350	225	370
hs, mm	bs, mm	ds, mm	ts, mm	H	bs1, mm / bs2, mm
500	200	1	1	620	200 / 0
Reinforcement					
Layer No.	Number of rebars	Diameter, mm	Area, mm^2	Distance, mm	
1	4	10	0.000314159	30	
Class of reinforcement	Es, GPa	Rs, MPa	Rsc, MPa	Epss	Epsc
A240	0	0	0	0	
Layer No.	Number of rebars	Diameter, mm	Area, mm^2	Distance, mm	
2	4	10	0.000314159	30	1.16254e-307
Class of reinforcement	Es, GPa	Rs, MPa	Rsc, MPa	Epss	Epsc
A240	0	0	0	0	0

Design forces

N, t	-100	My, tm	12		

Output data

Percentage of reinforcement for section, %	1.01955
Relative height of compression zone of concrete	ksi=1
Height of compression zone of concrete, mm	x=0.15
Width of crack propagation, mm	acrc=0
Max depth of cracks, mm	hcrc=0
Distance between cracks, m	lcrc=0
Coordinates of gravity centre of the section	Y=0 m Z=-0.0180962 m
Equivalent stiffnesses for the section	EAred=108453 t EIx=2.58365e+007 t*m2 EIy=6499.96 t*m2

Masonry and masonry reinforcing

Masonry and masonry reinforcing

The chapter contains modules for analysis of elements in masonry and masonry reinforcing structures as well as reference programs:
[Design compression strength in masonry](#)
[Brick pier](#)
[Masonry in local compression](#)
[Brick pier in tension](#)
[Brick pier by DBN V.2.6-162:2010](#)

Рис. 6.1

Design compression strength in masonry

The program presents values for design compression strength in all types of brickwork and stonework from structural clay tile with slit-like vertical cells of width up to 12mm and height of masonry course 50-150mm on heavy mortars depending on mortar mark. (Table 2 SNIP II-22-81 'Rock / reinforced masonry structures').

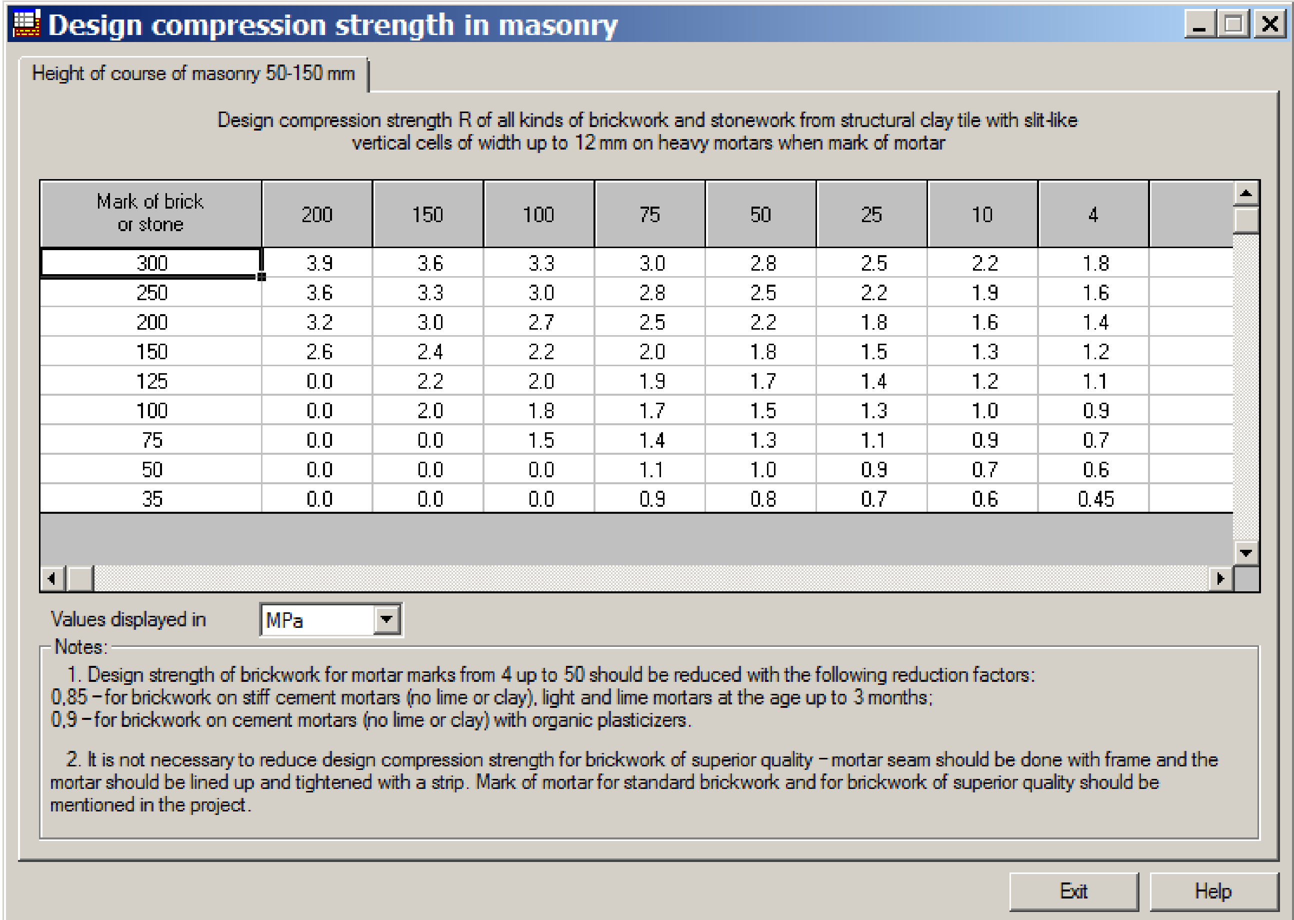


Figure 6.2

To present design strength values either in MPa or in kg/cm^2 , define appropriate data in the **Values dislpayed in** drop-down list.

Brick pier

The module enables you to analyse masonry and masonry reinforcing structures in compression according to SNIP II–22–81* 'Masonry and masonry reinforcing structures'.

Define brick pier of rectangular or T-section shape. Plane eccentric compression of brick pier is considered. Tensile and shear forces are considered in analysis. After analysis, if the pier should be reinforced, you will obtain number of courses in brickwork for which it is necessary to place wire mesh reinforcement with the specified cell and diameter of reinforcement and (or) area of longitudinal reinforcement.

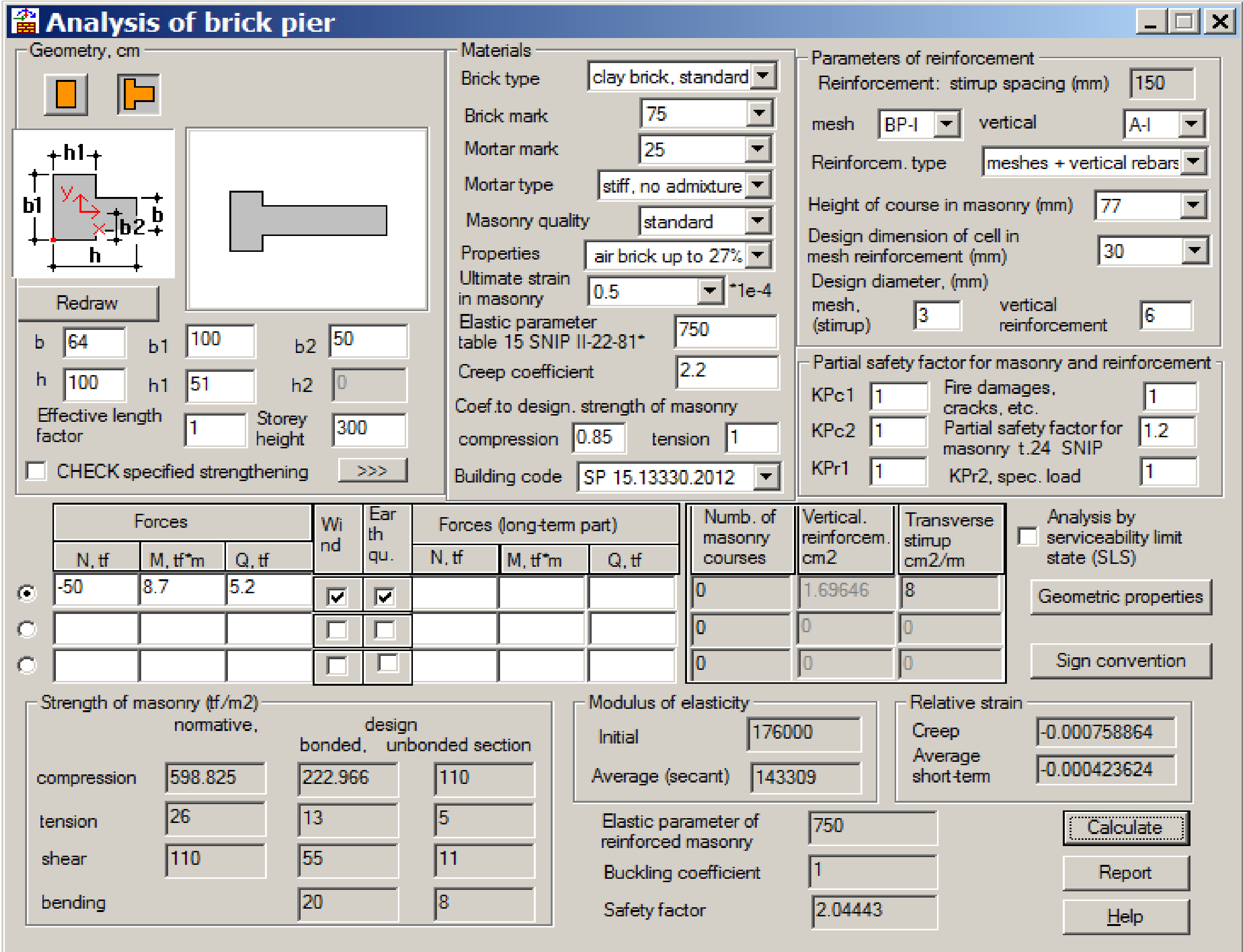


Figure 6.3

Input data

Click schematic presentation of the section that should be calculated and define its parameters.

In the appropriate boxes, type or select all data necessary for calculation by SNIP II–22–81* 'Masonry and masonry reinforcing structures'.

In the **Forces** table, define necessary values.

Click **Calculate**.

In this program you could also check the specified strengthening. It is allowed to strengthen the pier with steel or RC casing as well as with rebars in reinforced plaster.

To define parameters of strengthening, click >>> button. The **Types of strengthening of brick piers** dialog box appears on the screen. In this dialog box, select type of strengthening and define appropriate parameters. Click **OK** and close the dialog box. Then in the **Analysis of brick pier** dialog box, select the **CHECK specified strengthening** check box. Defined parameters will be displayed in this dialog box. When you define loads, it is possible to start calculation.

 www.liraland.com

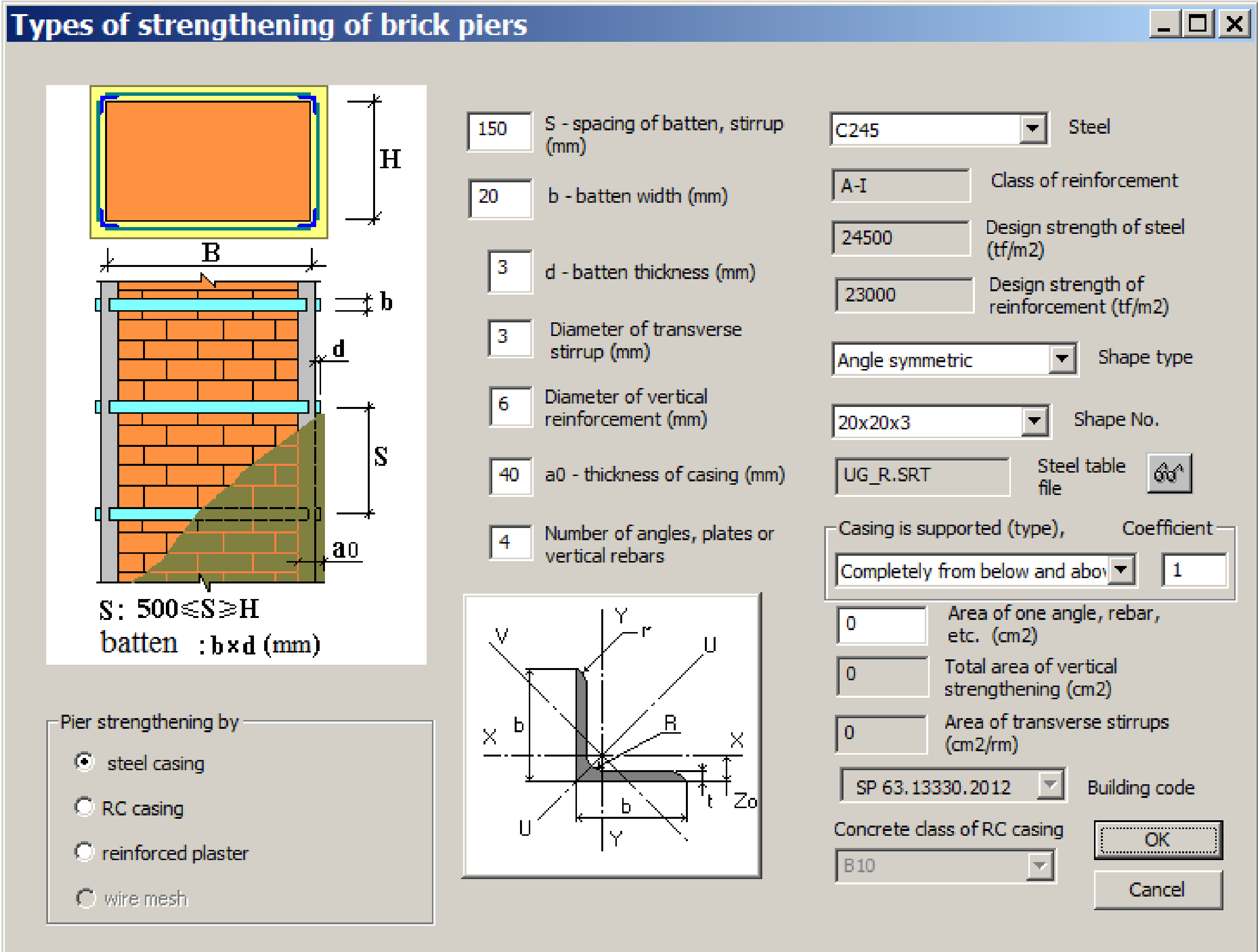
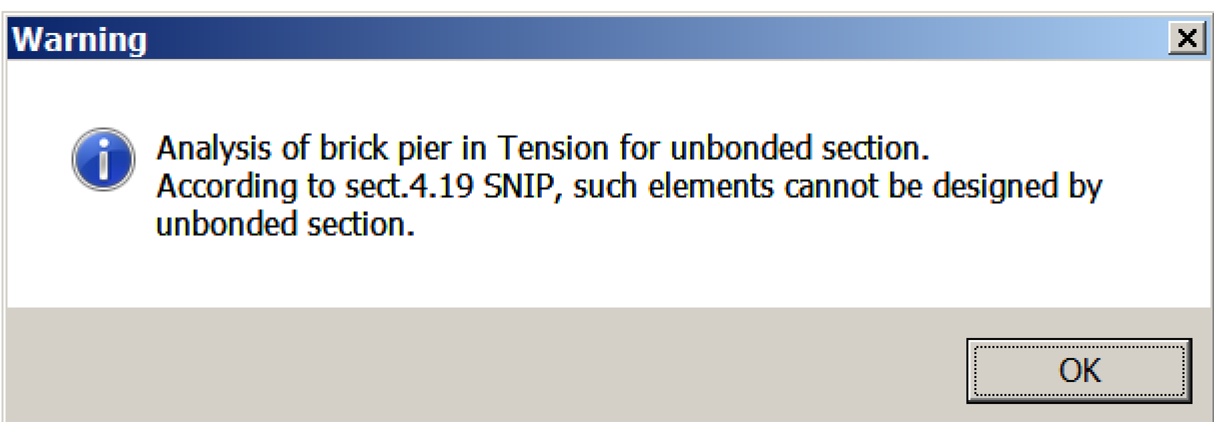


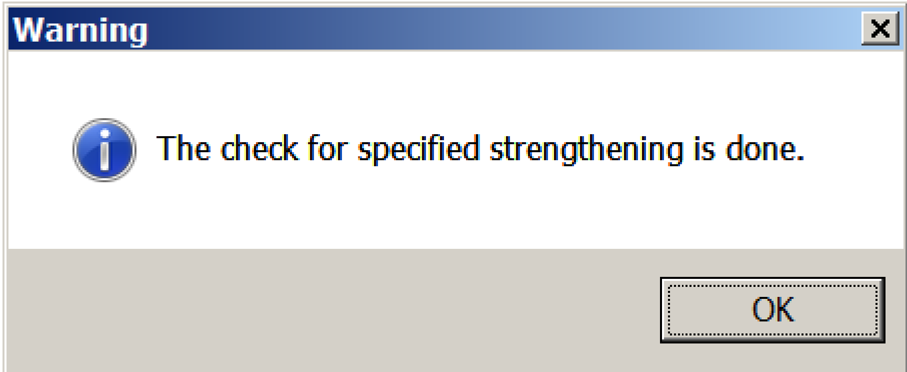
Figure 6.4

Output data

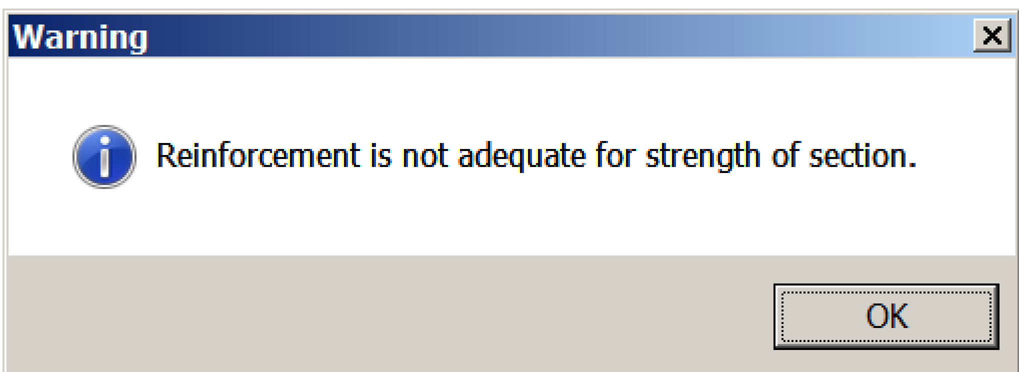
When you click **Calculate** button, you may see the following messages.



Warning according to notes to sect. 4.18 and 4.19 SNIP II–22–81* 'Masonry and masonry reinforcing structures' about provision that flexural and tensile elements of masonry structures cannot be designed by unbonded section.



Reinforcement in brick pier is selected or brick pier is adequate according to ultimate and serviceability limit states without reinforcement. If reinforcement with meshes is defined, then in the **Number of masonry courses** box, the figure from 1 to 5 will be displayed, That is, number of masonry courses through which it is necessary to place reinforcing mesh with the specified cell, diameter and class of reinforcement. When reinforcement with vertical bars is defined, then in the **Vertical reinforcement** box you will see the area of selected vertical rebars in cm.



It is impossible to select reinforcement or non-reinforced brick pier is adequate to strength and stability conditions. It is necessary to increase properties of masonry, section, properties of reinforcement or change the type of reinforcement.

In the **Strength of masonry** area, you will see normative and design values of strength for non-reinforced and reinforced masonry (depends on whether reinforcement was selected or not) along bonded and unbonded sections by different types of actions.

When you click **Report**, you will see protocol of input data and output data for masonry and masonry reinforcing piers.

Masonry in local compression

The module enables you to analyse masonry and masonry reinforcing structures in local compression according to SNIP II–22–81* 'Masonry and masonry reinforcing structures'.

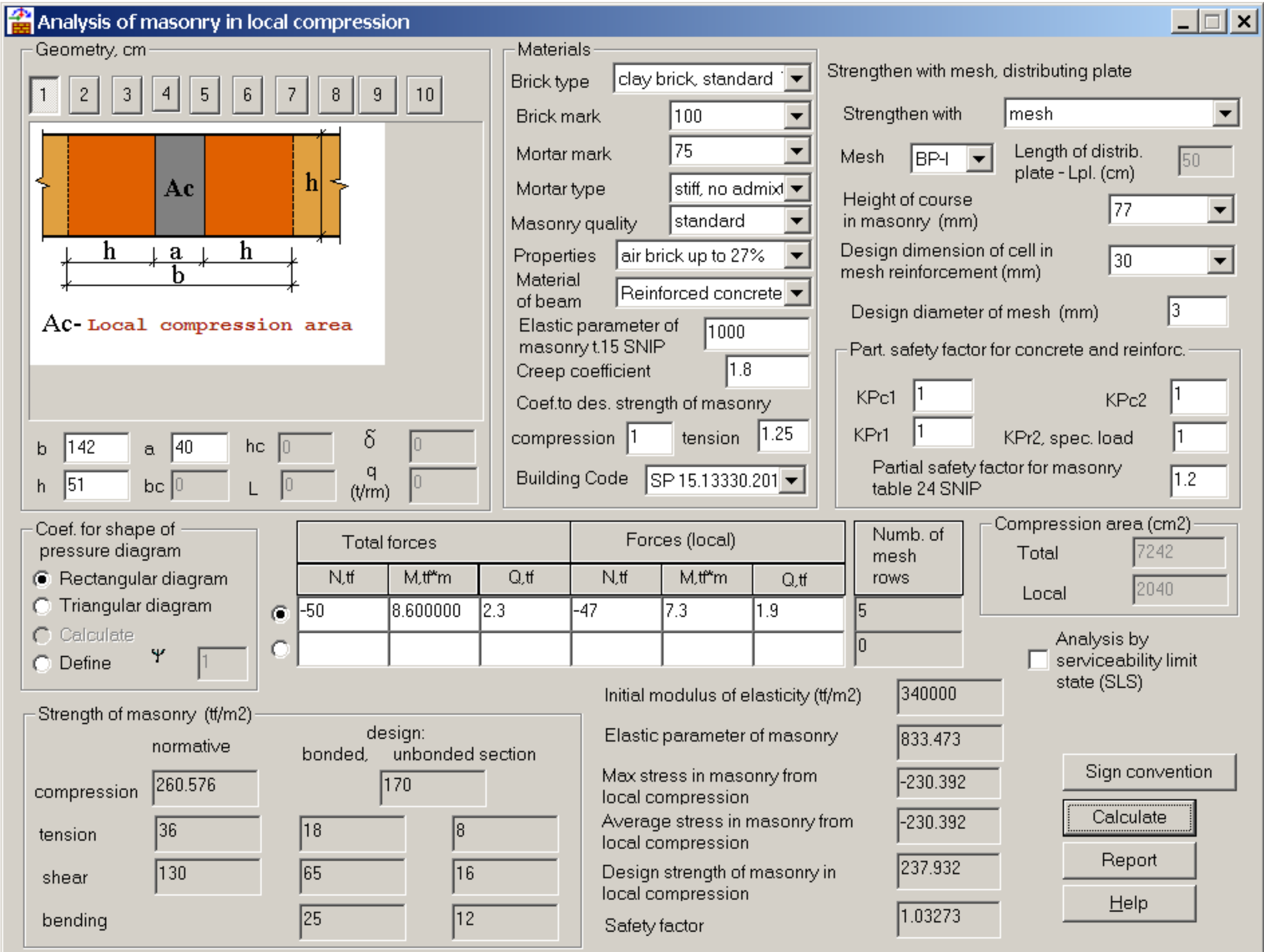
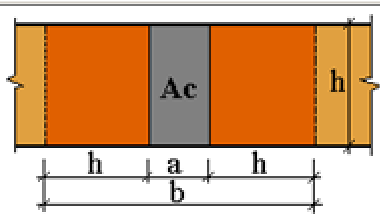
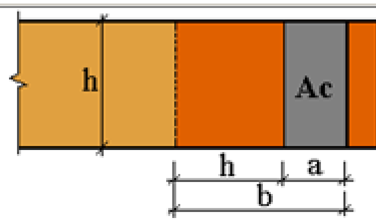
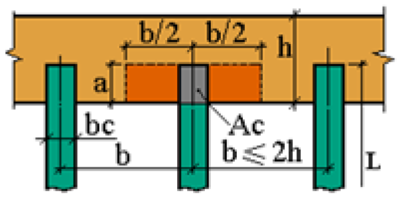
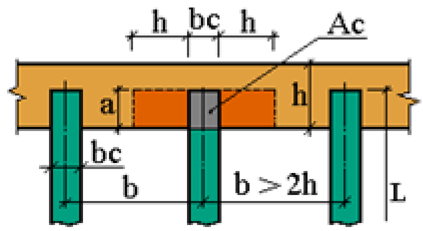
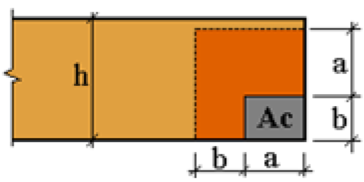
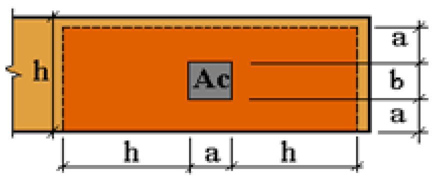
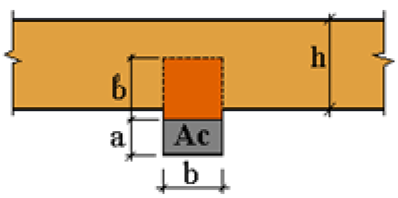
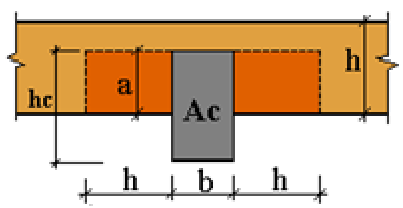
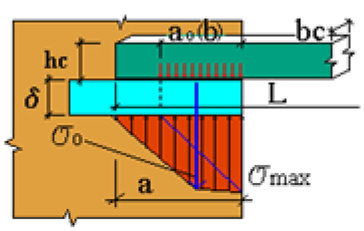
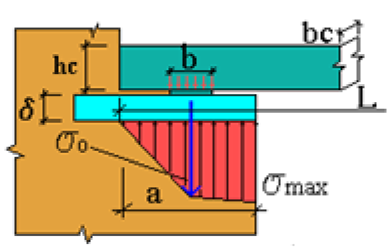


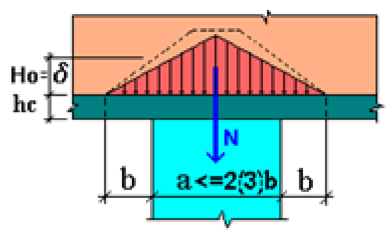
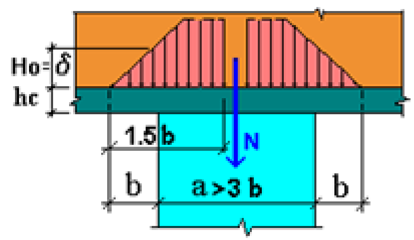
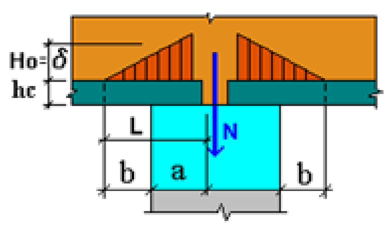
Figure 6.5

Analysis in local compression is available for eleven types of support sections (see table below).

Types of sections for analysis in local compression:

Section No.	Type of section	Type of section	Notes
1, 2			

	a – width of support zone Ac – area of local compression	a – width of support zone Ac – area of local compression	
3	 a – depth of beam setting b_c – width of beam	 a – depth of beam setting b_c – width of beam	Pinned connection between beams and walls. Variant of section is selected automatically based on dimensions **b, h**.
4, 5	 a, b – dimensions of support zone Ac – area of local compression	 a – width of support zone Ac – area of local compression	
6, 7	 a – length for area of local compression b – width of rib (pilaster) Ac – area of local compression	 a – depth of setting b – width of rib (pilaster) Ac – area of local compression	
8, 9	 a – length of beam supporting b_c , h_c – width and height of beam	 a – length of beam supporting b_c , h_c – width and height of beam	$b = a_0$ – usable length of support in pinned connection If b>0, b≤a – supporting on distributing plate a≤14cm. Lpl. – width of distributing plate

10	 a – length of beam supporting b_c , h_c – width and height of beam Variants: a) , b)	 a – length of beam supporting b_c , h_c – width and height of beam Variant: c)	h – thickness of hanging wall $\delta=H_0$ – equivalent height of spandrel beam $b = s=1.57H_0$ (δ, b are computed) Variants: a), b), c) – distribution of pressure is selected automatically (calculation should be made when coefficient for shape of pressure diagram ψ =1.0)
11	 a – length of beam supporting b_c , h_c – width and height of beam Variant: d)		For the check of masonry (d) above the utmost beams (lintels) and spandrel beams, define: a – length of support zone in spandrel beam; ψ=0.5 – coefficient for shape of pressure diagram.

Input data

With the mouse button, select schematic presentation for the section and define its parameters.

In the appropriate lists, either type or select the data necessary for analysis from SNIP II–22–81 'Masonry and masonry reinforcing structures'.

Define **Forces** values in the table.

Click **Calculate**.

Output data

In the **Strength of masonry** area of the dialog box, you will see normative and design values of strength for unreinforced and reinforced masonry (it depends on whether reinforcement was selected or not) along bonded and unbonded sections by different types of loads.

To present the report file in HTML-format, click **Report**. The report file contains the input data with the type of section and the output data as well.

Brick pier in tension

The module enables you to analyse masonry and masonry reinforcing structures in tension according to SNIP II–22–81* 'Masonry and masonry reinforcing structures'. In this program you could check strength, check and select required reinforcement in tension along bonded and unbonded sections, in tension with flexure and in local tension (breaking off an anchor) along bonded section.

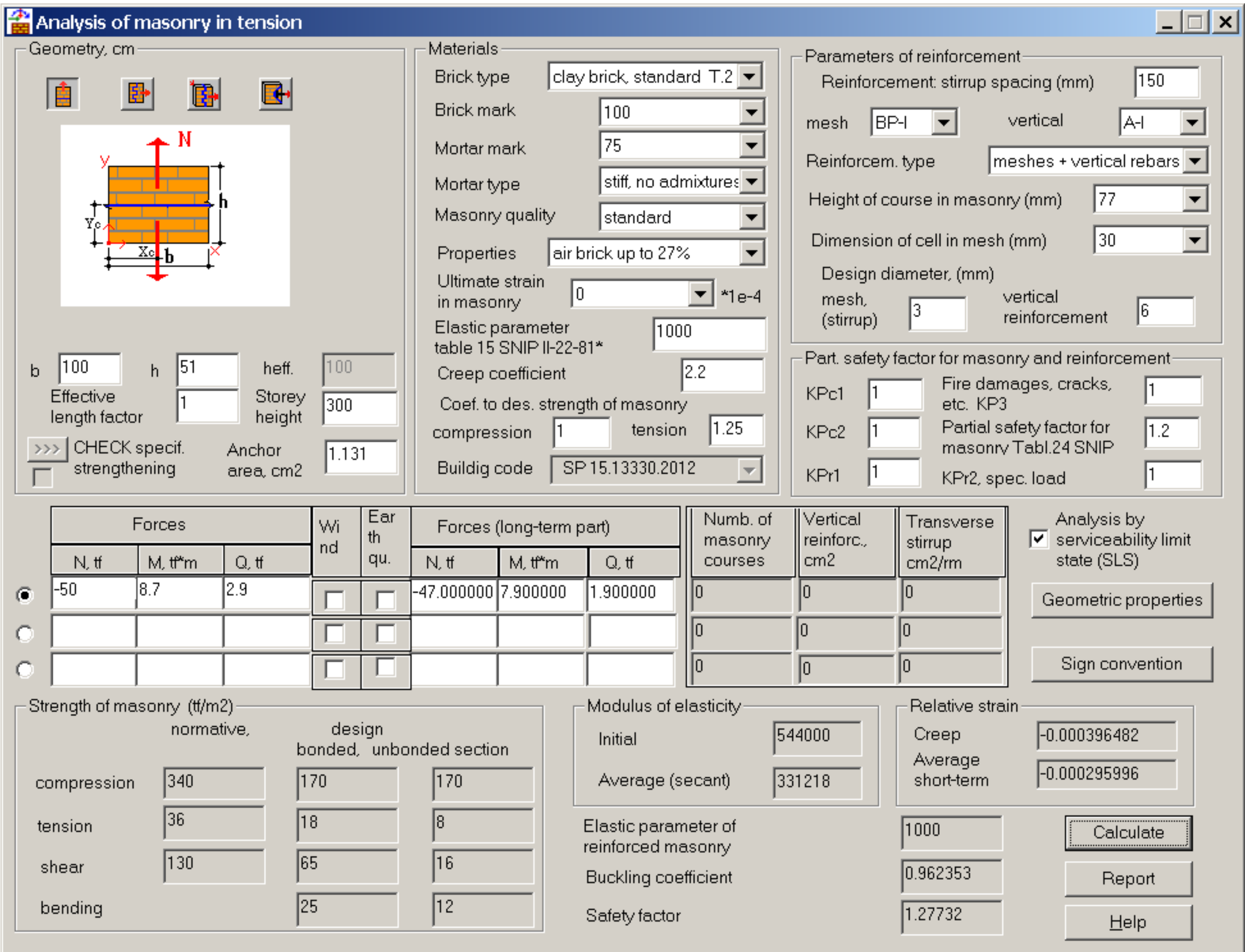
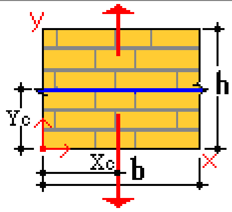
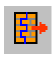
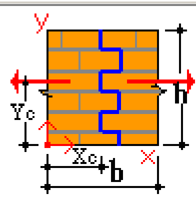
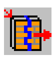
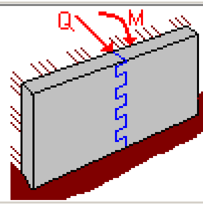
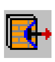
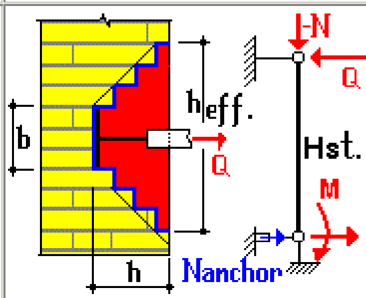


Figure 6.6

Analysis in tension is available for the following types of sections:

 www.liraland.com

No. Option button	Type of section	Notes
1 		Tension for unbonded section
2 		Tension for unbonded section
3 		Tension and flexure for bonded section
4 		Local tension (breaking off an anchor) for bonded section

Input data
With the mouse button, select schematic presentation for the section and define its parameters.
In the appropriate lists, either type or select the data necessary for analysis from SNIP II–22–81 'Masonry and masonry reinforcing structures'.
Define **Forces** values in the table.
Click **Calculate**.

Output data
In the **Strength of masonry** area of the dialog box, you will see normative and design values of strength for unreinforced and reinforced masonry (it depends on whether reinforcement was selected or not) along bonded and unbonded sections by different types of loads.

To present the report file in HTML-format, click **Report**. The report file contains the input data with the type of section and the output data as well.

Brick pier by DBN V.2.6-162:2010

Analysis of brick pier according to DBN B.2.6-162:2010 (EN 1996-1-1:2005 Eurocode 6).

The module enables you to analyse masonry piers of rectangular section, T-section, I-section and cruciform section according to DBN B.2.6-162:2010. Strength is checked and necessary reinforcement is selected in calculation. The specified strengthening of brick piers may be also checked.

The program window is presented in the Fig. 6.7.

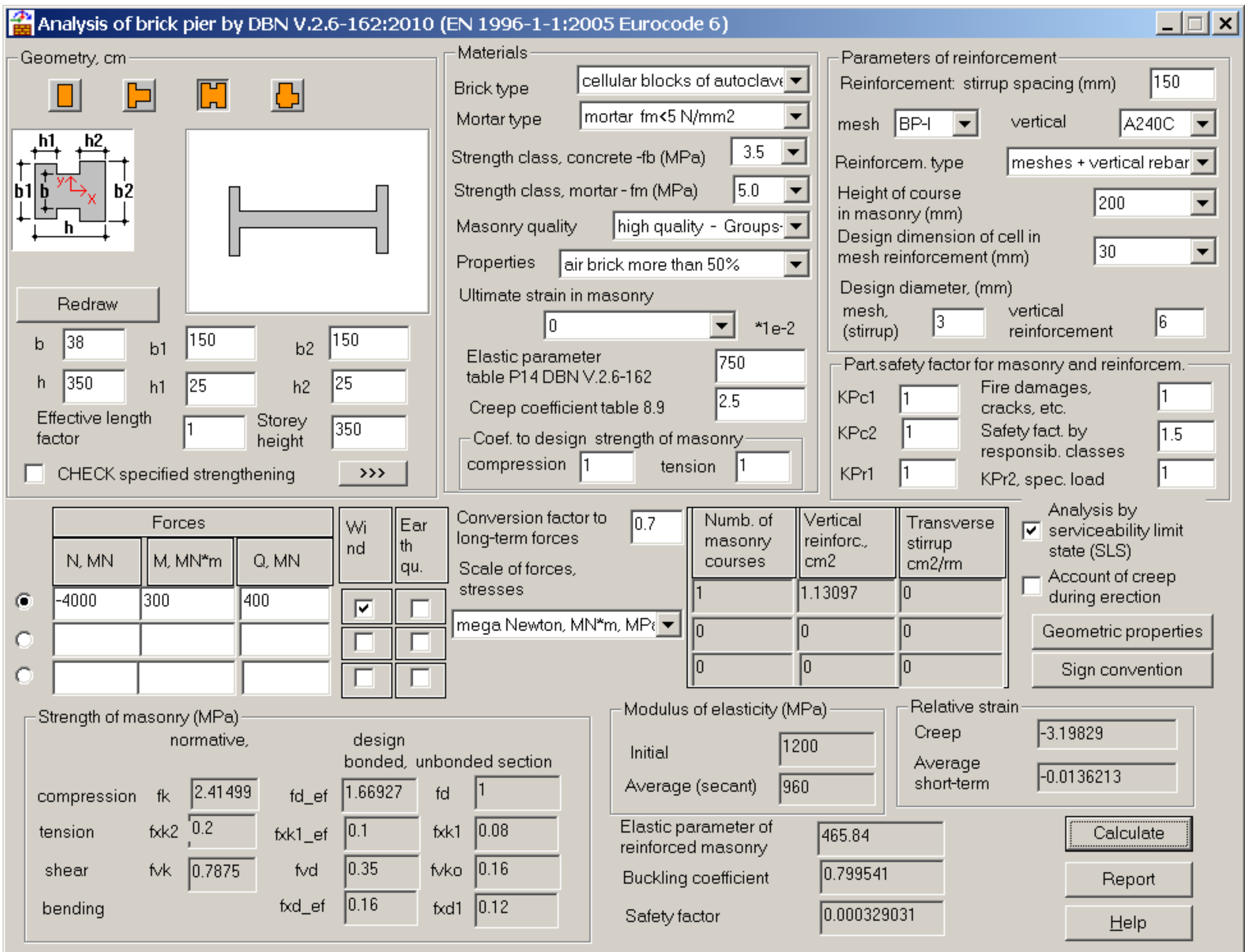


Figure 6.7

For piers of T-section, I-section and cruciform section, the lowest length of transverse wall (b1) is taken into account in calculation.

$b1 \leq 0.4*H_{storey} + b$;

$b1 \leq h$;

$b1 \leq 12*h1 + b$;

Types of masonry, mortar and marks stipulated for calculation are presented in the Table 1.

Table 1.

No.	Material of wall
1	Class of mortar strength f_m (MPa): M20, …M2.0 Class of (characteristic) strength for brick, stone f_b (MPa): 50,…M3- M0 (for packing of voids with concrete – class of strength for concrete: f_{ck} – characteristic compressive strength, f_{cvk} – shear strength: C12\15 –C25\30 and more)
2	clay brick, standard
3	sand-lime brick
4	vibrated masonry
5	rocks and large heavy concrete blocks Hcourse >50 cm
6	stones of concrete, gypsum concrete, etc. on porous aggregate Hcourse =20-30 cm
7	blocks of cellular concrete, slag concrete, etc. (hollow) Hcourse =20-30 cm (non-autoclave hardening)
8	natural stones (regular shape) low strength
9	natural stones from rubble
10	non-vibrated rubble concrete
11	vibrated rubble concrete
12	aerated concrete blocks of autoclave hardening with average density 400kg\m^3
13	concrete blocks of autoclave hardening from lightweight, cellular, aerated concrete; class is determined by average density D
14	Types of reinforcement for the wall: 0 – no reinforcement; 1 – mesh reinforcement, 2 – vertical rebars, 3 – meshes + vertical rebars, 4 – vertical external strengthening (Fig. 5.4)

The following types of mortar are available:
– mortar f_m<5 H/mm^2;
– mortar f_m≥5 H/mm^2;
– light with density ≤ 800 kg/m^3;
– light with density >800 kg/m^3;
– thin-layer adhesive, cement paste.

Masonry quality:
– Poor – group 5;
– Standard – group 4;
– High – group 3, group 2,
– Superior – group 1 (joint pointing or all vertical joint of masonry are filled with mortar).
– Compressive strength of mortar (f_m) that is used in horizontal joints of reinforced masonry should be $f_m \geq 2$ H/mm^2,(~204 tf/m^2), for reinforced masonry with filling of vertical joints $f_m \geq 4$ H/mm^2.

Class of concrete for packing of vertical voids in masonry should be not less than C12\15.

Loads

The following types of total forces and their long-term part are defined for the pier
N – axial force, minus sign '–' denotes compression (tf);
M – moment, positive moment denotes the rib in tension (tf*m);
Q – shear force (tf).

If the long-term part of forces is defined as equal to zero, then it is assumed that
$N_{long\text{-}term} = N/1.2$; $M_{long\text{-}term} = M/1.2$; $Q_{long\text{-}term} = Q/1.2$

Strength properties of material for wall are taken according to defined marks for brick, mortar, type and quality of masonry and after analysis they are displayed on the screen.

If elastic parameter of masonry is defined as $\alpha_o=0$, its value will be computed automatically. All design values for strength properties of material are taken automatically based on defined values for: brick/stone mark, mortar mark and mortar type (see Table 2).

Table 2.

No.	Material properties	Notation
1	Design compressive strength of masonry	$f_d = f_k/\gamma_m$
2	Design tensile strength for unbonded masonry section	f_{xd1}
3	Design tensile strength for bonded masonry section Conventional design flexural strength of reinforced masonry along bonded section	f_{xd2} f_{xd2_app}
4	Design flexural strength for bonded masonry section	f_{xd}
5	Design flexural strength for unbonded masonry section	f_{xd1}
6	Design shear strength for bonded masonry section	f_{vd}
7	Design shear strength for unbonded masonry section	f_{vko}/γ_m
8	Initial characteristic shear strength for masonry	f_{vko}
9	Characteristic shear strength for masonry: $f_{vk} = f_{vko}+0.4\sigma_o$ or $f_{vk} =0.5 f_{vko}+0.4\sigma_o$ if not all vertical courses of masonry are filled with mortar	$f_{vk} \leq 0.065 f_b$ $f_{vk} \leq 0.045 f_b$
10	Characteristic flexural strength for unbonded masonry	f_{xk1}

	section	
11	Characteristic flexural strength for bonded masonry section	f_{xk2}
12	Characteristic strength (temporary resistance) for unbonded masonry section	f_k
13	Normative mean compressive strength for masonry	f_b
14	Compressive strength of mortar	f_m
15	Temporary resistance of reinforced masonry R_{sku}	$f_{xd2\_app} * \gamma_m$
16	Elastic parameter of masonry $\alpha_o = E_o / R_u$ Initial modulus of elasticity of masonry $E_o = \alpha_o * R_u$	$\alpha_o = E_o / f_d$ $E_o = \alpha_o * f_d$
17	Ultimate creep coefficient	$\Phi\infty$
18	Elastic parameter of reinforced masonry $\alpha_{sk} = E_o / R_{sku}$	$\alpha_k = E_o / f_{xd2\_app}$

Strength properties for reinforcement and types of reinforcement (wire mesh or meshes + vertical rebars) are taken according to user-defined classes of reinforcement by DSTU 3760-07 'Reinforcing bars'.

Characteristic initial strength of masonry in shear for zero compression – $\mathbf{f_{vko}}$ is presented in the Table 3 H/mm^2=MPa (≈kg/cm^2)

Table 3.

Elements of masonry	Mortar of general purpose by class of strength *		Thin-layer mortar (adhesive)	Light mortar
Clay brick, standard	M10 – M20 M2.5 – M9 M1 – M2	0.3 (3.0) 0.2 (2.0) 0.1 (1.0)	0.3 (3.0)	0.15 (1.5)
Sand-lime brick	M10 – M20 M2.5 – M9 M1 – M2	0.2 (2.0) 0.15(1.5) 0.1 (1.0)	0.4 (4.0)	0.15 (1.5)
Concrete	M10 – M20	0.2 (2.0)	0.3 (3.0)	0.15 (1.5)
Aerated concrete (aeroc, etc.)	M2.5 – M9	0.15(1.5)	0.3 (3.0)	0.15 (1.5)
Precast concrete blocks, facing natural stone	M1 – M2	0.1 (1.0)	0.3 (3.0)	0.15 (1.5)

* Provided that the mortar of general purpose is manufactured by DSTU Б В.2.7-23, DSTU-П БВ.2.7-126 and does not contain additives.

Design fundamentals

In strength analysis and analysis of wire mesh reinforcement in piers, the following data is considered: defined height for the course of masonry, dimension for the cell of wire mesh, design diameter of wire mesh and vertical reinforcement. That's why, if after analysis of wire mesh reinforcement the program displays the message '*Reinforcement is not adequate for strength of section*', then it is necessary to increase design diameter or reduce dimension for the cell in wire mesh reinforcement, or modify the type of reinforcement to 'Meshes + vertical rebars'. In this case, if mesh reinforcement is not adequate, the program will automatically select vertical reinforcement.

In calculation of buckling coefficient Φ_m , the slenderness ratio of an element λ_h, λ_i is computed with account of effective length factor (height of pier H) - p_n. By default, $p_n=1$ is assumed for the case of pin joint on rigid (fixed) support.

$$\lambda_h = H_{ef} / t_{ef} ; \quad \lambda_i = H_{ef} / i$$

$$H_{ef} = H_{storey} \cdot p_n$$

where

t – wall thickness or

t_{ef} – effective thickness of wall;

H_{ef} – effective height of wall;

i – the smallest radius of gyration;

λ_h – slenderness ratio of masonry should not be greater than 27.

Effective length factor for the pier may be modified.

The calculation procedure includes the following checks.

1. Check of unreinforced masonry structures in compression of pier (by bearing capacity).

$$N \le \Phi_M \cdot \gamma_{b1} \cdot f_d \cdot L \cdot t_{ef}$$

where

Φ_m – coefficient to account slenderness and eccentricity of the section; coefficient is determined according to DBN as:

$$\Phi_m = A_1 \cdot e^{-u^2/2}$$

e – base of natural (Napierian) logarithm;

$$A_1 = 1 - 2\frac{e_0}{t_{ef}}$$

$$u = \frac{\lambda - 0.063}{0.73 - 1.17\frac{e_0}{t}}$$

$$\lambda = \frac{H_{ef}}{t_{ef}}\sqrt{\frac{f_k}{E}}$$

t_{ef} – effective equivalent thickness of the pier;
e_o – eccentricity at the middle of the wall height:
$e_o = e_m + e_k + e_{init}$,
where
$e_m = M_{design} / N_{design}$;
e_{init} – initial eccentricity that takes account of initial imperfections $e_{init} = h_{ef} / 450$.

$$e_k = 0.002 \cdot \Phi_\infty \cdot \frac{H_{ef}}{t_{ef}} \cdot \sqrt{e_m \cdot t}$$

- eccentricity from the creep may not be considered if λ_c =15 (recommended value for slenderness ratio for compressed part of the pier);
γ_{b1} – partial safety factor for masonry (influence of damages, strengthening, support method for the pier, etc.);
f_d – design compressive strength of masonry;
L – length of pier section.
If the area of pier section A is less than $0.1m^2$, then reduction factor (formula 6.3) k = (0.7 + 0.3 A) is applied in calculation;

2. Check in eccentric compression with flexure:

$$N \le \gamma_{b1} \cdot \varphi_1 \cdot f_d \cdot L_c \cdot t_{ef}$$

where

$$\varphi_1 = (\Phi_m + \Phi_c)/2$$

Φ_c – coefficient to account slenderness of compressed part of the section;
L_c – length of compressed part of the pier section;
t_{ef} – equivalent thickness of the section.

3. Check of unreinforced wall for ultimate moment in horizontal loads:

$$M \le f_{xd} \cdot W ; \qquad M \le f_{xd1\_app} \cdot W$$

W – elastic section modulus per unit of length or height of the section;
f_{xd} – design flexural strength for unbonded section;

$$f_{xd1\_app} = f_{xd1} + \sigma_0$$

For the constant compression of wall and taking into account $\sigma_0 \leq 0.2\ f_d$.

4. Check of unreinforced masonry structures in shear along horizontal unbonded sections (by bearing capacity) – the smallest value is applied:

$$V_{Ed} \leq V_{Rd} = \gamma_{b1}\left(\frac{f_{vko}}{\gamma_M} + 0.4 \cdot \sigma_0\right) \cdot A_c$$

$$V_{Rd} \leq \gamma_{b1} \cdot f_{vd} \cdot L_c \cdot t_{ef}$$

where
f_{vko}/γ_M – design shear resistance for unbonded section;
σ_o – mean value of compressive stress;
For the constant compression of wall and taking into account $\sigma_0 \leq 0.2\ f_d$.

5. For the serviceability limit states (SLS), the check with account of long-term creep modulus E_{long}:

$$E_{long} = \frac{E_k}{1 + \Phi_\infty}$$

Φ_∞ – ultimate creep coefficient is taken according to material type of masonry;

$f_k = f_d * \gamma_M$ - characteristic (ultimate) strength of masonry;
f_d – design strength of masonry;
γ_M – coefficient for responsibility of material is assigned according to classes (groups) from: group – 1 (highest), up to group – 5 (lowest). By default, it is accepted that $\gamma_M = 1.5$ for structures of the 1st group with high architectural standards. This parameter may be modified.

Table 4.

	Coefficient for responsibility of material	γ_M for classes				
	Classes of responsibility	1	2	3	4	5
A	Stonework from: blocks of category I, design building mortar according to EN998-2, EN1996-2.	1.5	1.7	2.0	2.2	2.5
B	Stonework from: blocks of category II, specified building mortar according to EN998-2, EN1996-2.	1.7	2.0	2.2	2.5	2.7
C	Stonework from: blocks of category II, arbitrary building mortar according to EN998-2, EN1996-2, *e.	2.0	2.2	2.5	2.7	3.0
D	Anchor from reinforced steel	1.7	2.0	2.2	2.5	2.7
E	Reinforcing steel and pre-tensioned steel	1.15				
F	Additional components *c *d	1.7	2.0	2.2	2.5	2.7
*c	Accepted values are mean ones.					
*d	It is accepted that waterproofing should be covered with building mortar.					
*e	If variability coefficient for masonry of category II does not exceed 25%.					

6. For masonry reinforced with meshes in horizontal joints, additional check of conditional design strength in flexure for unbonded section - f_{xd2_app} . If mesh reinforcement is not adequate, the program will select vertical reinforcement:

$$M \le f_{xd2\_app} \cdot W$$

$$f_{xd2\_app} = \frac{6 \cdot A \cdot f_{yd} \cdot Z}{t^2}$$

where
f_{yd} – design strength of reinforcement in horizontal joints;
A – area of cross section of tensile reinforcement of meshes per 1 r.m.;
Z – lever arm of internal pair of forces ≤ 0.95*d;
X – distance to the neutral axis.

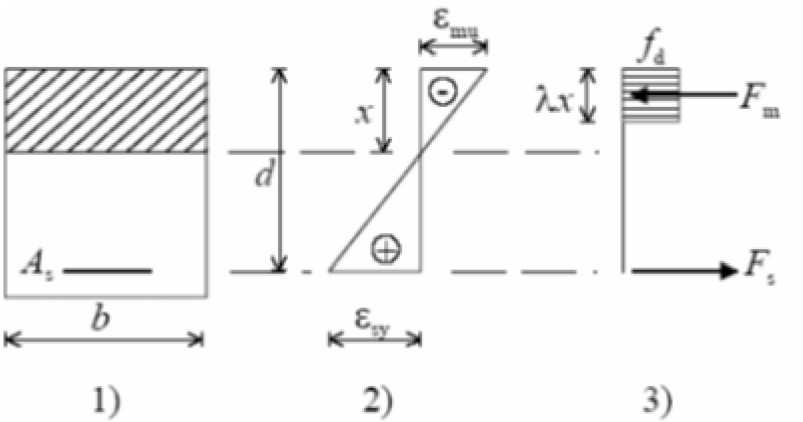


Figure 6.8. Distribution of stresses in reinforced section.
1 – design section; 2 – strain; 3 – internal forces.

Ultimate moment for elements of masonry of group 1 (except masonry with light aggregates) in flexure could not be greater than:

$$M \leq 0.4 \cdot f_d \cdot b \cdot d^2$$

or for elements of groups 2,3,4, and group 1 from lightweight concrete:

$$M \leq 0.3 \cdot f_d \cdot b \cdot d^2$$

7. Wire mesh reinforcement is selected by defined dimension of cell in wire mesh, diameter and class of reinforcement. The wire mesh reinforcement with the following dimensions are allowed: 30, 35, 40, 45, 50, 55, 60, 65, 70,75, 80, 85, 90, 95, 100 mm.

8. Check of reinforced masonry structures in shear along bonded sections :

$$V \leq \gamma_{b1} \cdot f_{vd} \cdot A + \frac{\gamma_s \cdot f_{yd} \cdot Fa_{xy} \cdot A}{100}$$

where
f_{vd} – design shear resistance for bonded section ;
f_{yd} – design strength of transverse (wire meshes in vertical section or vertical) reinforcement;
Fa_{xy} – percentage of reinforcement for the section reinforced with wire meshes along vertical section.

In this case, f_{vd} (H/mm^2) is increased (Supplement П, DBN) in vertical reinforcement located in voids:

 www.liraland.com

$$f_{vd} = \frac{0.35 + 0.175\ Fa_z}{\gamma_m} \le \frac{0.7}{\gamma_m}$$

9. If conditions by sect.6 and sect.8 are not satisfied, then vertical reinforcement is selected.

$$N \le \gamma_{b1} \cdot f_k \cdot A + \frac{\gamma_s \cdot f_{yd\_z} \cdot Fa_z \cdot A}{100}$$

where
f_{yd_z} – design compressive strength of vertical reinforcement;
Fa_z – percentage of vertical reinforcement in the section;
f_{sk} – design strength of masonry reinforced with wire meshes:

$$f_{sk} = f_k + \frac{2 \cdot Fa_x \cdot f_{yd\_x}}{100} \le 2 \cdot f_k$$

10. For the check of sect.9, vertical reinforcement is selected and additional check is carried out by ultimate moment based on double vertical reinforcement:

$$M \le \gamma_{b1} \cdot f_k \cdot A_c \cdot \left(h_0 - 0.5\ x\right) + \gamma_s \cdot f_{yd\_z} \cdot A_z^{'} \cdot \left(h_0 - a^{'}\right)$$

where
X – height of compressed zone of the section;
a – thickness of concrete cover (assumed as a = a') ;
h_o – effective height of section.

The following calculations are made:

1. Mean short-term relative strain is calculated as:

$$\varepsilon = \frac{-1.1 \cdot \ln\left(\dfrac{1-\sigma}{1.1 \cdot f_k}\right)}{\alpha_0}$$

where f_k – characteristic strength of masonry.

2. Secant modulus (short-term) of strain in masonry is calculated as (see Fig.6.9):

$$E_k = \frac{\sigma}{\varepsilon}$$

where
ε – relative strain of masonry.

To evaluate complete strain of the structure with account of long-term loads (or with account of creep in freezing) and check masonry by ultimate strain, the program computes complete ultimate creep strain in masonry as:

$$\varepsilon_{total} = \varepsilon \cdot \Phi$$

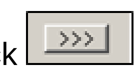
$$E_{long} = \frac{E_k}{1+\Phi}$$

where
Φ – creep coefficient:
Φ =0.5 - 1.5 – for masonry of structural clay tile with vertical slit-like cells;
Φ =1.0 -2.0 – for masonry of large concrete blocks and artificial stones;
Φ = 1.0 -2.0 – for masonry of sand-lime brick solid and hollow and of concrete on porous aggregates (or aerated) and large sand-lime blocks;
Φ=1.0 -3.0 – for masonry of ultralight precast concrete (small and large blocks);
Φ =0.5 - 1.5 – for masonry of aerated autoclave cellular concrete;
ε_u = -0,0035 – for masonry of group 1;
ε_u = -0,0020 – for masonry of group 2, 3, 4 (see Fig.6.9).

Strength of brickwork during the bricklaying (for f_k <2.5 MPa) is checked according to obtained creep strain in masonry. If design strain exceeds allowed values, it is necessary to provide measures for strengthening of masonry up to the moment when it reaches necessary strength (temporary supports, etc.).

Strength is not checked according to serviceability limit states (SLS).

Check of specified strengthening

To check the specified strengthening of the brick pier, click 

button and define the type of strengthening and appropriate numerical data.

Then in the main window of the program, select the 

check box and defined parameters of strengthening (area of vertical strengthening and area of transverse stirrups) will be automatically displayed in the main window of the program.

The following patterns for strengthening of brick posts with casings are available:
1. **Strengthening with steel casing** (strengthening with angles, stirrups of strips or circular rebars, etc., Fig. 6.10-a) – additional check by axial force by formula 71 Manual [2].
2. **Strengthening with reinforced concrete (RC) casing** (strengthening with angles, stirrups of strips or circular rebars, etc., Fig. 6.10-b) – additional check by axial force by formula 72 Manual [2].
3. **Strengthening with rebars in reinforced plaster \mortar casing** (see Fig. 6.10-c) – additional check by axial force by formula 73 Manual [2].

Strengthening of masonry with steel, RC and mortar casing is checked based on:

– vertical reinforcement defined in the main window of the program;
– type of strengthening defined in additional window of the program (Fig. 6.11).
It is also possible to select area of vertical strengthening.
Check of stirrup sections is made in accordance with the following data: specified diameter and class of wire mesh reinforcement, specified spacing of stirrups and area of stirrups per 1r.m. of the section height.

Calculation of battens in steel casing as elements of trusses (without diagonal elements) in shear force and bending moment in the plane of battens – this option is not available in current version of the program.

Recommendations.

1. For analysis of masonry with damages (cracks) – partial safety factor for masonry should be taken according to method of TSNIISK (see Table 5):

Table 5.

No.	Nature of damage in masonry of walls, posts and piers	Unreinforced masonry	Reinforced masonry
1	Cracks in certain bricks - does not intersect mortar joints	1.0	1.0
2	Hairline cracks that intersect not more than two courses of masonry (length 15-18cm)	0.9	1.0
3	Hairline cracks that intersect not more than four courses of masonry (length up to 35cm) if number of cracks is not more than four cracks per 1r.m. of width (thickness) of wall, post, pier	0.75	0.9
4	Cracks with propagation up to 2mm that intersect not more than eight courses of masonry (length up to 60-65cm) if number of cracks is not more than four cracks per 1r.m. of width (thickness) of wall, post, pier	0.5	0.7
5	Cracks with propagation up to 2mm that intersect more than eight courses of masonry (length up more than 60-65cm) if number of cracks is not more than four cracks per 1r.m. of width (thickness) of wall, post, pier	0	0.5

2. Based on support conditions for the pier, partial safety factor for strengthened masonry is taken as equal to:
KPc1=1.0 – when load is transferred to the casing and support is at the bottom of casing.
KPc1=0.7 – when load is transferred to the casing and there is no support at the bottom of casing.
KPc1=0.35 – without load transferring to the casing and if there is no support at the bottom of casing.
If there are two of above-mentioned factors, then factors will be multiplied.

3. To consider reduced design strength of reinforcement applied to the casing (table 10 Manual [2]), reduced partial safety factors for reinforcement are introduced KPr1= 0.8 etc. in case conditions of age-hardening of reinforcement are taken into account (corrosion, fire damages, etc.).

4. Check for strength in walls and brick piers at places where beams and other elements are supported with them is made as check for local bearing strength in analysis on local compression.

To check strength for both non-damaged elements and elements that have cracks in the masonry (e.g. fire damages because of temperature, fire or old masonry), it is recommended to use regulations [2] and [3]. In this case, the program checks bearing capacity of centrally and eccentrically loaded elements strengthened with steel, RC and mortar casing. The user could select appropriate elements for strengthening, e.g. angles, plates or exterior vertical rebars connected with transverse stirrups (see Fig. 2.4) and define the area of exterior strengthening.

Messages at the time of computation:
1. If selected wire meshes do not satisfy condition of 'Max allowed reinforcement', the following message is displayed:
'Reinforcement - exceeds max allowed value.'

2. If it is not possible to arrange reinforcement, the following message is displayed:
'Reinforcement is not adequate for strength of section.'

3. If reinforcement is selected successfully, the following message is displayed:
'Reinforcement in masonry is selected.'

4. In analysis in tension, the following warning is displayed:
'Analysis of brick pier in Tension for unbonded section. According to DBN, such elements cannot be designed for unbonded section.'

5. If eccentricity of axial force is outside the section (or near the utmost edge of the section), the following warning is displayed:
'Distance from point of load application up to compressed edge of section is less than 2cm. It is not allowed to design such structures without vertical reinforcement.'

6. If the strength of section is not adequate, the following message is displayed:

'Strength of section is not adequate. Increase mark of brick and mortar'.

7. If the slenderness ratio of the brick pier exceeds 27, the following message is displayed: *'Slenderness of the wall exceeds allowed value'.*

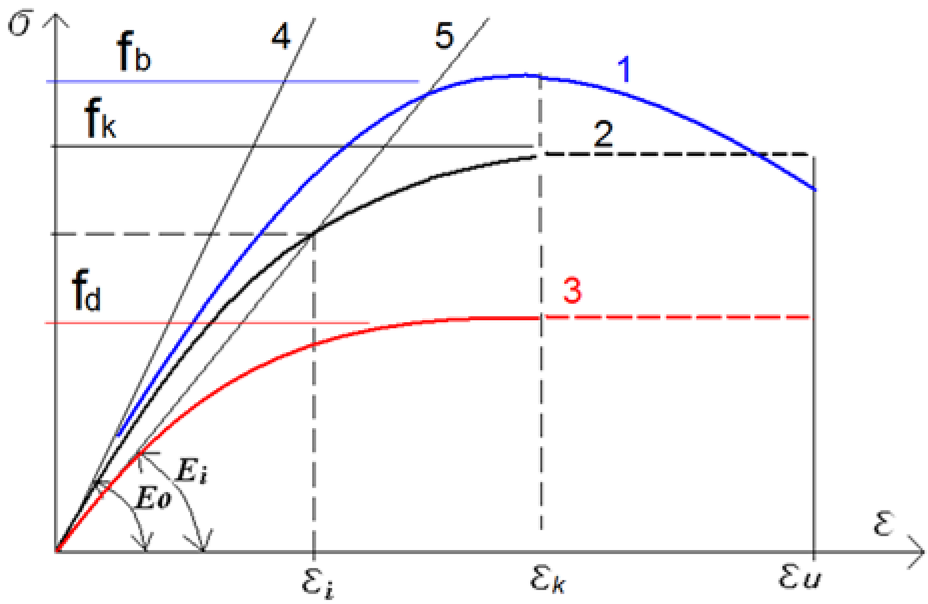


Figure 6.9. Modulus of elasticity of masonry.

E_0 – initial modulus of elasticity (4)
E_i – secant modulus of elasticity (5)
ε_k – ultimate short-term strain
ε_u – ultimate long-term strain
1 – physically nonlinear normalised compressive strength of a masonry unit - f_b
2 – idealized characteristic compressive strength of masonry - f_k
3 – idealized design compressive strength of masonry - f_d.

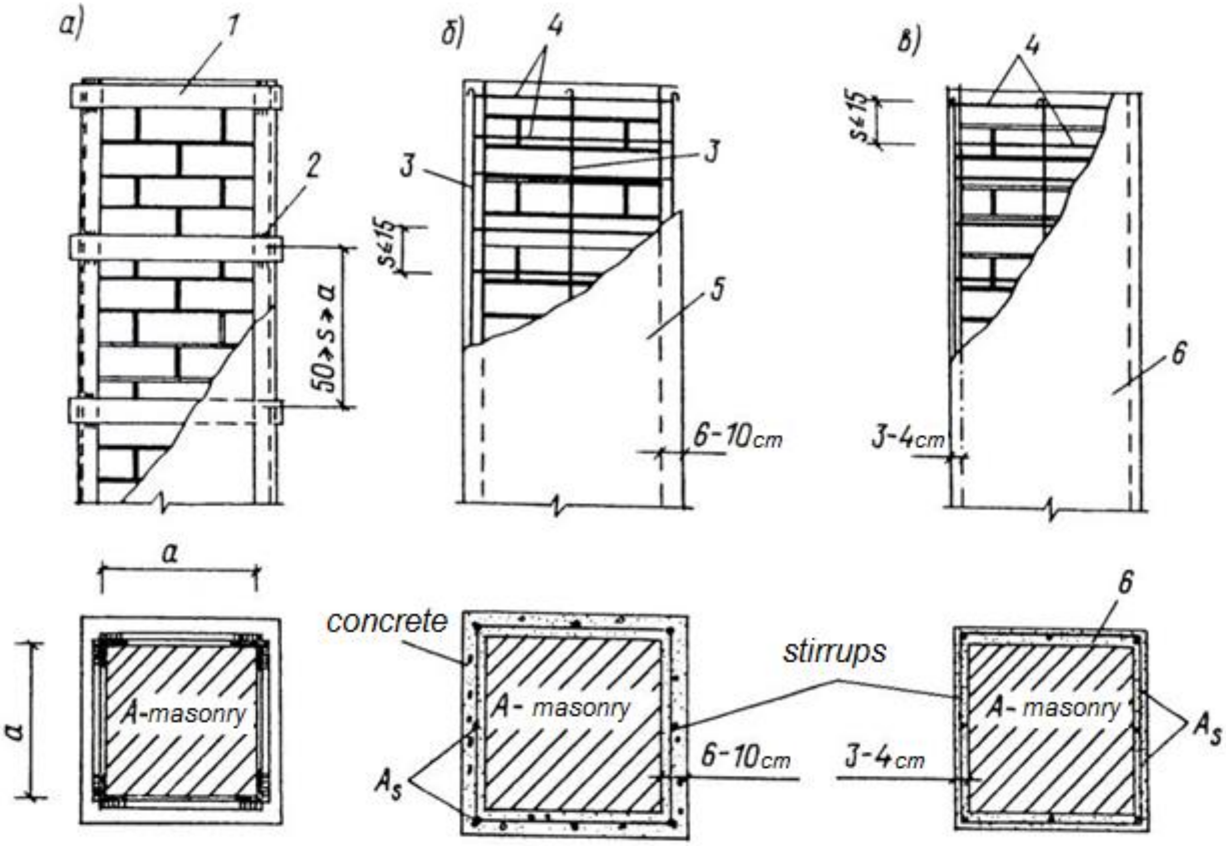
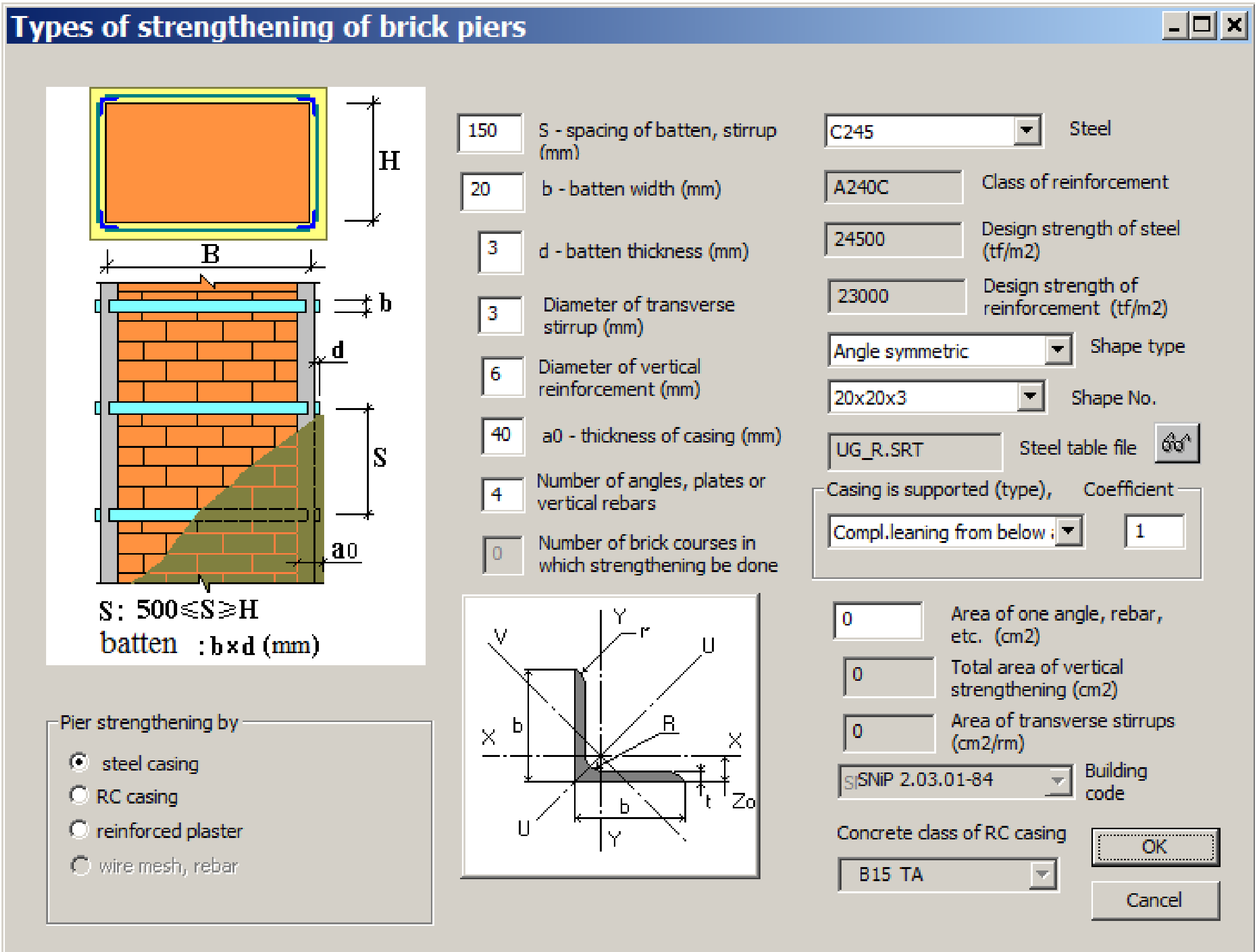


Fig. 6.10. Patterns for strengthening of brick piers with casing:
a) steel casing; b) RC casing; c) reinforced plaster.

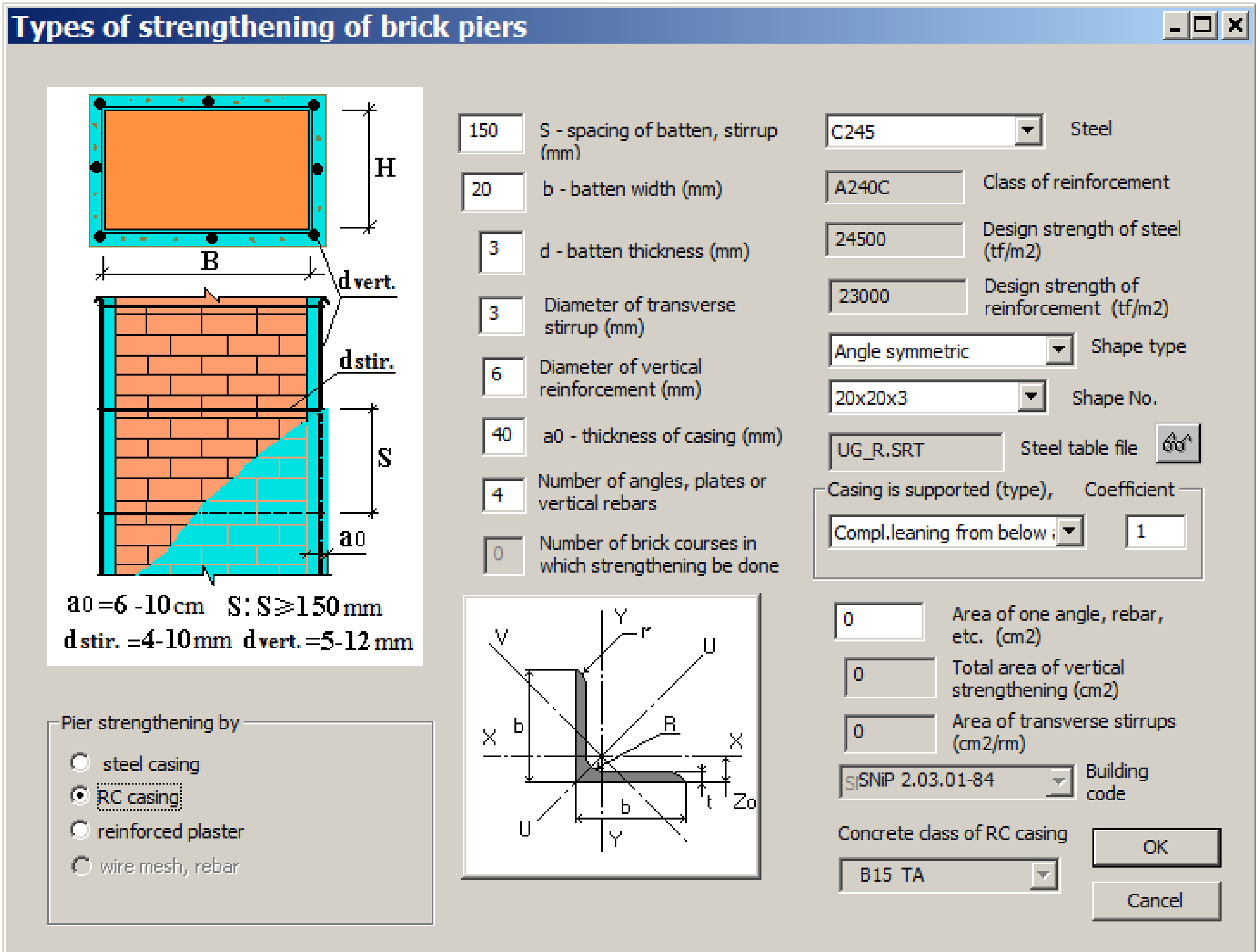
1 – batten with section 35x5- 60x12mm;
2 – welding;
3 – rebars with diameter 5-12mm;
4 – stirrups with diameter 4-10mm;
5 – concrete of class B7.5- B15;
6 – plaster (mortar of mark M50 – M100).

Fig. 6.11. Types of strengthening of brick piers with casing:

a) Strengthening with steel casing

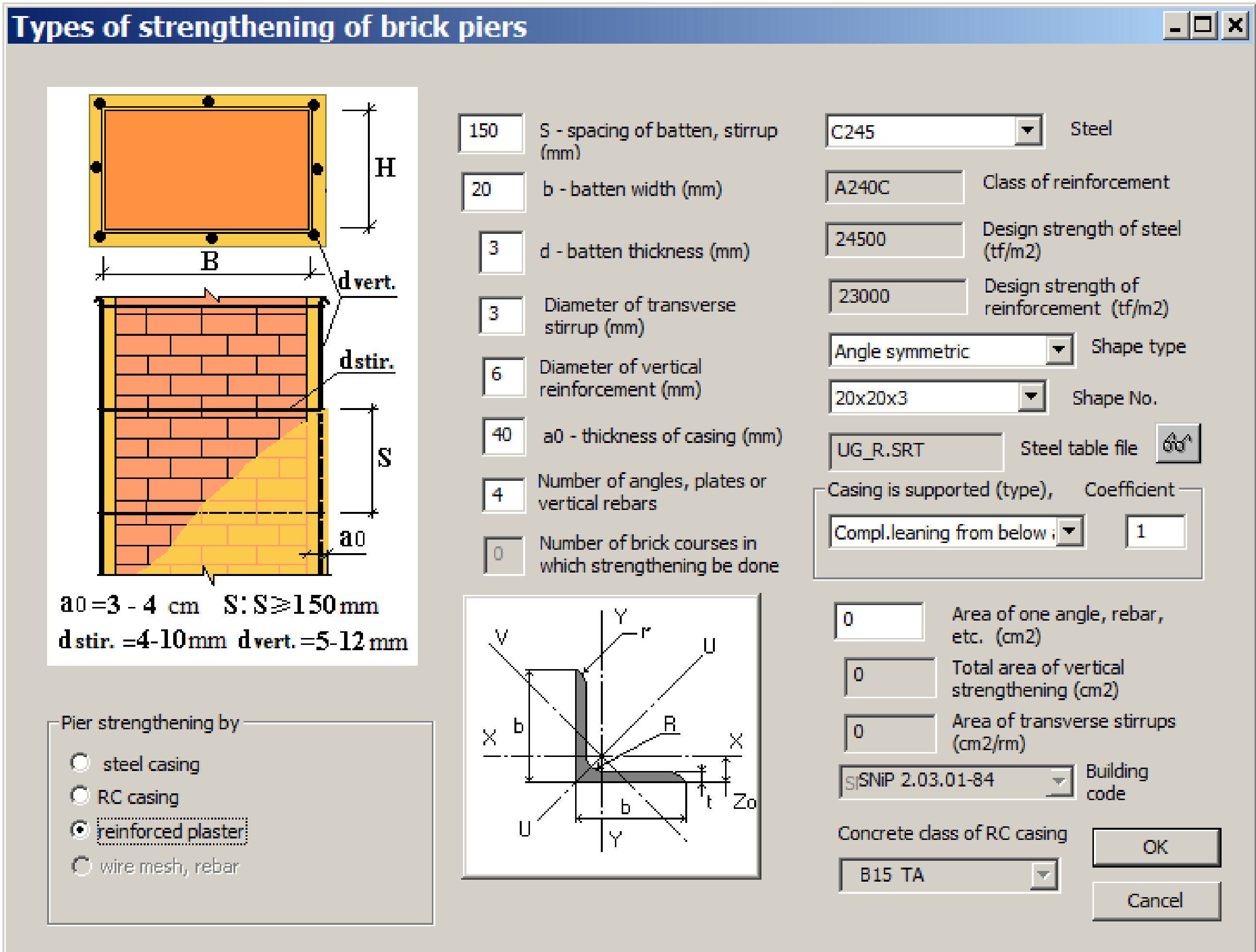


b) Strengthening with RC casing



 www.liraland.com

c) Strengthening with mortar casing



Timber structures

Timber structures

The chapter contains 6 modules for analysis of elements in timber structures:
Sections of solid timber by SP 64.13330 2011;
Sections of glued timber by SP 64.13330 2011;
Sections of composite timber by SP 64.13330 2011;
Sections of solid timber by Eurocode 5;
Sections of glued timber by Eurocode 5;
Sections of composite timber by Eurocode 5.

Figure 7.1

Sections of solid timber

These modules enable you to carry out analysis of rectangular and round bars in solid timber sections according to SP 64.13330.2011 and Eurocode 5.

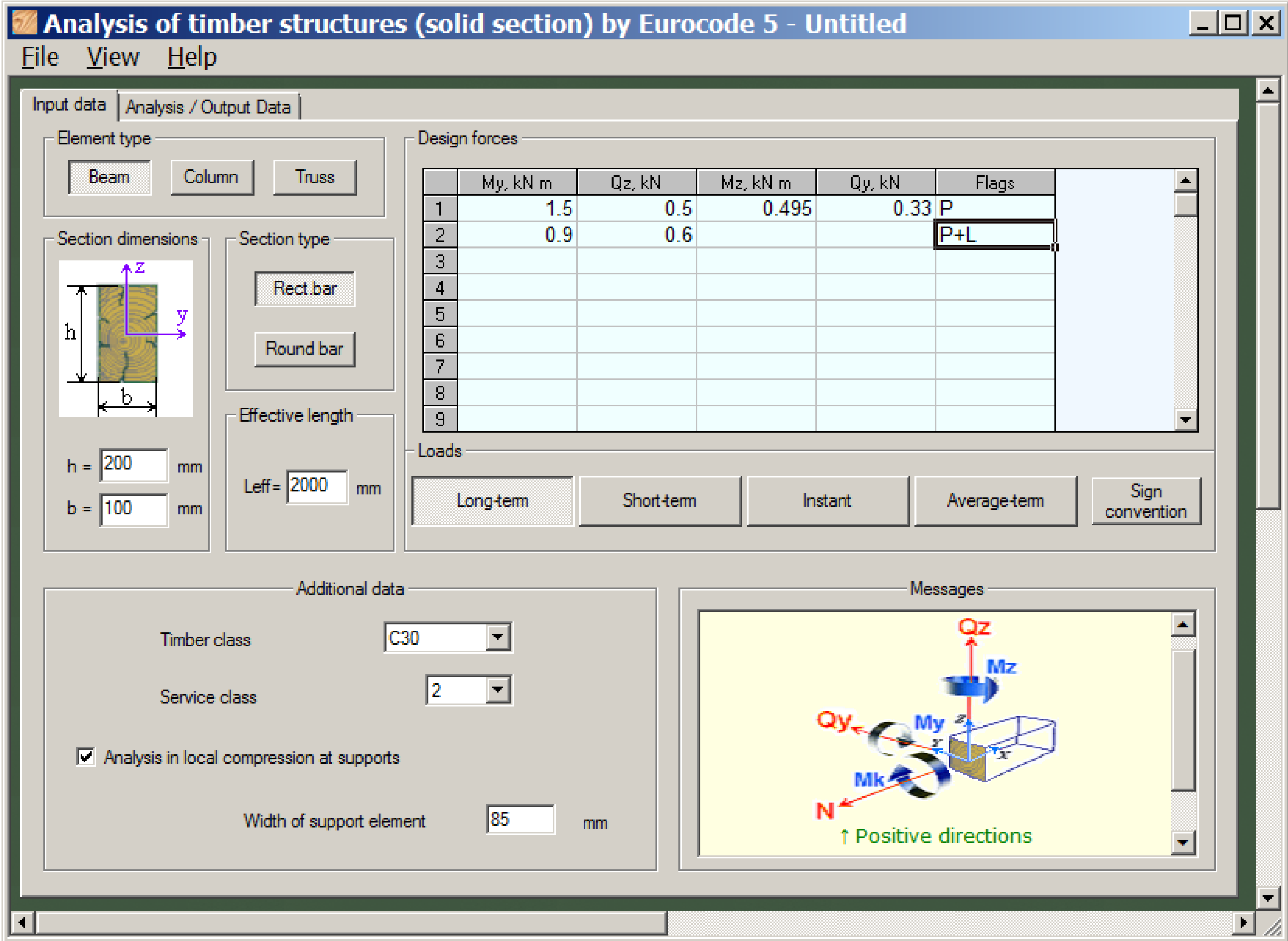


Figure 7.2

Output data may be presented either in abridged format or in detail with formulas and utilization ratio for sections according to appropriate check procedures.

Input data

In the **Element type** area, click appropriate button to select the type of element:
 – **Beam** – flexural element (design forces – My, Mz, Qy, Qz);
 – **Column** – eccentrically-compressed (tensile) element (design forces – N, My, Mz, Qy, Qz);
 – **Truss** – axially-compressed (tensile) element (design force – N).
Select the **Section type**: **Rectangular bar** or **Round bar**.
Define **Section dimensions**: height *h* and width *b* for rectangular section or diameter *d* for round section.

If the **Beam** type is selected, then define:
- for Eurocode 5 – effective length of element *Leff*;
- for SP 64.13330.2011 – length between supports *Lp* (that is, distance between fixities of compressed fibre of an element from displacement out of plane of bending in intermediate points).

If **Column** or **Truss** type is selected, then define *loy, loz* – effective lengths of element relative to The Y-axis and Z-axis respectively ('relative axis' means the plane that is perpendicular to the axis).
For the round section of column or truss element, define greatest effective length *lo*.

In the **Additional data** area, define all required parameters for the element according to SP 64.13330.2011 or Eurocode 5 and coefficient *γ* (design resistance of timber will be multiplied by this coefficient). Coefficient is defined by the user according to appropriate building code.

In the **Design forces** table, define one or several values for design combinations of forces. In this case, to determine design resistance of materials according to SP, for the specified rows of design combinations of forces (DCF) it is necessary to select the **Stresses from dead and live loads exceed 80% of all loads** check box if it is true for the certain DCF row.* (* - Only for calculation by SP 64.13330.2011).
To define short-term loads available in the combination, click appropriate buttons in the **Loads** area. These loads will be denoted in the **Flags** column in the **Design forces** table.

To display schematic presentation with positive directions of forces in element sections, click **Sign convention**.

Output data

To carry out analysis and then evaluate results, click **Analysis / Output data tab**. Analysis results are presented as utilization percentage by appropriate criteria – ratio of bearing capacity of the section to actual stress, slenderness, etc. You will also obtain summary info whether certain criteria (strength, ultimate slenderness, etc.) are adequate or not.
Output data may be printed directly from the program window or you could save it to the report file (see FILE menu, **Save report as** command).
To display max values for each check, on the VIEW menu, click **Characteristic DCF only**.
To display the report in detail with numerical formulas for check procedures, on the VIEW menu, unselect the **Report abridged** command.

Sections of glued timber

These modules enable you to carry out analysis of glued sections in timber structures according to SP 64.13330.2011 and Eurocode 5.

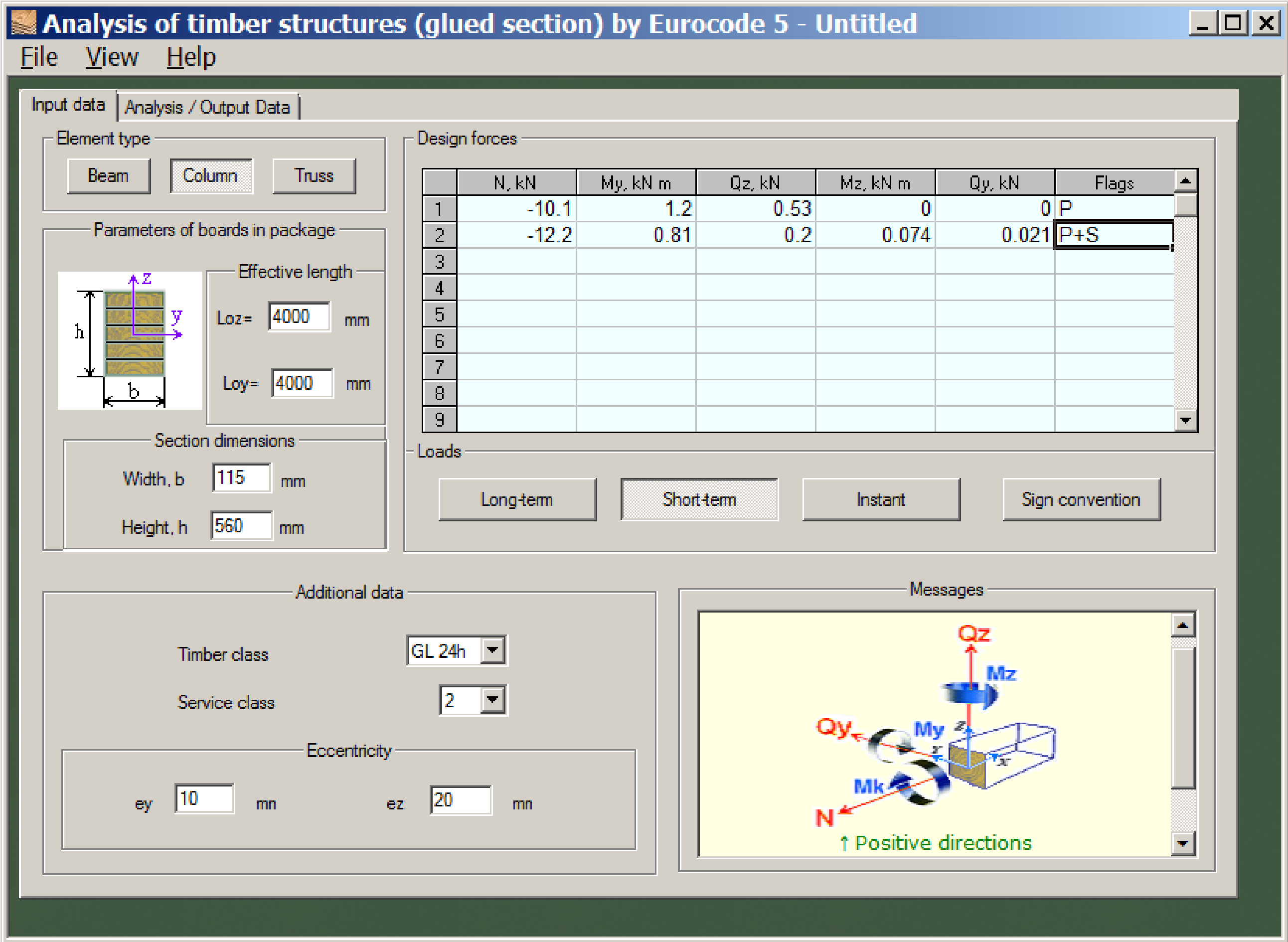


Figure 7.3

Input data

In the **Element type** area, click appropriate button to select the type of element:

– **Beam** – flexural element (design forces – My, Mz, Qy, Qz);

– **Column** – eccentrically-compressed (tensile) element (design forces – N, My, Mz, Qy, Qz);

– **Truss** – axially-compressed (tensile) element (design force – N).

Define the following data:
- for analysis by SP - parameters of boards in package;
- for analysis by Eurocode - section dimensions.

If the **Beam** type is selected, then define:
- for Eurocode 5 – effective length of element *Leff*;
- for SP 64.13330.2011 – length between supports *Lp* (that is, distance between fixities of compressed fibre of an element from displacement out of plane of bending in intermediate points).

If **Column** or **Truss** type is selected, then define *loy, loz* – effective lengths of element relative to The Y-axis and Z-axis respectively ('relative axis' means the plane that is perpendicular to the axis).

In the **Additional data** area, define all required parameters for the element according to SP 64.13330.2011 or Eurocode 5 and coefficient γ (design resistance of timber will be multiplied by this coefficient). Coefficient is defined by the user according to appropriate building code.

In the **Design forces** table, define one or several values for design combinations of forces. In this case, to determine design resistance of materials according to SP, for the specified rows of design combinations of forces (DCF) it is necessary to select the **Stresses from dead and live loads exceed 80% of all loads** check box if it is true for the certain DCF row.* (* - Only for calculation by SP 64.13330.2011).
To define short-term loads available in the combination, click appropriate buttons in the **Loads** area. These loads will be denoted in the **Flags** column in the **Design forces** table.

To display schematic presentation with positive directions of forces in element sections, click **Sign convention**.

Output data

To carry out analysis and then evaluate results, click **Analysis / Output data tab**. Analysis results are presented as utilization percentage by appropriate criteria – ratio of bearing capacity of the section to actual stress, slenderness, etc. You will also obtain summary info whether certain criteria (strength, ultimate slenderness, strength in bending (chipping), etc.) are adequate or not.
Output data may be printed directly from the program window or you could save it to the report file (see FILE menu, **Save report as** command).
To display max values for each check, on the VIEW menu, click **Characteristic DCF only**.
To display the report in detail with numerical formulas for check procedures, on the VIEW menu, unselect the **Report abridged** command.

Sections of composite timber

These modules enable you to carry out analysis of composite sections in timber structures according to SP 64.13330.2011 and Eurocode 5.

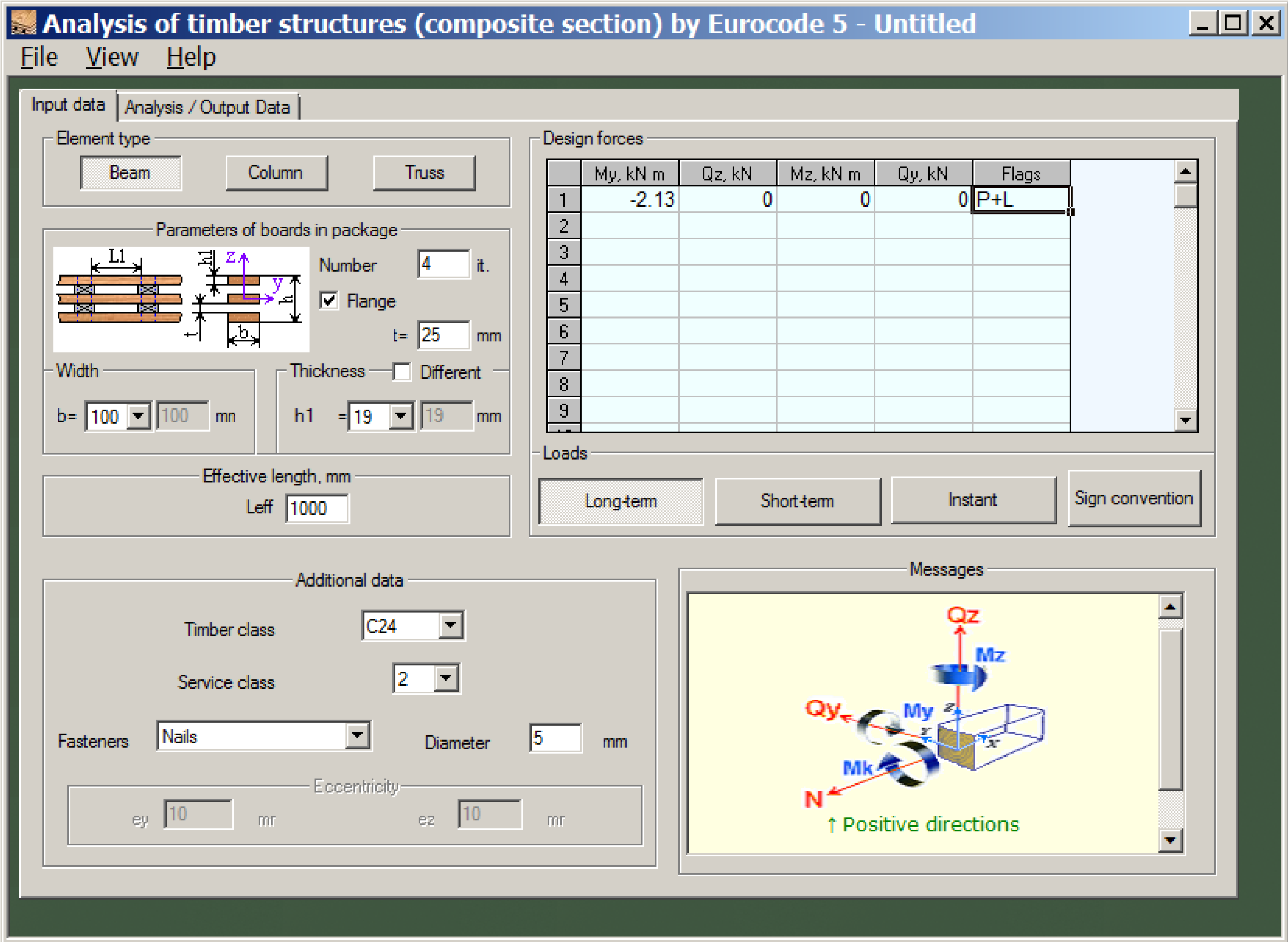


Figure 7.4

Input data

In the **Element type** area, click appropriate button to select the type of element:

– **Beam** – flexural element (design forces – My, Mz, Qy, Qz);

– **Column** – eccentrically-compressed (tensile) element (design forces – N, My, Mz, Qy, Qz);

– **Truss** – axially-compressed (tensile) element (design force – N).

Define section dimensions. To do this, in the **Parameters of boards in package** area, define number of boards (not less than 2 items), section dimensions depend on the number of boards in package. Thickness may be defined as different for every board or the same for all boards in the package. Width and height of the cross-section of boards in package may be selected according to the certain building code in the drop-down list or it is possible to define these values manually. To define these values manually, in the appropriate drop-down list, select <..> and then input required value.

If height of the section is increased with intermediate pieces (flanges), select the **Flanges** check box and define their thickness.

If the **Beam** type is selected, then define:

- for Eurocode 5 – effective length of element *Leff*;

- for SP 64.13330.2011 – length of span *L* and length between supports *Lp* (that is, distance between fixities of compressed fibre of an element from displacement out of plane of bending in intermediate points).
If **Column** or **Truss** type is selected, then define web length *l1* and *loy, loz* – effective lengths of element relative to The Y-axis and Z-axis respectively ('relative axis' means the plane that is perpendicular to the axis).

In the **Additional data** area, define all required parameters for the element according to SP 64.13330.2011 or Eurocode 5 and coefficient γ (design resistance of timber will be multiplied by this coefficient). Coefficient is defined by the user according to appropriate building code.

In the **Design forces** table, define one or several values for design combinations of forces. In this case, to determine design resistance of materials according to SP, for the specified rows of design combinations of forces (DCF) it is necessary to select the **Stresses from dead and live loads exceed 80% of all loads** check box if it is true for the certain DCF row.* (* - Only for calculation by SP 64.13330.2011).
To define short-term loads available in the combination, click appropriate buttons in the **Loads** area. These loads will be denoted in the **Flags** column in the **Design forces** table.

To display schematic presentation with positive directions of forces in element sections, click **Sign convention**.

Output data
To carry out analysis and then evaluate results, click **Analysis / Output data tab**. Analysis results are presented as utilization percentage by appropriate criteria – ratio of bearing capacity of the section to actual stress, slenderness, etc. You will also obtain summary info whether certain criteria (strength, ultimate slenderness, strength in bending (chipping), etc.) are adequate or not.
Output data may be printed directly from the program window or you could save it to the report file (see FILE menu, **Save report as** command).
To display max values for each check, on the VIEW menu, click **Characteristic DCF only**.
To display the report in detail with numerical formulas for check procedures, on the VIEW menu, unselect the **Report abridged** command.

Foundations and beddings

Foundations and beddings

The chapter contains modules for analysis of footings and foundations:

[Moduli of subgrade reaction C1, C2](#)
[Single pile](#)
[Pile under combined action of loads](#)
[Settlement of equivalent footing](#)
[Principal and equivalent stresses in soil](#)
[Slope stability](#)
[Stability of multi-layer slope](#)
[Bearing capacity of piles by field test results](#)
[Combined piled-raft foundation](#)

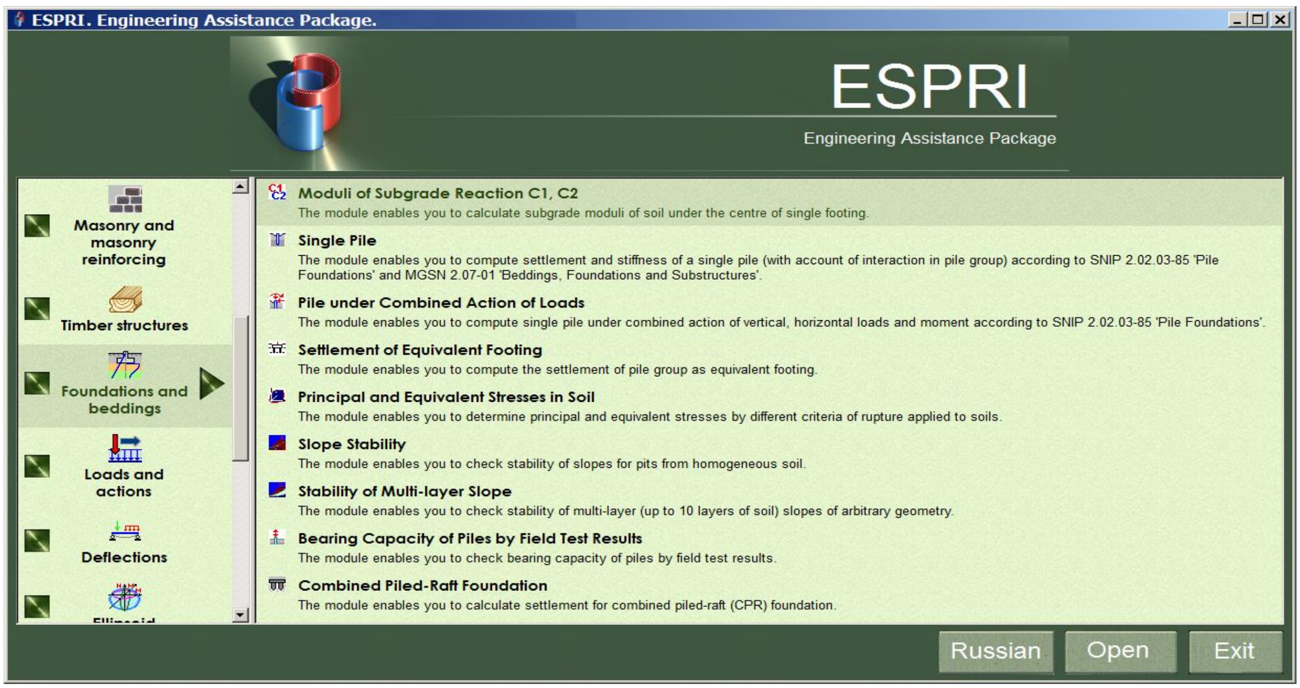


Figure 8.1

The following building codes are supported in the module:
SNIP 2.02.01– 83*. Foundation beds for buildings and structures;
SP 50–101–2004. Foundation beds for buildings and structures;
SNIP 2.02.03–85. Pile foundations;
SP 50–102–2003. Design and installation of pile foundations;
MGSN 2.07–01. Footings, foundations and underground structures;
DBN V.2.1-10:2009. Footings and foundations of structures;
SP 22.13330.2011. Foundation beds for buildings and structures;
SP 24.13330.2011. Pile foundations.

Moduli of subgrade reaction C1, C2

This module enables you to calculate moduli of subgrade reaction of the soil **C1** and **C2** under the centre of column foundation or foundation slab. Analysis may be carried out by different methods based on computation of settlement according to various building codes.

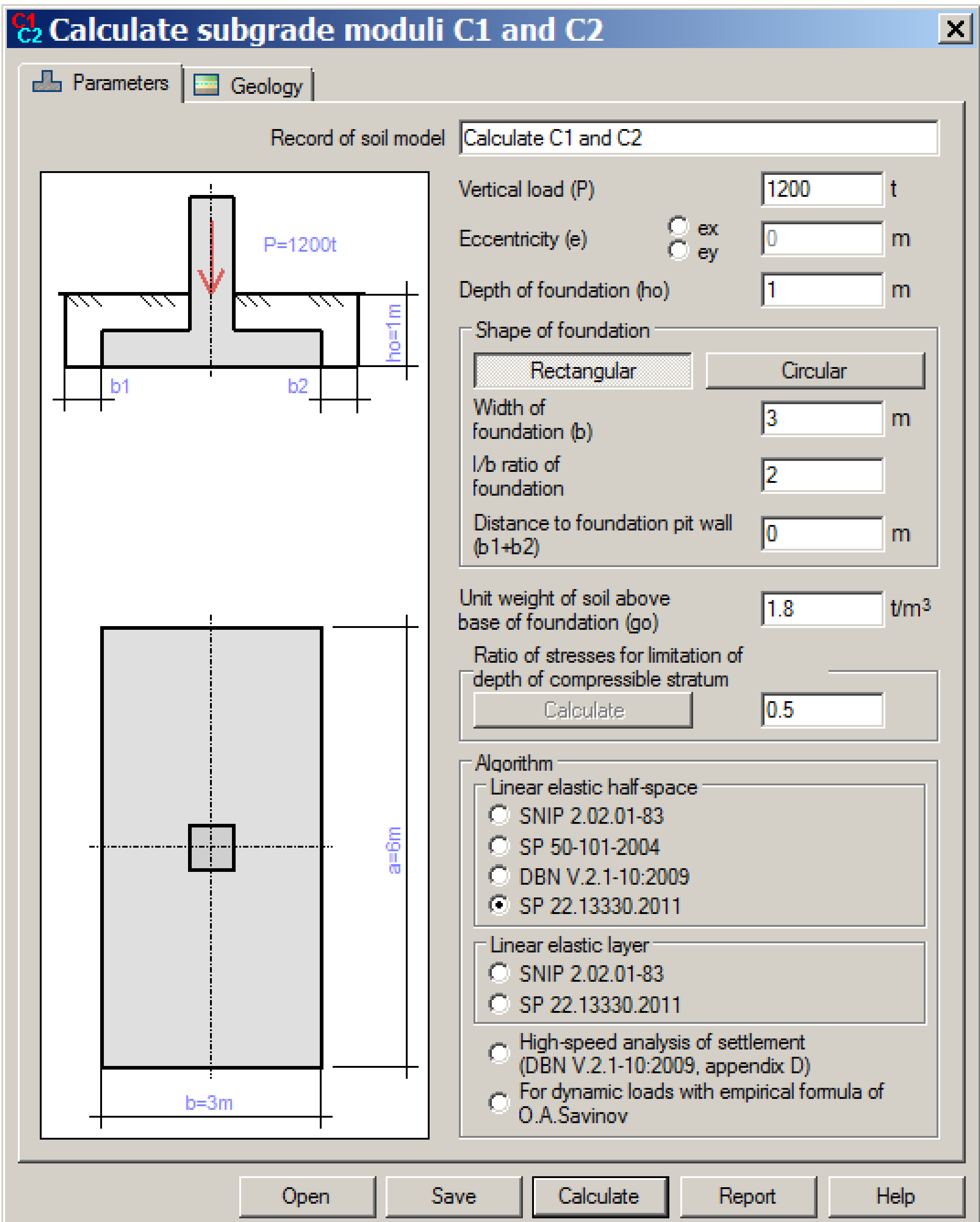


Figure 8.2 a)

Input data

On the **Parameters** tab, in the **Shape of foundation** area, click **Rectangular** or **Circular** and define dimensions for foundation.
On this tab, define the following input data:
– vertical load on foundation;
– eccentricity of load application;
– depth of foundation;
– unit weight of soil above foundation base;
– ratio of stresses for limitation of depth of compressible stratum.

To determine the lower depth of compressible stratum, you should define the ratio of additional vertical stress to vertical stress from dead weight of soil.

Define appropriate algorithm and building code for calculation:
– analysis by linear elastic half-space (LHS) according to sect. 1–6 Appendix 2 SNIP 2.02.01–83*, according to SP 50-101-2004 or DBN V.2.1-10:2009;
– analysis by linear elastic layer (LEL) according to sect. 7–8 Appendix 2 SNIP 2.02.01–83*;
– high-speed analysis of settlement according to DBN V.2.1-10:2009;
– for dynamic loads, with empirical formula of O.A.Savinov.

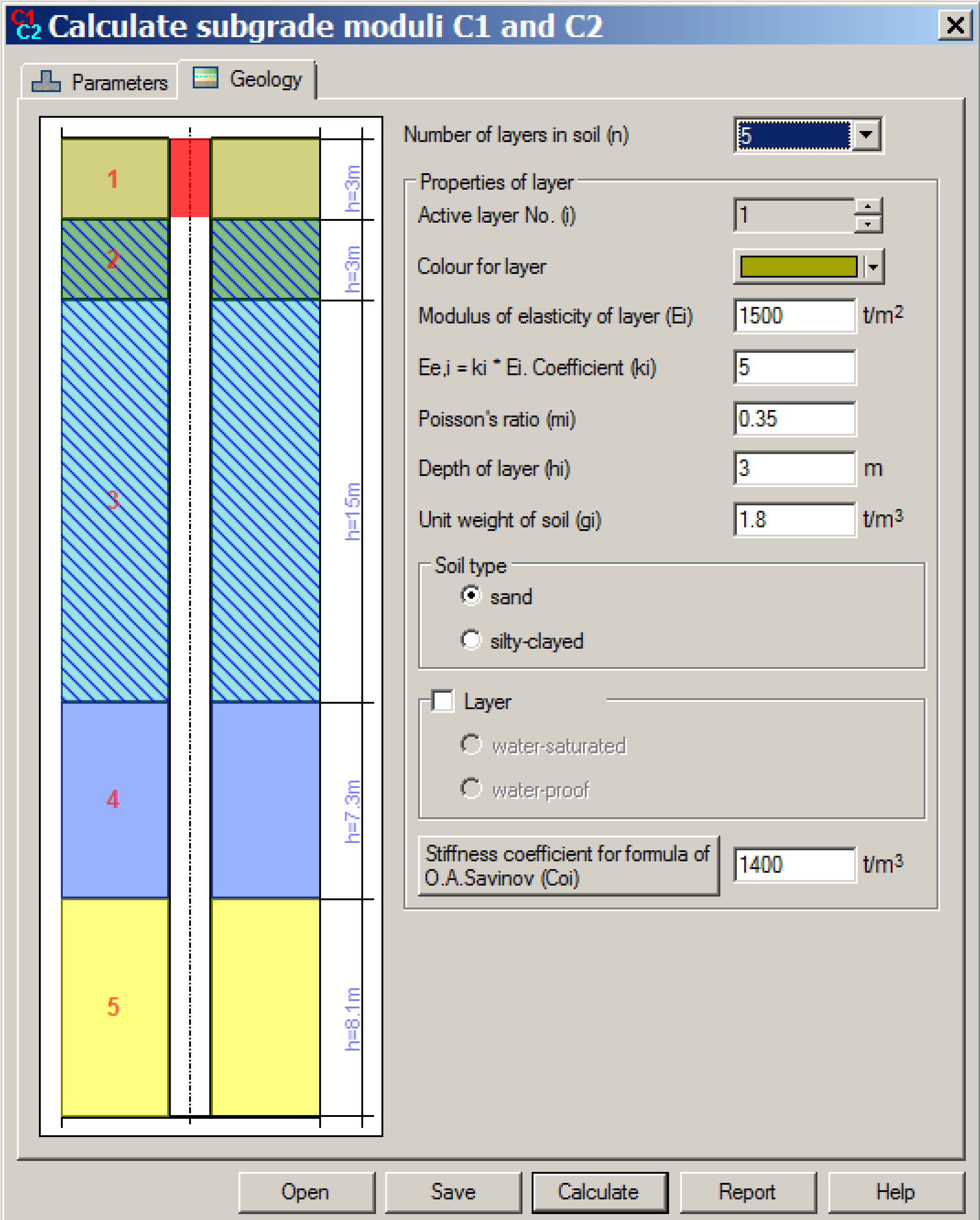


Figure 8.2 b)

On the **Geology** tab, define the number of soil layers and properties for every soil layer:
E – modulus of elasticity for soil;
υ – Poisson's ratio;
h – thickness of layer;
γ – unit weight of soil.

For analysis by linear elastic layer (LEL) , define the soil type: either sand or silty-clayed.
In the **Layer** area, define whether the layer is water-saturated or water-proof.
To apply empirical formula of O.A.Savinov, define the stiffness coefficient appropriate for soil layer.

 www.liraland.com

Click **Calculate**.

Output data

The output data contains the following values:
S – settlement of foundation;
Hc, Hm – depth of compressible stratum for algorithm of linear elastic half-space and linear elastic layer respectively;

E_{gr} – mean value of modulus of elasticity of soil;

m_{gr} – mean value of Poisson's ratio;

E_{gr3} – modified modulus of elasticity of soil. It is calculated on the assumption that modulus of elasticity of soil increases along the depth and additional stress is distributed uniformly;

i – tangent of rotation angle of foundation;
C1, C2 – moduli of subgrade reaction of soil in compression and shear.

Diagram for distribution of vertical stresses is presented according to selected building code.
To present analysis results as report, click appropriate button. In the **Report generation** dialog box, define the path and name for the report file, click **Generate** and then click **Preview**.
Report may be generated for the following data: parameters, geology and analysis results.

Brief description of algorithm.

1) Regardless of selected algorithm, the depth of compressible stratum **Hc** is determined according to LHS (linear elastic half-space) with account of requirements of sect.7-8 Appendix 2 SNIP 2.02.01-83*, SP 50-101-2004, SP 22.13330.2011, DBN V.2.1-10:2009 based on formula (1):

$\sigma_{zp} = \lambda \sigma_{zg}$, (1)

where

σ_{zp} – additional vertical stress at depth $Z = H_c$;

σ_{zg} – vertical stress from dead weight of soil.

λ – coefficient stipulated in the appropriate building codes and depending on type of soil and type of structure, as a rule λ = 0.2.

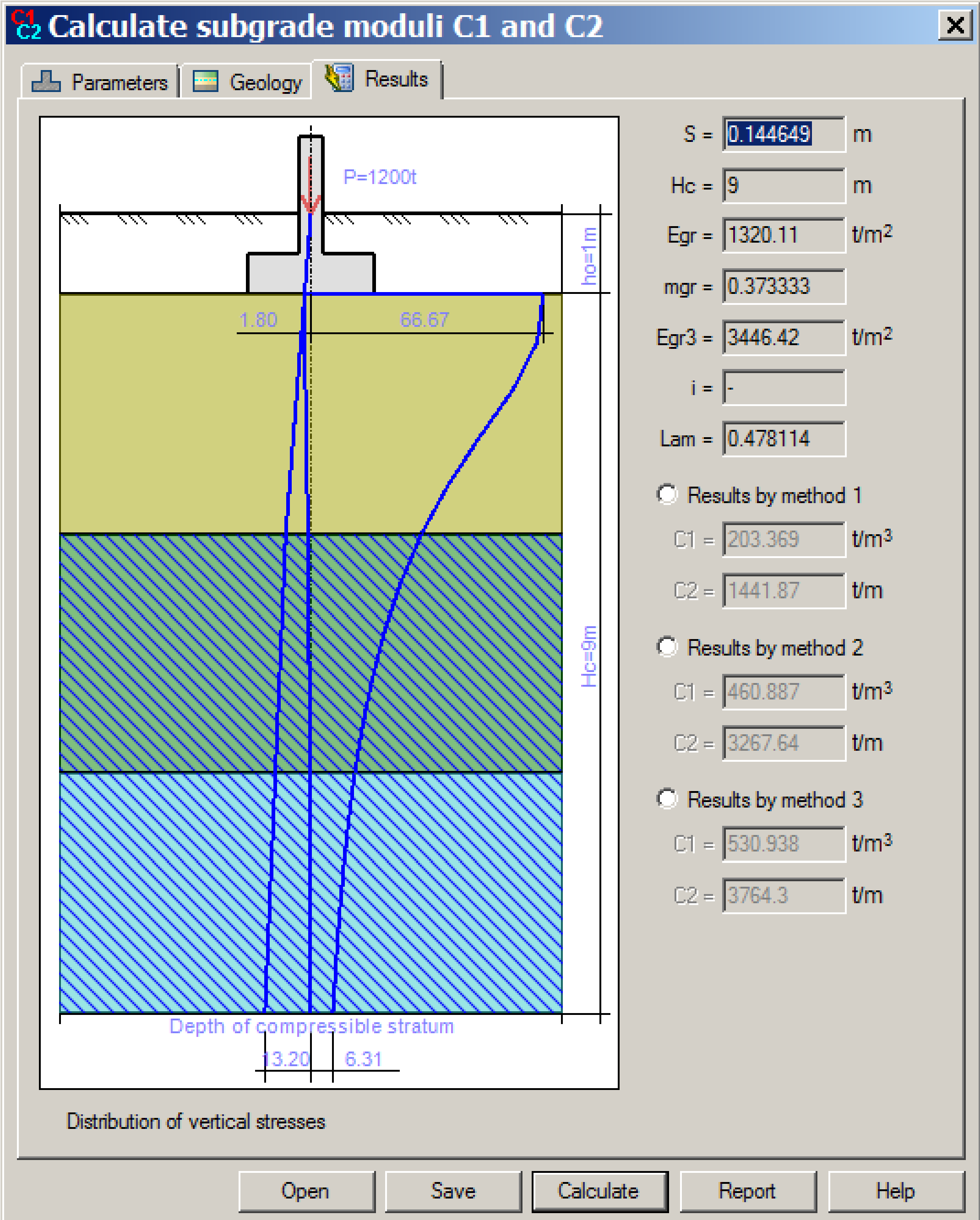


Figure 8.3

Analysis by linear elastic half-space (LHS) according to SNIP 2.02.01-83*

2) Settlement (**S**) of foundation according to linear elastic half-space is calculated by layer-by-layer sum according to the formula:

$$S = 0.8W \quad (2)$$

where

$$W = \sum_{i=1}^{n} \frac{\sigma_{zp,i} h_i}{E_i}; \quad (3)$$

 www.liraland.com

h_i, E_i – thickness and modulus of elasticity of the **i-th** layer of soil (with account of its division into sublayers) respectively;

n – number of soil layers from the base of foundation up to the depth $Z = H_c$ with account of division into sublayers (**i=1~n**).

Analysis by linear elastic half-space (LHS) according to SP 50-101-2004 and DBN V.2.1-10:2009

3) The following limitations are imposed on the values of **λ** and depth **Hc**:
a) when b ≤ 5 - λ = 0.2, but **Hc** not less than b/2;
b) when 5 < b ≤ 10 - λ = 0.02*b+0.1, but **Hc** not less than b/2;
c) when 10 < b ≤ 20 - λ = 0.02*b+0.1, but **Hc** not less than (4+0.1*b);
d) when 20 < b - λ = 0.02*b+0.1, but **Hc** not less than (4+0.1*b).
b - smaller side of foundation.

Coefficient **λ** and ultimate values of **Hc**.

b, m	***λ***	***Hc*, m not less than**	***b,*** m	***λ***	***Hc*, m not less than**
1 -:- 5	0.2	2.5	15	0.4	5.5
6	0.22	3.0	16	0.42	5.6
7	0.24	3.5	17	0.44	5.7
8	0.26	4.0	18	0.46	5.8
9	0.28	4.5	19	0.48	5.9
10	0.3	5.0	20	0.5	6.0
11	0.32	5.1	30	0.5	7.0
12	0.34	5.2	40	0.5	8.0
13	0.36	5.3	50	0.5	9.0
14	0.38	5.4	100	0.5	10.0

4) Settlement is calculated in the following way:

$$W_1 = \sum_1^n \frac{(\sigma_{zp,i} - \sigma_{zy,i})h_i}{E_i};$$

$$W_2 = \sum_1^n \frac{\sigma_{zy,i}\, h_i}{E_{ei}};$$

$$W_3 = \sum_1^n \frac{\sigma_{zp,i}\, h_i}{E_{ei}} \quad (5)$$

where
E_i – modulus of elasticity of the i-th layer of soil along the loading path;

E_{ei} – modulus of elasticity of the i-th layer of soil along the unloading path;
by default $E_{ei} = E_i$;
$\sigma_{zp,i}$ – stress at the i-th layer of soil from external load;
$\sigma_{zy,i}$ – stress at the i-th layer of soil from dead weight.

If dead weight of soil at the level of foundation base is greater than mean pressure under the foundation base, then W =W_3, otherwise W = W_1 + W_2. Settlement is calculated according to formula (2).

5) Mean (within the limits of fixed depth of compressible stratum **Hc**) values of modulus of elasticity E_{gr} and Poisson's ratio m_{gr} are used for calculation of subgrade moduli. These values are calculated according to formulas 11 and 12 Appendix 2 SNIP 2.02.01-83*.

$$E_{gr} = \frac{\sum_{i=1}^{n} \sigma_{zp,i} h_i}{H_c}; \quad m_{gr} = \frac{\sum_{i=1}^{n} \nu_i h_i}{H_c} \quad (5)$$

6) Modulus of subgrade reaction **C1** is calculated by three methods.

Method 1. Subgrade modulus **C1** is calculated by formula:

$$C_1 = \frac{E_{gr}}{H_c\left(1 - 2m_{gr}^2\right)}; \quad (6)$$

Method 2. Subgrade modulus **C1** is calculated according to Winkler method:

$$C_1 = \frac{q}{S}, \quad (7)$$

where

$q = \frac{P}{\eta * b^2}$ – mean pressure under the base of foundation;
b – width of foundation;
η – ratio of the longest dimension of foundation to the shortest one;
S – settlement of soil.

Method 3. Just as in method 1, formula (6) is used to determine subgrade modulus **C1**. The difference is that in this case correction factor u to modulus of elasticity of the **i-th** sublayer is introduced for calculation of mean modulus of elasticity. This factor varies from $u_1 = 1$ at the level of foundation base up to $u_n = 12$ at the level of calculated depth of compressible stratum. It is accepted that factor u varies according to parabola law:

$$u = \frac{11z^2}{H_C^2} + 1 \quad . \quad (8)$$

Moreover, it is also accepted that additional vertical stress is distributed uniformly along the depth. Then

$$E_{gr3} = \frac{H_C}{\sum_{i=1}^{n} \frac{h_i}{u_i E_i}} \quad (9)$$

Method 3 is suggested in order to remove limitations of the first two methods.
Limitation of method 1 – it is impossible to take into account that modulus of elasticity increases along the depth. It causes too high values of settlements and therefore too low values of subgrade modulus **C1**.

7) For all methods, subgrade modulus **C2** is calculated according to the formula:

$$C_2 = \frac{C_1 H_C^2 \left(1 - 2m_{gr}^2\right)}{6\left(1 + m_{gr}\right)} \quad . \quad (10)$$

Analysis by linear elastic layer (LEL)

8) Depth of compressible stratum **Hm** for linear elastic layer is determined with account of requirements of paragraph 2.40 and paragraphs 7-8 Appendix 2 SNIP 2.02.01-83*.
If these requirements are not satisfied, then the program displays a message and asks whether you want to continue calculation process. If yes, then it is accepted that **Hm=Hc** and calculation procedure is continued. Otherwise, it is terminated.

9) Mean (with account of layers **j=1~n** within the limits of fixed depth of compressible stratum **Hm**) values of modulus of elasticity E_{gr}**,** E_{gr3} and Poisson’s ratio m_{gr} are used for calculation of subgrade moduli by three methods. Correction factor u is calculated according to formula (8).

$$E_{gr} = \frac{\sum_{j=1}^{n}(k_j - k_{j-1})}{\sum_{j=1}^{n}\frac{(k_j - k_{j-1})}{E_j}};$$

$$m_{gr} = \frac{\sum_{j=1}^{n}\nu_j h_j}{H_m};$$

$$E_{gr3} = \frac{\sum_{j=1}^{n}(k_j - k_{j-1})}{\sum_{j=1}^{n}\frac{(k_j - k_{j-1})}{u_j E_j}} \quad (11)$$

10) Settlement **S** is calculated according to formula 7 Appendix 2 SNIP 2.02.01-83*.

11) Modulus of subgrade reaction **C1** is calculated by three methods. Formula (6) where **Hc** is replaced with **Hm** and formula (7) are used. Subgrade modulus **C2** is calculated according to formula (10) where **Hc** is replaced with **Hm**.

Quick calculation of settlement

12) The settlement may be calculated quickly according to formulas D.9 – D.11 and table D2, Appendix D, DBN V.2.1-1-:2009.

$$S = 1.44\frac{\eta}{\eta + 1}\frac{(p - \sigma_{zg,0})b}{E_c}, \quad (12)$$

where:

η =l/b – ratio of foundation length to foundation width;

p – mean pressure below foundation base;

$\sigma_{zg,0}$ – vertical pressure from the dead weight of soil at the level of foundation base;

E_c – mean modulus of elasticity, computed by formula

$$E_c = \frac{\sum_{1}^{n} E_i\, h_i\, z_i}{0.5\, H_c^2}, \quad (13)$$

where:

E_i – modulus of elasticity of the i–th layer of soil;

z_i – distance from the middle of the i–th layer to the lower depth of compressible stratum;

H_c – depth of compressible stratum, computed by formula

$H_c = k \cdot b$, (14)

k – coefficient from the table below.

η= l / b	Circular foundation	1.0	1.2	1.4	1.8	2.0	2.4	3.2	5.0	≥ 6.0
k	1.8	2.0	2.2	2.4	2.8	3.0	3.4	3.8	4.3	6.0

Analysis by formula of O.A.Savinov

13) Mean value of stiffness coefficient is determined based on stiffness coefficients C_{0i} specified for every layer of soil and calculated depth of compressible stratum **Hc**. Mean value of stiffness coefficient is calculated by formula

$$C_{gr0} = \frac{\sum_{i=1}^{n} C_{0i}\, h_i}{H_C} \quad (12)$$

14) Modulus of subgrade reaction C1 is calculated according to the following formulas: for rectangular foundation with dimensions **l*b**

$$C_1 = C_0\left(1 + \frac{2(l+b)}{\Delta l b}\right)\sqrt{\frac{q}{p_0}}; \quad (13)$$

for circular foundation with radius **R**

$$C_1 = C_0\left(1 + \frac{2}{\Delta R}\right)\sqrt{\frac{q}{p_0}}, \quad (14)$$

where C_{0i} – stiffness coefficient taken from the table;

p_0 – pressure determined from laboratory tests and equal to 2 t/m^2;

q – mean pressure under the base (t/m^2);

Δ – constant of elasticity of soil taken as equal to 1 (in 1/m).

15) Modulus of subgrade reaction **C2** is calculated according to the formula (10).

Table with stiffness coefficients
(N.A. Tsytovich, Soil mechanics, page 358)

No.	*Name of soil*	*C_0, tf/m3*
1	Sand: 1) silty, greatly saturated; 2) fine, regardless of density and humidity; 3) medium, coarse and gravel, regardless of density and humidity.	 800 – 1000 1000 – 1200 1200 – 1600
2	Clay, clay loam and sandy clay: 1) soft; 2) firm; 3) stiff.	 500 – 1000 1000 – 2000 2000 – 3000

Single pile

The module enables you to compute bearing capacity of single pile. It is possible to compute end-bearing piles, friction piles and screw piles according to SNIP 2.02.03-85 'Pile foundations', MGSN 2.07-01 and requirements of 'Manual for design of pile foundations'.

Input data

End-bearing piles and friction piles

– In the **Pile type** area, select the type of piles: driven piles, bored and drilled piles, shell piles.
– Select the type of pile cross-section and define its dimensions.
– Depending on the pile type, in the left part of the dialog box there may be some check boxes to define additional information.
– To compute settlement (for friction piles), select appropriate check box. In another dialog box that appear, select the building code, define appropriate parameters and data (if required) to take account of interaction of piles in the pile group.
– For **friction piles**, define soil properties along the side surface of the pile.

Screw piles

L – pile length ≤ 10;
D – diameter of pile shaft;
d1 – diameter of pile helix;
h1 – depth of penetration of pile helix;
A – in **compressive and sign variable loads** - area of projection of pile helix OR
– in **pullout loads** - difference between projection area of pile helix and projection area of pile shaft.

C1 – design value for soil resistance within work space (within soil layer with thickness d1 adjacent to the pile helix from above and below);
Angle of internal friction of soil within work space.
Load type – compression, pullout, sign variable.

In the **soil type** list, select required type of the *i*-th layer and define its thickness ($\boldsymbol{t_i}$), design value for unit weight ($\boldsymbol{\gamma_i}$) and design value for soil resistance along side surface of pile shaft ($\boldsymbol{f_i}$). To add or delete the soil layer, use appropriate buttons in the dialog box.

Click **Calculate**.

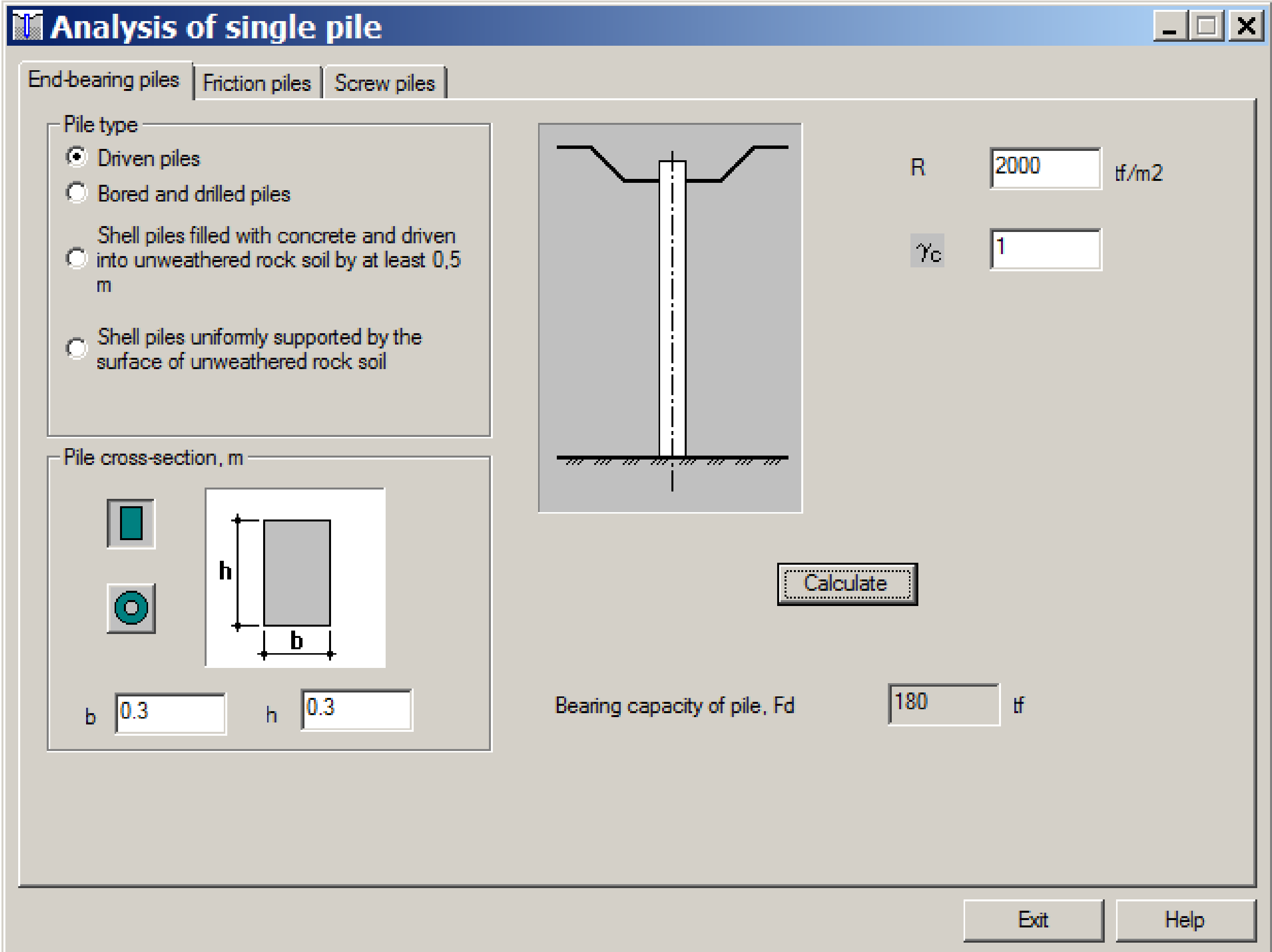

Analysis of single pile
End-bearing piles
Friction piles
Screw piles
Pile type
Driven piles
Bored and drilled piles
Shell piles filled with concrete and driven into unweathered rock soil by at least 0,5 m
Shell piles uniformly supported by the surface of unweathered rock soil
Pile cross-section, m
h
b
b 0.3
h 0.3
R 2000 tf/m2
γc 1
Calculate
Bearing capacity of pile, Fd 180 tf
Exit
Help

Figure 8.4 a)

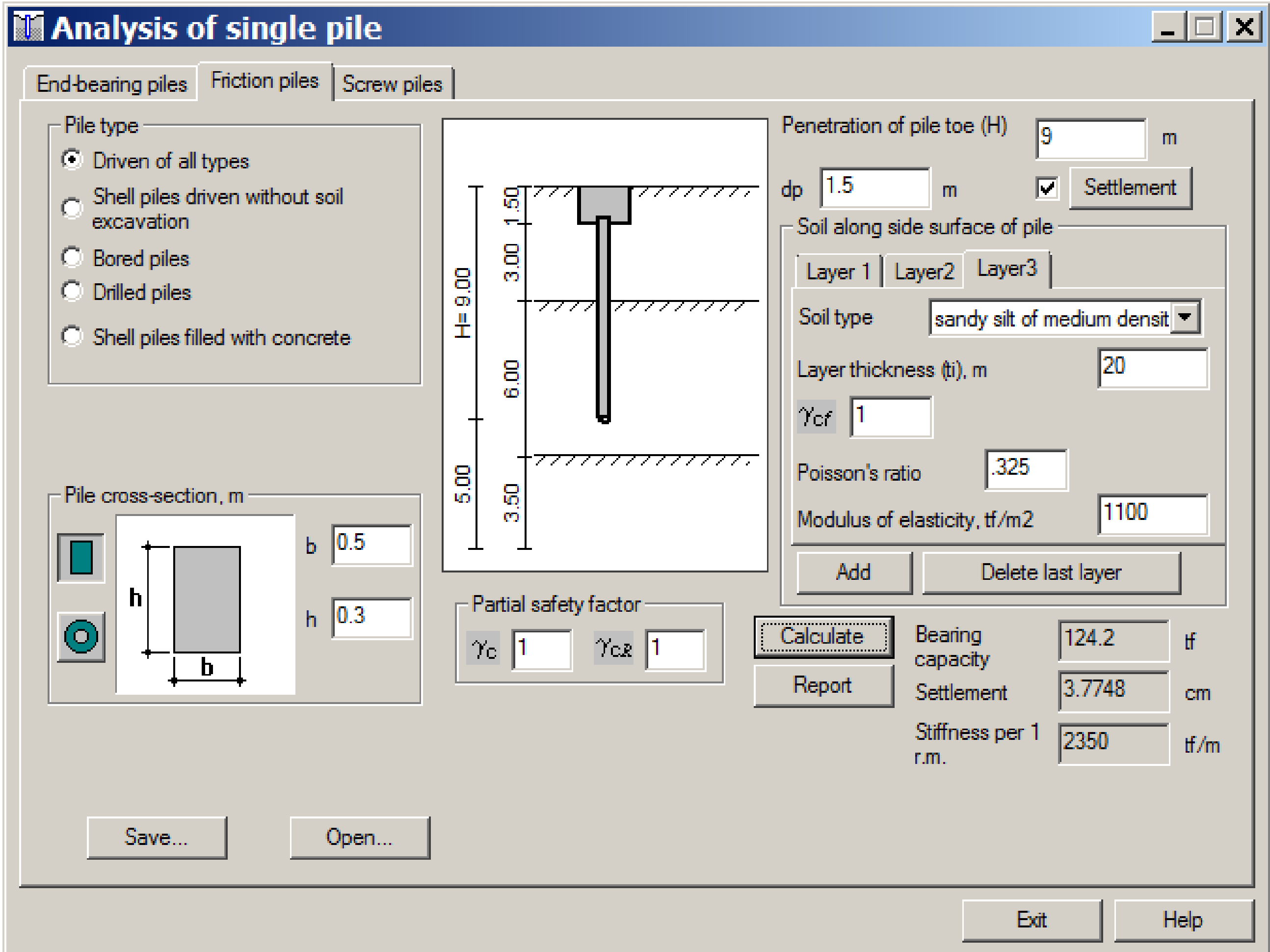

Analysis of single pile
End-bearing piles
Friction piles
Screw piles
Pile type
Driven of all types
Shell piles driven without soil excavation
Bored piles
Drilled piles
Shell piles filled with concrete
H= 9.00
1.50
3.00
6.00
5.00
3.50
Penetration of pile toe (H)
9
m
dp
1.5
m
Settlement
Soil along side surface of pile
Layer 1
Layer2
Layer3
Soil type
sandy silt of medium densit
Layer thickness (ti), m
20
γcf
1
Poisson's ratio
.325
Modulus of elasticity, tf/m2
1100
Add
Delete last layer
Pile cross-section, m
h
b
b
0.5
h
0.3
Partial safety factor
γc
1
γcR
1
Calculate
Report
Bearing capacity
124.2
tf
Settlement
3.7748
cm
Stiffness per 1 r.m.
2350
tf/m
Save...
Open...
Exit
Help

Figure 8.4 b)

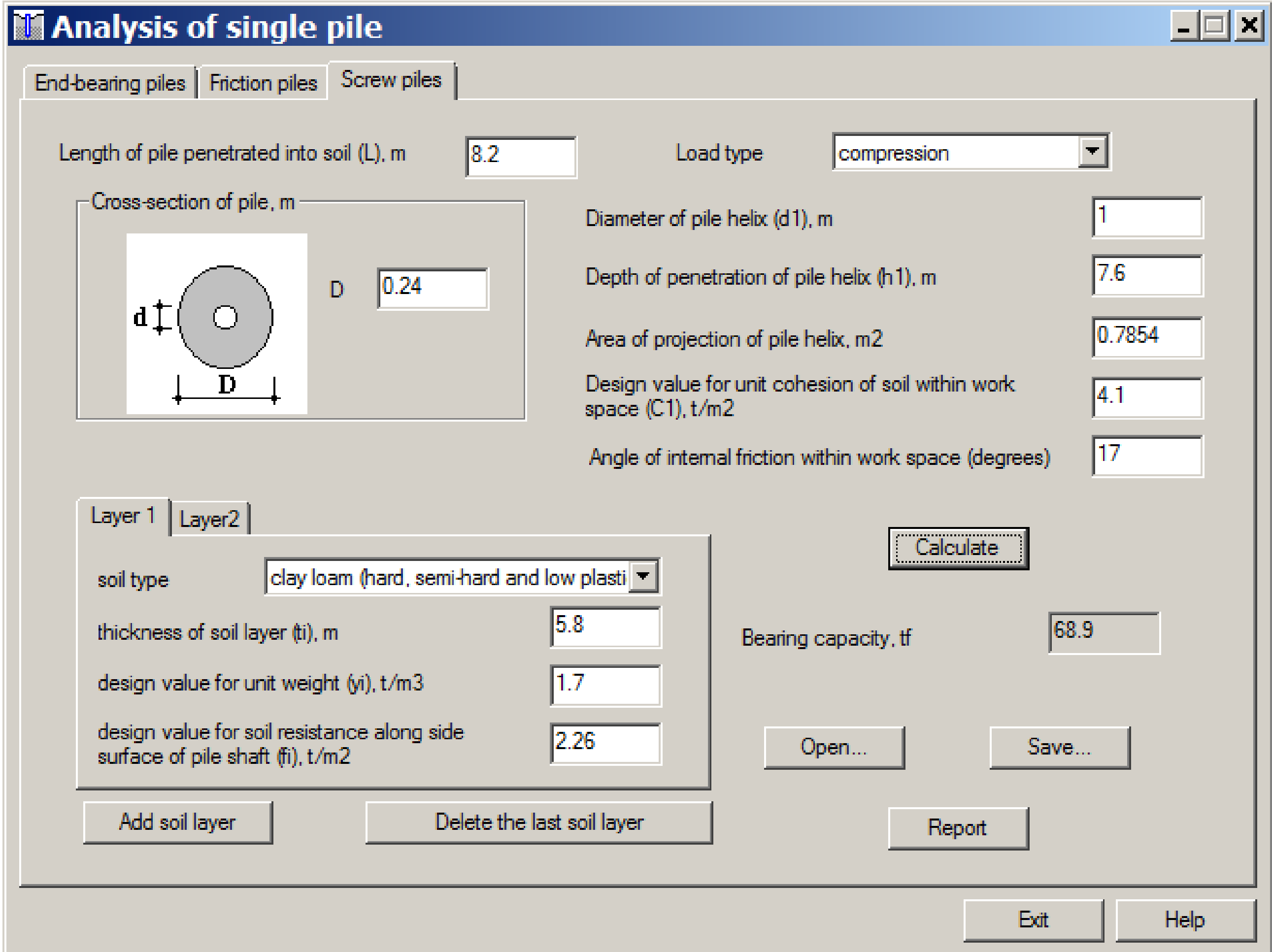


Figure 8.4 c)

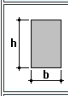
Output data

Output data is presented in the **Bearing capacity of pile** box.
For friction piles and screw piles, the output data may be presented as a report in HTML file.
For friction piles, the following data is displayed in the report file: bearing capacity of pile, its settlement, including the one with account of interaction of piles in the pile group, stiffness of pile per 1 running metre.
For screw piles, the following data is displayed in the report file: bearing capacity of pile helix, pile shaft and the pile itself.

The output data for the friction pile is saved to the file with extension **.pile* .
The output data for the screw pile is saved to the file with extension **.vpl* .

Bearing capacity of friction pile

Input data

Driven pile		
	Pile cross-section: b=0.5 m, h=0.3 m	
γ_c	Partial safety factor for pile	1
γ_{cR}	Partial safety factor for soil under pile toe	1
H	Penetration of pile toe, m	9
dp	Height of grillage, m	1.5

Soil along side surface of pile		
Layer 1	gravel sand of medium density	
ti	Layer thickness, m	3
γ_{cf}	Partial safety factor for soil	1
v	Poisson's ratio	0.4
E	Modulus of elasticity, tf/m²	1000
Layer 2	medium sand of medium density	
ti	Layer thickness, m	6
γ_{cf}	Partial safety factor for soil	1
v	Poisson's ratio	0.4
E	Modulus of elasticity, tf/m²	800
Layer 3	sandy silt of medium density	
ti	Layer thickness, m	20
γ_{cf}	Partial safety factor for soil	1
v	Poisson's ratio	0.325
E	Modulus of elasticity, tf/m²	1100

Output data

Fd	Bearing capacity of pile, tf	124.19
U	Perimeter of pile cross-section, m	1.6
A	Area of pile cross-section, m²	0.15
R	Design strength of soil under pile toe, tf/m²	390
Design strength of soil along side surface of pile		
hi - thickness of the i-th layer of soil, m		

Figure 8.5

Pile under combined action of loads

The module enables you to compute single pile in strain and in stability from combined action of vertical and horizontal loads and moment.

The following building codes are supported:
- appendix 1 SNIP 2.02.03-85*;
- appendix B SP 24.13330.2011 with account of modification No.1.

Analysis by SNIP 2.02.03-85* includes:
- Analysis of bearing capacity of piles in case the 2nd stage of the stress-strain state of soil is developing H ≤ Fd / γk;
- Check for soil stability according to sect.13 only for the 1st stage of stress-strain state of soil;
- Analysis of piles according to serviceability limit state with the following check Up≤Uu; ψp≤ψu;
- Analysis of pile section according to ultimate and serviceability limit states (sect. 8-9 assuming that the 2nd stage of the stress-strain state of soil is developing). In other cases, according to sect. 14,15.

It is supposed that in loading process there are two stages of stress strain state for the 'pile-soil' system.

At the first stage the soil enveloping the pile behaves as linearly elastic medium.
Elastic properties of soil are described with modulus of subgrade reaction that is increased linearly with the depth.

At the second stage, in the upper part of soil enveloping the pile there is an area of limit equilibrium (plastic zone). Stiffness of soil within the limits of limit equilibrium area is described with strength factor of proportionality. Below, the soil behaves as in the first stage (linearly elastic medium).

The ultimate state of the 'pile-soil' system is taken to become at the moment when the plastic hinge is generated within the limits and on the edge of limit equilibrium area for soil.

Horizontal displacement and angle of rotation of the pile head are determined after calculation. In case of one-stage calculation, stability analysis of soil is carried out according to sect. 13 Appendix 1 SNIP 2.02.03-85. If the account of the second stage of stress strain state of soil is defined, the program computes bearing capacity of pile according to condition $H \le Fd / \gamma k$, where

- H – design value of transverse load applied to pile;
- Fd – bearing capacity of pile, determined according to requirements of sect. 10;
- γk – partial factor for an action, taken as equal to 1.4 .

Analysis by SP 24.13330.2011 includes:

- Analysis of bearing capacity of piles in case the 2nd stage of the stress-strain state of soil is developing $H \le Fd / \gamma k$;
- Check for soil stability according to sect. B.7 ;
- Analysis of piles according to serviceability limit state with the following check $Up \le Uu$; $\psi p \le \psi u$;
- Analysis of pile section according to ultimate and serviceability limit states.

Input data may be saved to **.mnh* file. It is also possible to download input data from the previously saved file. To do this, click **Save** or **Open** on the System menu of the program. To access the System menu, either left-click on the upper-left icon of the dialog box or press the ALT and SPACE keys. The name of the current file is displayed in the dialog box.

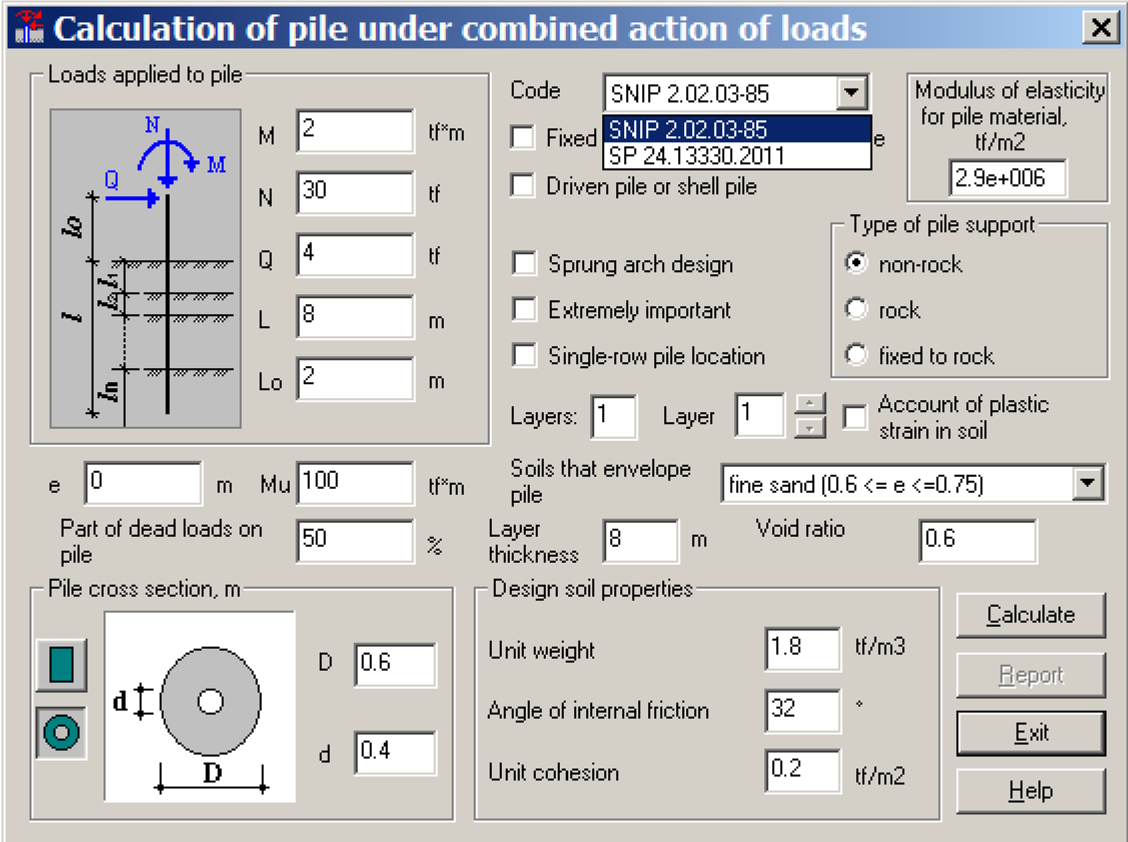


Figure 8.6

Input data
In the **Code** list, select the building code: either SNIP 2.02.03-85 or SP 24.13330.2011.
In the **Loads applied to pile** area, define the moment, vertical and horizontal loads, length of pile.
Select the type of pile cross-section and define its dimensions.
In the right part of the dialog box, input **Modulus of elasticity for pile material**, define **Type of pile support**, number of soil layers and for every layer define **Soils that envelope pile**, **Design properties of soil** and other parameters.
To take account of design situation, select either **Fixed connection of pile and grillage** or **Driven pile or shell pile** option.
If the **Account of plastic strain in soil** check box is selected, analysis is carried out with account of the second stage of stress-strain state of the soil.
If the **Account of plastic strain in soil** check box is not selected, then analysis of pile according to serviceability limit state and stability analysis of soil are carried out. In this case, in analysis procedure, the following message may appear: *'Stability of soil that envelopes the pile is NOT adequate. Perform iterations by reducing K?'*. If you click **Yes**, analysis will be continued but with reduced values of proportionality factor **K**. If you click **No**, analysis is complete.
Click **Calculate**.

Output data
Output data is presented as report file in HTML format. This file appears on the screen after calculation or when you click **Report**.

Output data

1. Calculation of pile by deformations

I	Moment of inertia for pile cross section, m^4	0.005105
bc	Conventional width of pile, m	1.40
K	Factor of proportionality, (tf/m4)	1800.00
a	Strength factor of proportionality, (tf/m3)	7.10
α_e	Strain coefficient, 1/m	0.701778
l_	Reduced value of pile penetration	5.614226
Qo	Design value of shear force in pile section, tf	4.0000
Mo	Design value of bending moment in pile section, tf*m	10.0000
Up	**Design value of horizontal displacement of pile head, mm**	10.271343
Ψp	**Design value of rotation angle of pile head, rad*1000**	3.385170

2. Check for bearing capacity of pile

	Part of dead loads in total horizontal load, %	50.00
Mu	Ultimate bending moment with account of normal force, tf*m	100.000000
e	Load eccentricity, m	0.000000
Qo	Design value of shear force in pile section, tf	4.0000
Zz	Distance from soil surface to plastic hinge, m	3.1135
Fd	**Bearing capacity of pile, tf**	27.530123
γ_k	Partial factor for an action	1.40

Condition $H \le F_d / \gamma_k$ is satisfied (H=Qo).

Figure 8.7

Settlement of equivalent footing

The module enables you to compute the settlement of pile group as equivalent footing according to SNIP 2.02.03-85 sect.6 and SP 24.13330.2011 with account of SP 22.13330.2011.
The input and output data is presented in Fig. 8.8.

Input data
– Select the shape of pile section.
– Define **Section dimensions**, **Pile penetration to soil**, **Number of piles**, **Contour dimensions of pile group**, parameters for grillage.
– Define **Vertical load on foundation**, **Number of layers in soil**.
– Define **Soil properties**.
– Click **Calculate**.

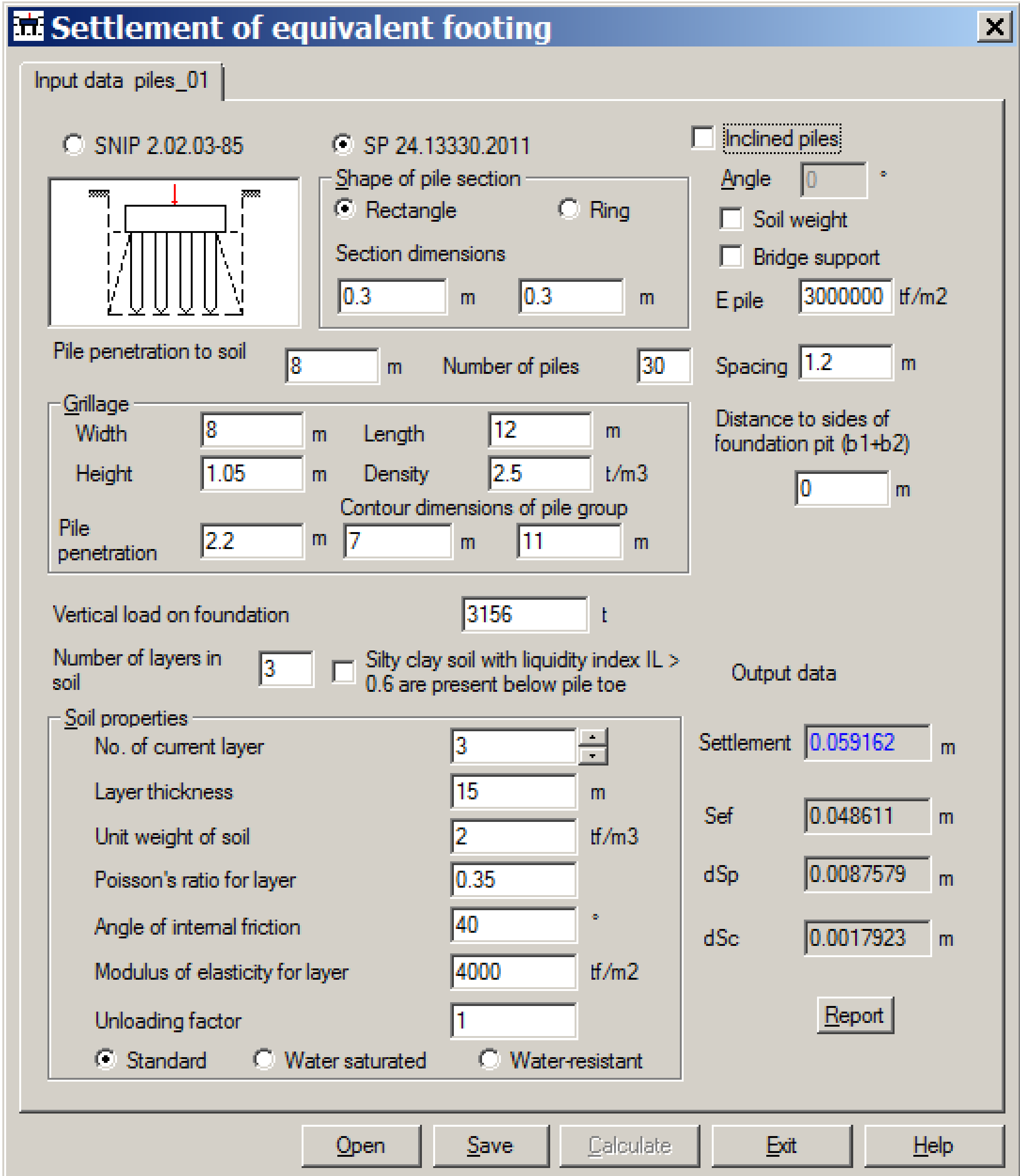


Figure 8.8

Output data

Output data (dimensions of equivalent footing, its dead weight, depth of compressible stratum and settlement) is presented as report file. To present a report document in HTML format, click **Report**.

Soil properties							
Layer	Layer thickness, m	Unit weight of soil, tf/m^3	Angle of internal friction, °	Modulus of elasticity, tf/m^2	Poisson's ratio for layer	Coefficient for unloading	Notes
No.	L_I	γ_I	φ_I	E_I	μ_I	$K_{E,I}$	
1	4.00	1.80	14.00	4000.00	0.35	1.00	
2	4.50	1.90	24.00	4000.00	0.35	1.00	
3	15.00	2.00	40.00	4000.00	0.35	1.00	

Vertical load on foundation, tf
300.00

Output data

Weight of grillage, tf	Weight of piles, tf	Weight of soil in volume of equivalent footing, tf
360.00	54.00	1478.70

Actual settlement, mm	Punching shear settlement, mm	Compression settlement, mm
6.09	0.84	0.17

Dead weight of equivalent footing, tf	Dimensions for base of equivalent footing, m		Depth of compressible stratum, m	Settlement, mm
414.00	8.20	12.20	4.96	7.10

Vertical stresses from dead weight of soil at the base of equivalent footing are not considered in additional vertical pressure at the level of the base.

Figure 8.9

Principal and equivalent stresses in soil

The module enables you to calculate principal and equivalent stresses σ_1, σ_2, σ_3 in soil by the specified stress tensor $\{\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{xz}, \tau_{yz}\}$.

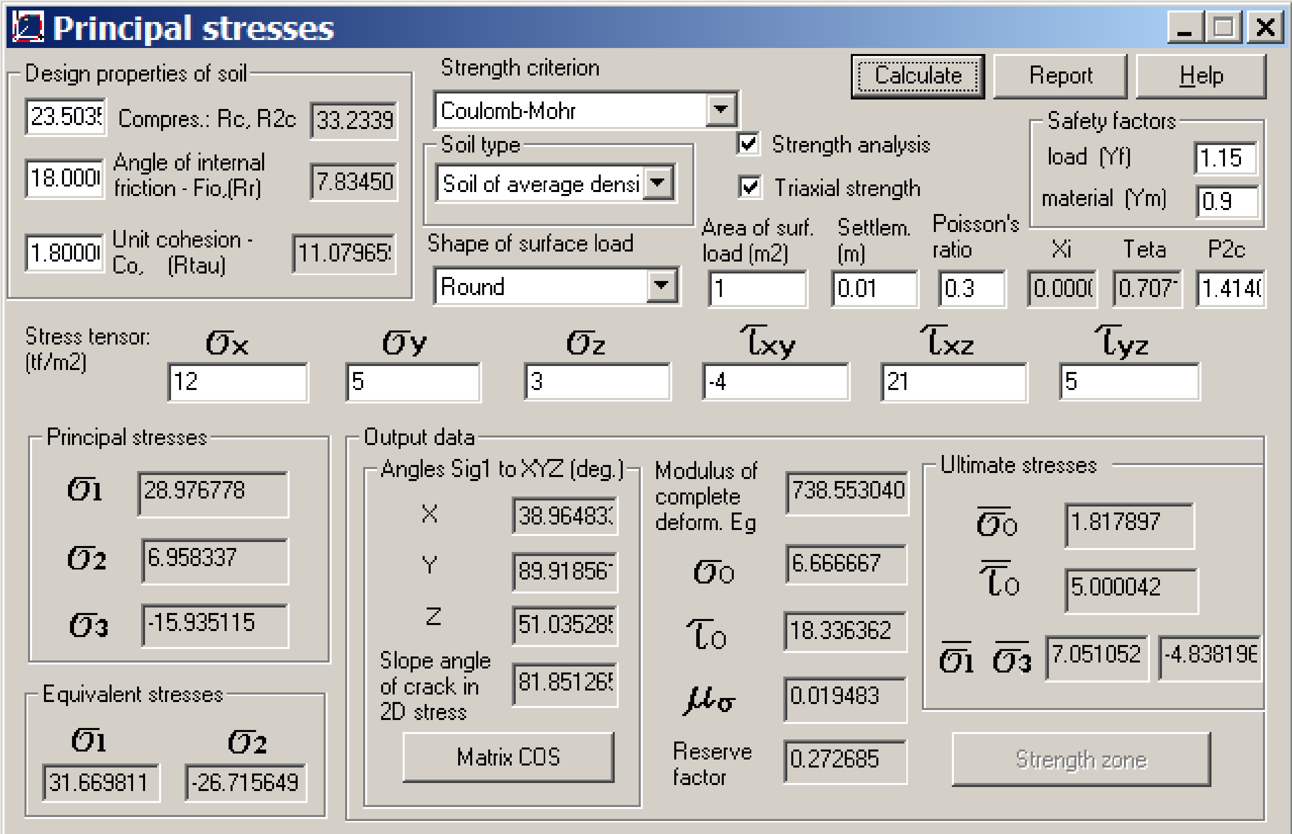


Figure 8.10

Input data

In the appropriate list, define the soil type: soil of average density, dense soil or rock.
In the **Design properties of soil** area, define the following values:

– strength in compression: **Rc**, if **Rc=0** – the value is determined automatically;
– angle of internal friction: - **Fi** (in degrees);
– cohesion – **Co** (tf/m^2) or strength in pure shear (negative value – **Rtau** for soils where cohesion is close to zero). For soils where cohesion is close to zero, it is possible to define strength of soil in compression. Then required strength in pure shear is taken as **Rtau = 0.5*Rc / 1.732**.

To determine modulus of complete deformations of soil Eg according to SNIP 2.02.01–83 and DBN V.1.1–5–2000, define the following data:
Shape of surface load (rectangular or round);
Area of surface load (m^2);
Design value of settlement (m);
Poisson's ratio.
Click **Calculate**.

Modulus of complete deformations is determined by field test results with surface load according to the model of linear elastic half-space (LHS) and it is valid only for deformed zone below the surface load.

The following limit equilibrium equations are realized for soil: Coulomb-Mohr criterion and Mises-Schleicher-Botkin criterion (modified Coulomb-Mohr criterion).

Output data

When the **Strength analysis** check box is selected, after strength analysis, the following data is displayed:

σ_1, σ_3 – ultimate allowed values of principal stresses;
σ_0, τ_0 – ultimate axial and shear octahedral stresses;
– reserve factor (analysis by serviceability limit state is supposed, that is, normative strength parameters of material are considered in analysis);
– Lode-Nadai parameter is computed as: $\mu_\sigma = \frac{(2\sigma_2 - \sigma_1 - \sigma_3)}{(\sigma_1 - \sigma_3)}$;

To generate report file in HTML format, click **Report**. The report file contains the following data:
– principal stresses as: σ1 > σ2 > σ3;
– matrix of directional cosines MC(3,3) – orientation of principal stresses in the space relative to the X,Y,Z axes:

$$MC = \begin{bmatrix} l1 & m1 & n1 \\ l2 & m2 & n2 \\ l3 & m3 & n3 \end{bmatrix}$$

– slope angles AL1, AL2, AL3 (l1, l2, l3) of principal stress σ_1 to the X, Y, Z-axes; angles define orientation of the crack plane in the space;
– slope angle α3 of principal stress σ_3 to the X-axis; angle defines orientation of the crack plane relative to the X-axis ;
– Euler angles TETA, PSI, FI (θ, ψ, φ) that define location of principal stresses σ_1, σ_2, σ_3 relative to the X, Y, Z-axes of global coordinate system;
– additional strength parameters:
  – strength in biaxial uniform compression – R2c = Rc*P2c,
  – strength in biaxial uniform tension – R2p.
– whether cracks are present TB=1, 2, 3 – one, two, three; TB=10 (20) – destruction of element in compression. TB=0 – no cracks.

Stress tensor		
Sig_X = 12.000	Sig_Y = 5.000	Sig_Z = 3.000
Tau_XY= -4.000	Tau_XZ= 21.000	Tau_YZ= 5.000
Principal stress		
SIG_1= 28.977	SIG_2= 6.958	SIG_3= -15.935
Equivalent stress by theory	Max principal deformations	
SIG_1E= 31.670	SIG_2E= -26.716	
Octahedral stress - Normal	Sig_O = 6.667	
Octahedral stress - Shear	Tau_O = 18.336	
Lode-Nadai coefficient	Param_Lode = 0.0195	
Slope angle of crack	Alfa_crack_2x= 81.851	(Degrees)
Slope angle Sig_1	to X, Y, Z-axes	of principal coordinate system
AL1= 0.680 (Radians)	AL2= 1.569	AL3= 0.891
AL1= 38.965 (Degrees)	AL2= 89.919	AL3= 51.035
Matrix of direction cosines	Orientation SIG_1,2,3 in triaxial stress strain state	MC(3,3)
L1= 0.77753	m1= 0.18498	n1= 0.00000
L2= 0.00142	m2=-0.95627	n2= 0.00000
L3= 0.62884	m3=-0.22656	n3= 1.00000
Soil properties	Soil of average density	
Angle of internal friction	Fi =18.000	(deg.)
Unit cohesion	Co =1.800	(tf/m2)
Poisson's ratio	Nu_g =0.300	
Safety factor by load	kNad_P=1.150	
Safety factor by strength	k_R1C =0.900	
Conversion factor to biaxial strength	P2C =1.414	
Normative strength in compression	Rc_n=23.504	(tf/m2)
Normative strength in tension	Rr_n=7.835	(tf/m2)
Strength in pure shear	R_tau=11.080	(tf/m2)
Strength in biaxial compression	R2c=33.234	(tf/m2)
Strength in biaxial tension	R2r=5.540	(tf/m2)
Design compression strength	Rgr_c =21.153	(tf/m2)
Design tensile strength	Rgr_r =7.051	(tf/m2)
Modulus of total deformations	Egr =738.553	(tf/m2)
Shape of surface load	Rectangular	
Area of surface load	F =1.000	(m2)
Effective settlement	S=0.01000	(m)
Principal strain, elastic	Def_1 =0.03411698	Def_3 =0.03411698
Principal strain, inelastic	Def_1n=0.03923453	Def_3n=-0.02157613

Figure 8.11

Slope stability

These modules enable you to check stability of slope by:
- 1st type: Plane slip surface of homogeneous soil (see Fig.8.12 a);
- 2nd type: Cylindrical slip surface of homogeneous soil (see Fig.8.12 b).

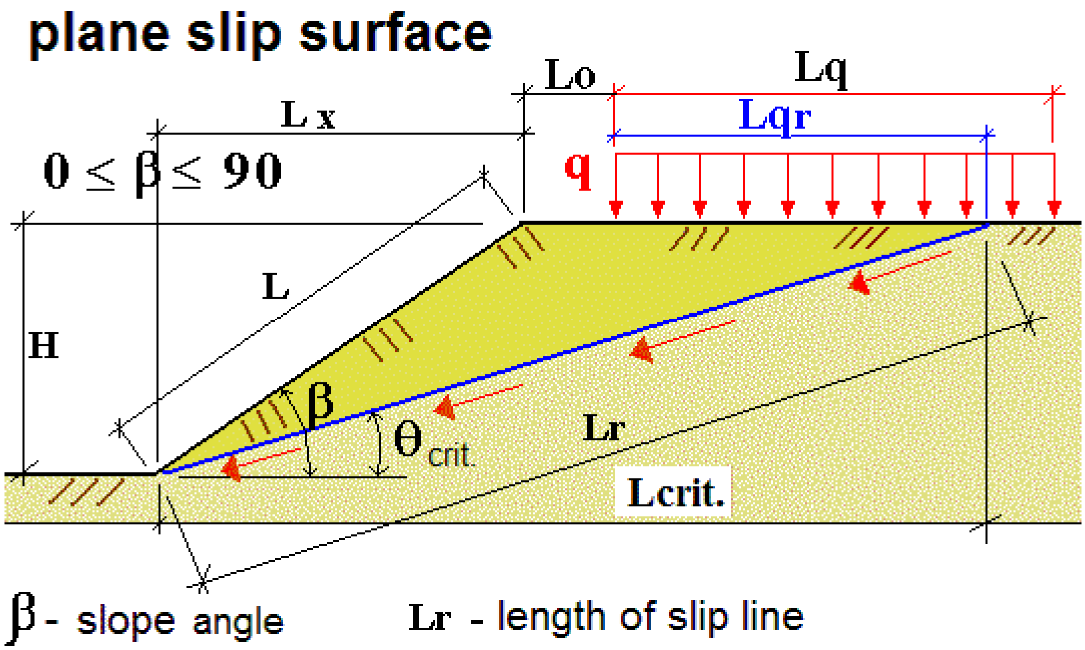


Figure 8.12 a.

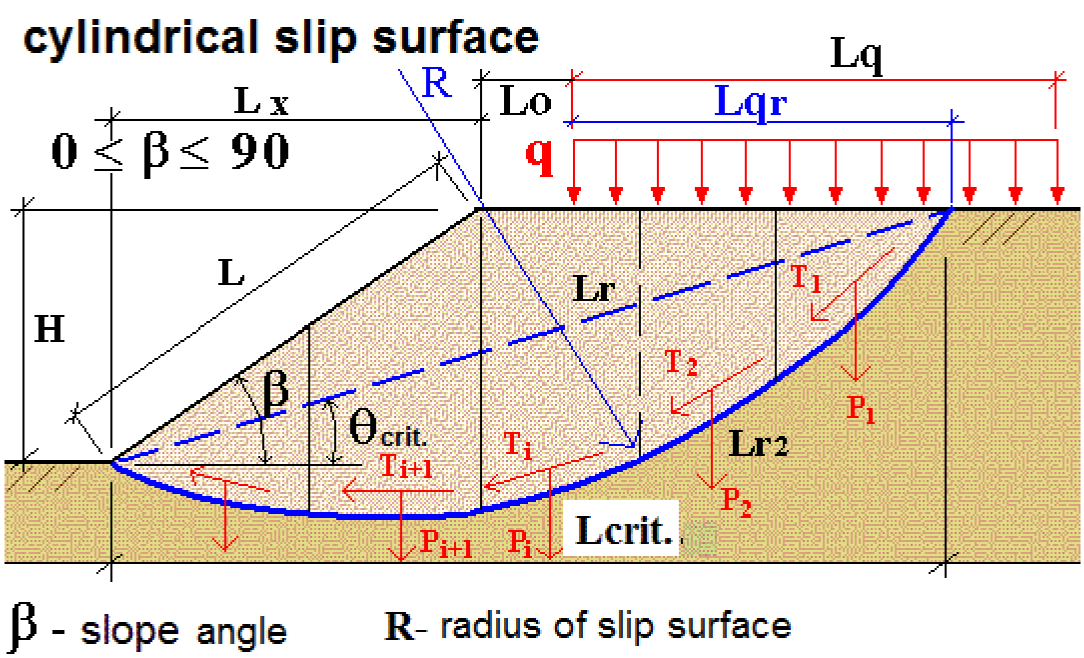


Figure 8.12 b.

For types 1 and 2, simplified integration by the 1st point is realized. Stability factor of slope for the 1st type (**Kstab**) is computed taking into account that the length of slip surface (**Lr2**) is increased relative to the plane slip surface (**Lr**).

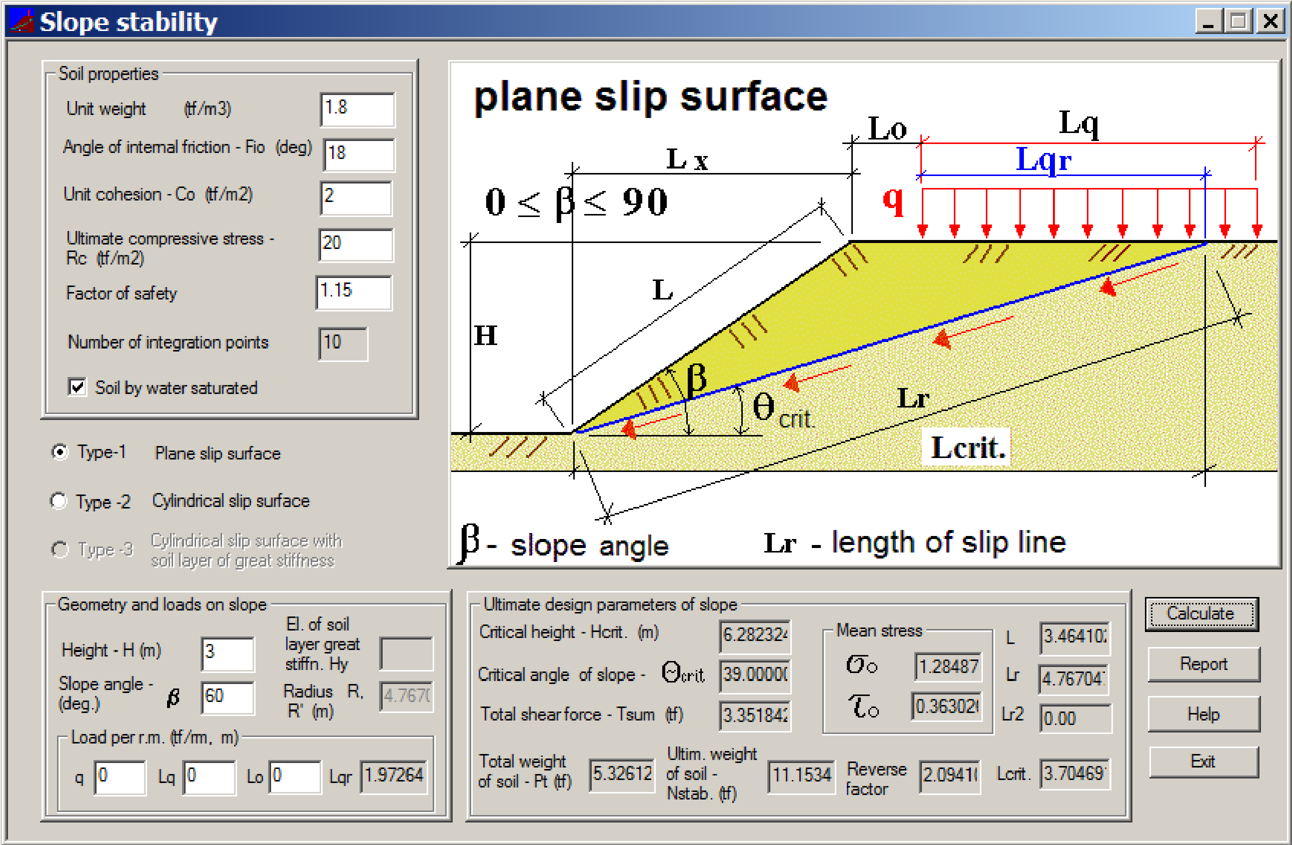


Figure 8.13

Input data

In the **Slope stability** dialog box, define parameters of slope: height, slope angle and load per running metre.
In the **Soil properties** area, define angle of internal friction, unit cohesion, unit weight, ultimate compressive stress, safety factor, number of integration points.
With option button, select the type of slip surface.
Click **Calculate**.

Output data

When calculation is complete, you will obtain the following values:
– critical height of slope – **Hcrit**;
– critical angle of slip surface – **θcrit** ;
– total weight of soil above slip surface – **Psum**;
– total shear force from weight of soil along slip surface – **Tsum**;
– ultimate weight of soil – **Nstab**;
– length of slip surface (plane) – **Lr** by Type 1;
– length of cylindrical slip surface – **Lr2** by Type 2;
– critical distance from the slope base up to the upper safe (relatively safe) point – **Lcrit**.
– stability factor for slope – **Kstab**;
– mean normal (**Sig**) and shear (**Tau**) stresses at slip area.

Date: 4 11 2016	Module << Stability of slope from homogeneous soil >>	Ver.:4.08.09
INPUT DATA		
Slope height	H	3.000 (m)
Slope angle	Beta	60.000 (deg.)
Radius of slip surface	R	3.000 (m)
Length of plane slip surface	Lr1	4.767 (m)
Length of cylindrical slip surface	Lr2	4.992 (m)
Design loads on slope		
Distributed load	Q	0.000 (tf/rm)
Distance to distributed load	L0	0.000 (m)
Length of distributed load	Lq	0.000 (m)
Effective length of distributed load	Lqr	1.973 (m)
Soil properties		
Angle of internal friction	Fio	18.000 (deg.)
Unit cohesion	Co	2.000 (tf/m2)
Unit weight of soil	Gama	1.800 (tf/m3)
Ultimate compressive stress	Rc	20.000 (tf/m2)
Safety factor	Kn	1.150
Water saturated soil	Yes	
OUTPUT DATA		
Calculation by plane slip surface		
Total weight of soil above slip surface	Psum.	5.326 (tf)
Max shear force along slide surface	Tsum.	3.352 (tf)
Ultimate weight of soil	Nstab.	11.153 (tf)
Length of slope	L	3.464 (m)
Length of plane slip surface	Lr	4.767 (m)
Length from foot up to top of slope	Lx	4.992 (m)
Critical length from foot of slope	Lcr.	3.705 (m)
Critical height of slope	Hcr.	6.282 (m)
Critical angle of slip surface	Ocr.	39.000 (deg)
Stability factor	Kstab	2.094

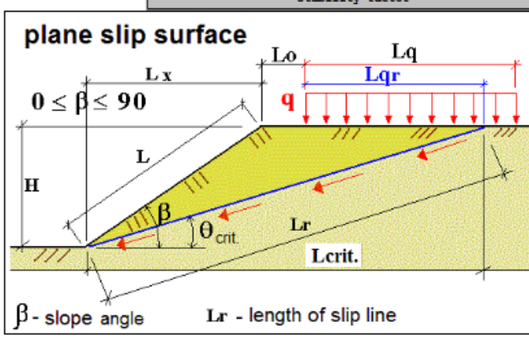


Figure 8.14

The following limit equilibrium equations are realized for soil: Coulomb-Mohr criterion and Mises-Schleicher-Botkin criterion.
To present the output data as a report file in HTML-format, click **Report**.

Stability of multi-layer slope

These modules enable you to check stability of slope by cylindrical slip surface with multi-layer slope. Calculation is made by Swedish method (method by Swedish Society for Geomechanics). This method is experimental. Up to 10 base layers are allowed.

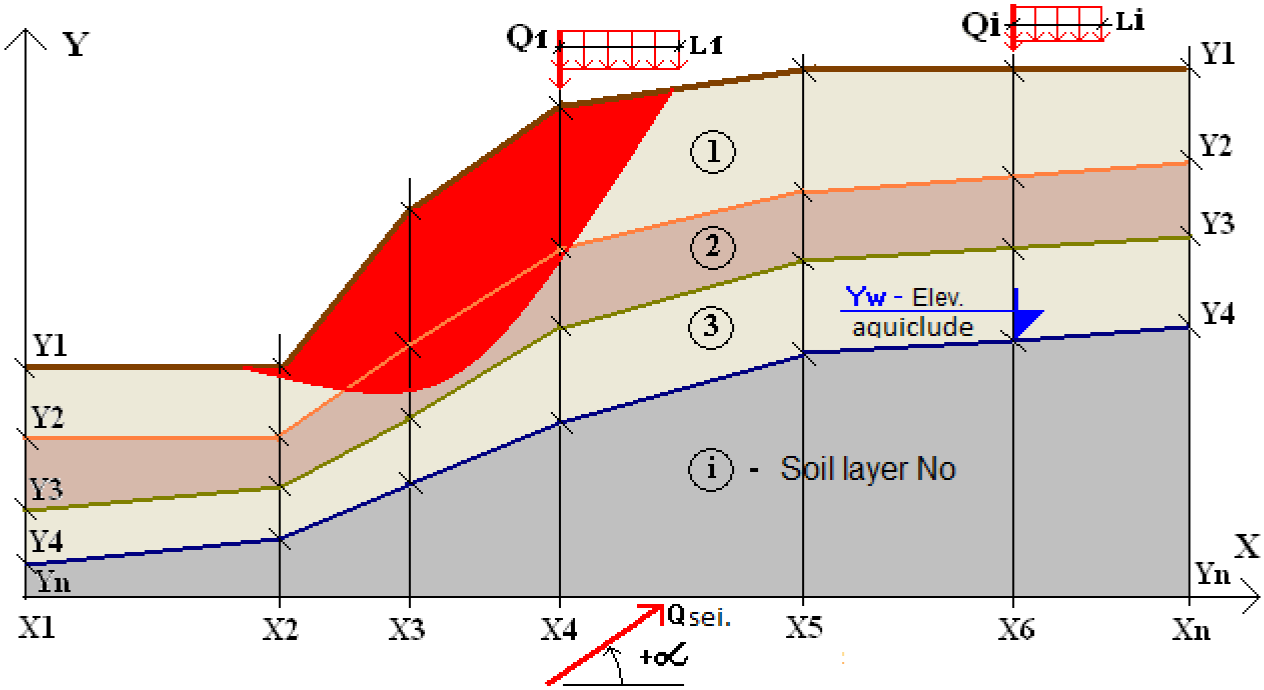


Figure 8.15

Coordinates of landslide Xi and Yi as well as ordinates of the diagram of landslide pressure Pi are determined after calculation.

Safety factor – 1.150
Reserve factor for static load – 0.914
Reserve factor for dynamic load – 0.000
Total active normal force (tf) – -49.287
Active component of shear forces (tf) – 5.099
Reactive component from cohesion (tf) – 0.000
Radius of cylindrical sliding surface (m) – 60.675

It is possible to evaluate graphically geometry of multi-layer slope and the slip surface of multi-layer slope.

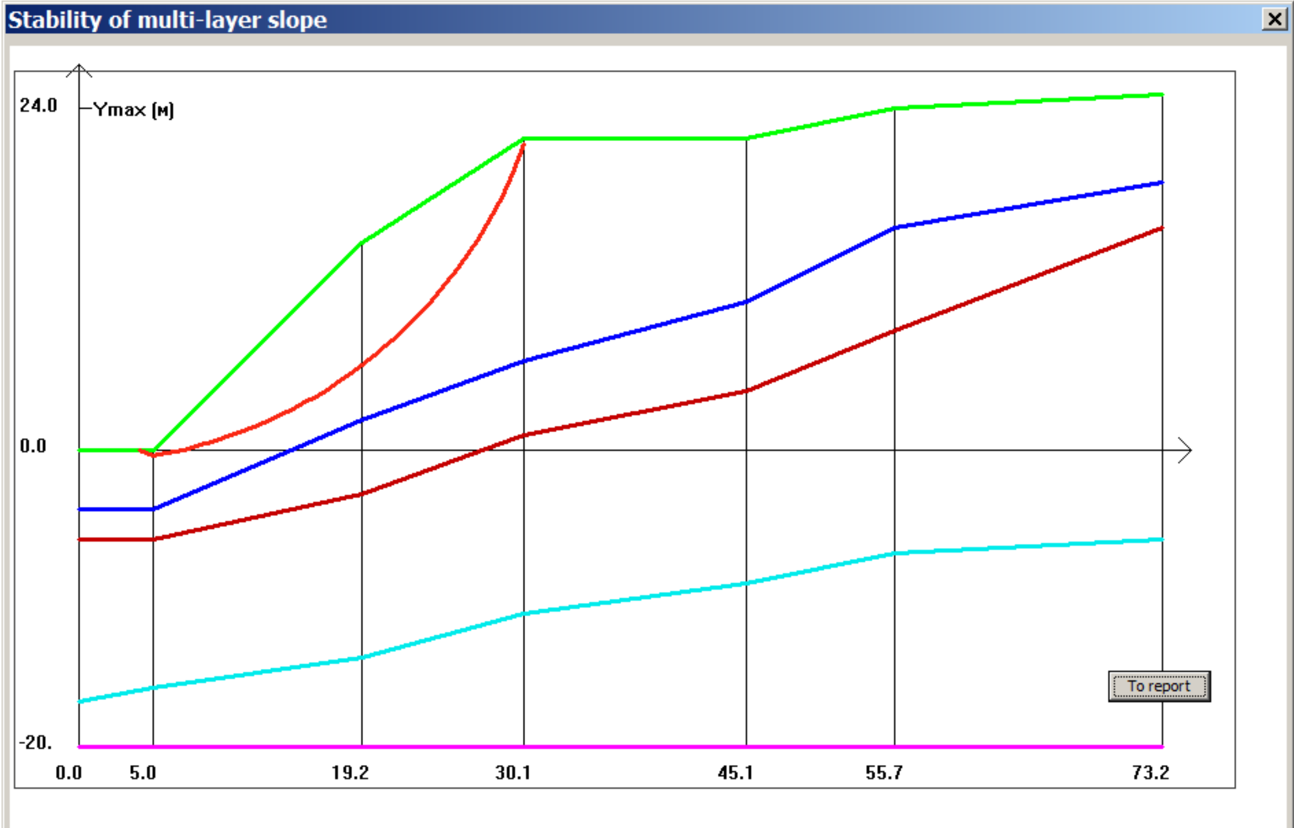


Figure 8.16

To present the output data as a report file in HTML-format, click **Report**.

Basic assumptions for calculation

1 r.m. of the slope width is taken into consideration. The weight of the whole soil with account of additional active load above the damage surface $\mathbf{P_{sum}}$ is determined:

$$P_{sum} = \frac{0.5 \cdot H^2 \cdot \sin(\beta - \theta)}{(\sin \beta \cdot \sin \theta)} + q \cdot Lqr$$

Critical angle of slip surface – $\theta_{crit} = \dfrac{(\beta + f)}{2}$.

Ultimate weight of soil according to [3] :

$$N_{stab} = \frac{(g \cdot H_{crit})}{C_o \cdot K_n}$$

where **g** – design value for unit weight of soil (tf/m^3);

K_n – safety factor (by default, K_n =1.15).

Stability factor K_{stab} according to [3]:

$$K_{stab} = K_{stab1} = \frac{\left(P_{sum} \cdot \cos\theta \cdot tg\, f + C_o \cdot Lr\right)}{\left(P_{sum} \cdot \sin\theta\right)}$$

To find out arbitrary slip surface for arbitrary distributed load at the whole part of multi-layer slope, the so-called Swedish method (method by Swedish Society for Geomechanics) is applied in the program in the way presented in Fig. 8.17.

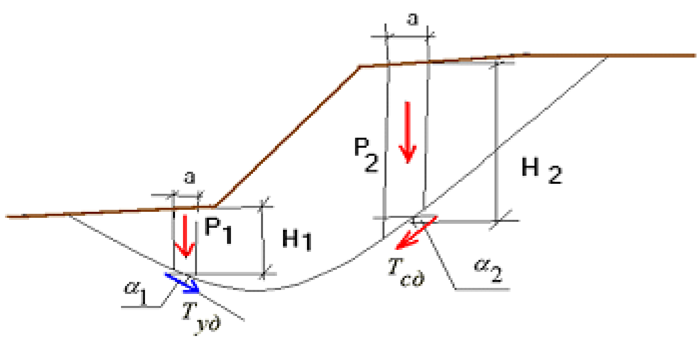


Figure 8.17

Shear force is computed by formula:

$$T_{shear} = P_2 \cdot \sin(\alpha_2) = a \cdot H_2 \cdot 1m \cdot \gamma_{shear} \cdot \sin(\alpha_2) = a \cdot H_2 \cdot 1m \cdot \frac{\gamma_n}{1-|\rho|} \cdot \sin(\alpha_2)$$

Restraining force is computed by formula:

$$T_{restr} = P_1 \cdot \sin(\alpha_1) = a \cdot H_1 \cdot 1m \cdot \gamma_{restr} \cdot \sin(\alpha_1) = a \cdot H_1 \cdot 1m \cdot \frac{\gamma_n}{1+|\rho|} \cdot \sin(\alpha_1)$$

where

a – min area of integration (it is agreed that there are 30 fixed points of integration between adjacent boreholes);

γ_n – normative value for unit weight of soil;

$\frac{1}{1 \pm |\rho|}$ – safety factor for soil, different for restraining and shear forces.

Bearing capacity of piles by field test results

The module enables you to check bearing capacity of piles by field test results.
The module supports the following building codes:
– section 7 SP 24.13330.2011,
– section 7 SP 50-102-2003,
– section 5 SNIP 2.02.03-85*,
– sections 5, 6 DSTU B V.2.1-27:2010.

The dialog box contains 6 tabs:
1. Driven piles at probe point;
2. Screw piles for static probing;
3. Drilled piles at probe point;
4. Driven piles at test point of model pile;
5. Driven piles at test point of probe pile;
6. Dynamic pile tests.

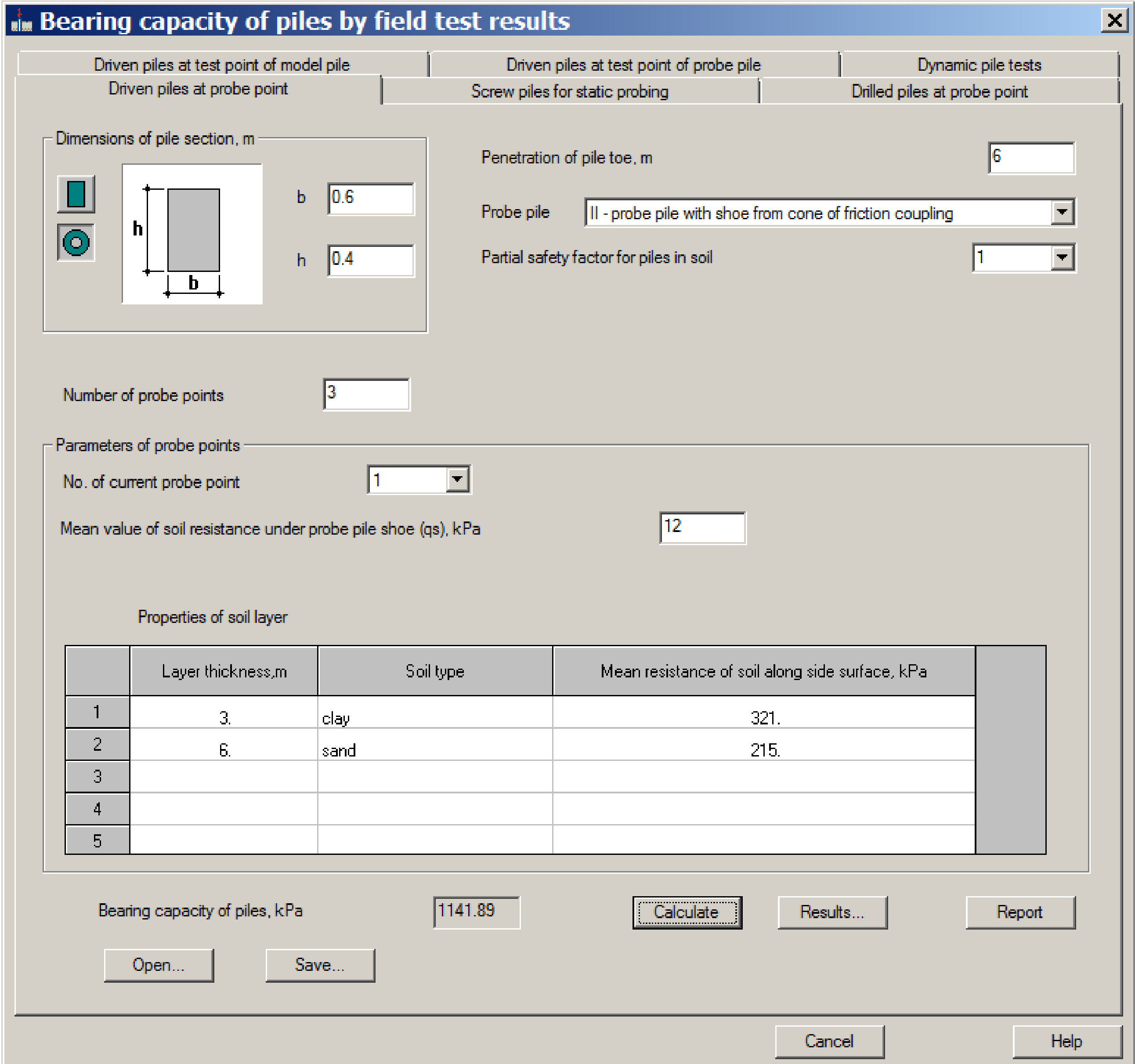


Figure 8.18

Combined piled-raft foundation

The module enables you to calculate settlement for combined piled-raft (CPR) foundation according to MGSN 2.07-01 Appendix **i**.

The dialog box of the program is presented in the figure below.

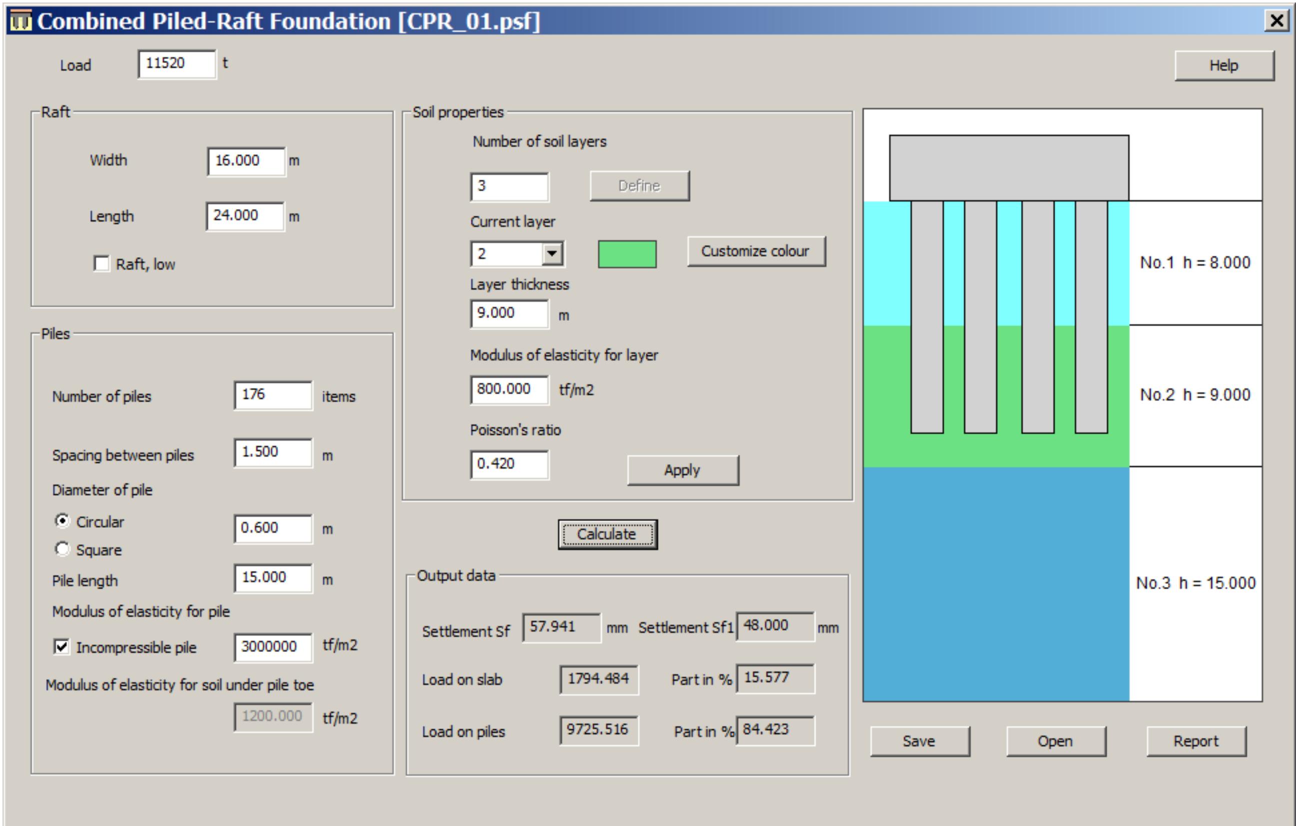


Figure 8.20

Input data

In the appropriate box, defile load on foundation and dimensions

Dimensions for raft are defined in metres. If raft is low, select appropriate check box.

The following data is defined for piles:
– number of piles;
– distance (spacing) between piles, m;
– diameter for circular pile or side for rectangular pile, m; (to define the shape for the pile, just click appropriate option button);
– length of pile, m;
– modulus of elasticity for pile, t/m2;

If the pile is incompressible, select appropriate check box.

To define parameters for soil layers, indicate the number of soil layers and click **Define**. Then you will be able to define the number of current layer and its properties:
– thickness, m;
– modulus of elasticity, t/m2;
– Poisson's ratio.

To modify the colour for the current soil layer, click **Customize colour**.

When you define soil properties and click Apply, the program automatically displays the value for modulus of elasticity for soil under pile toe.

The problem may be saved to the **.psf* file.

To start calculation, click **Calculate**.

Calculation

Intermediate data computed in calculation:
– Coefficient for pile settlement **Is**;
– Coefficient **Lambda**;
– Stiffness for single pile, t/m, **K1**;
– Coefficient **Rs**;
– Stiffness for pile group, t/m, **Kp**;
– Mean modulus of elasticity for soil, tf/m^2, **Es**;
– Mean value of Poisson's ratio for soil **Vs**;
– Ratio of slab length to slab width **Lp/B**;
– Coefficient for area **m0**;
– Stiffness for slab, t/m, **Kc**;
– Total stiffness of foundation, t/m, **Kf**;
– Mean modulus of elasticity for soil under pile toe, tf/m^2, **Esb**.

Output data

The following data is displayed in the dialog box:
– settlement **Sf**, m, determined by dividing the load by total stiffness of combined piled-raft (CPR) foundation, m;
– settlement **Sf1**, m, determined according to mean modulus of elasticity for soil under pile toe, m;
– load on slab, t, and its part from the total load in %;
– load on piles, t, and its part from the total load in %;

To present a report document in HTML format, click **Report**. The report file contains all intermediate and final results with necessary comments.

Loads and actions

Loads and actions

The chapter contains modules that enable you to compute loads according to SNIP 2.01.07– 85 and DBN V.1.2-2:2006:

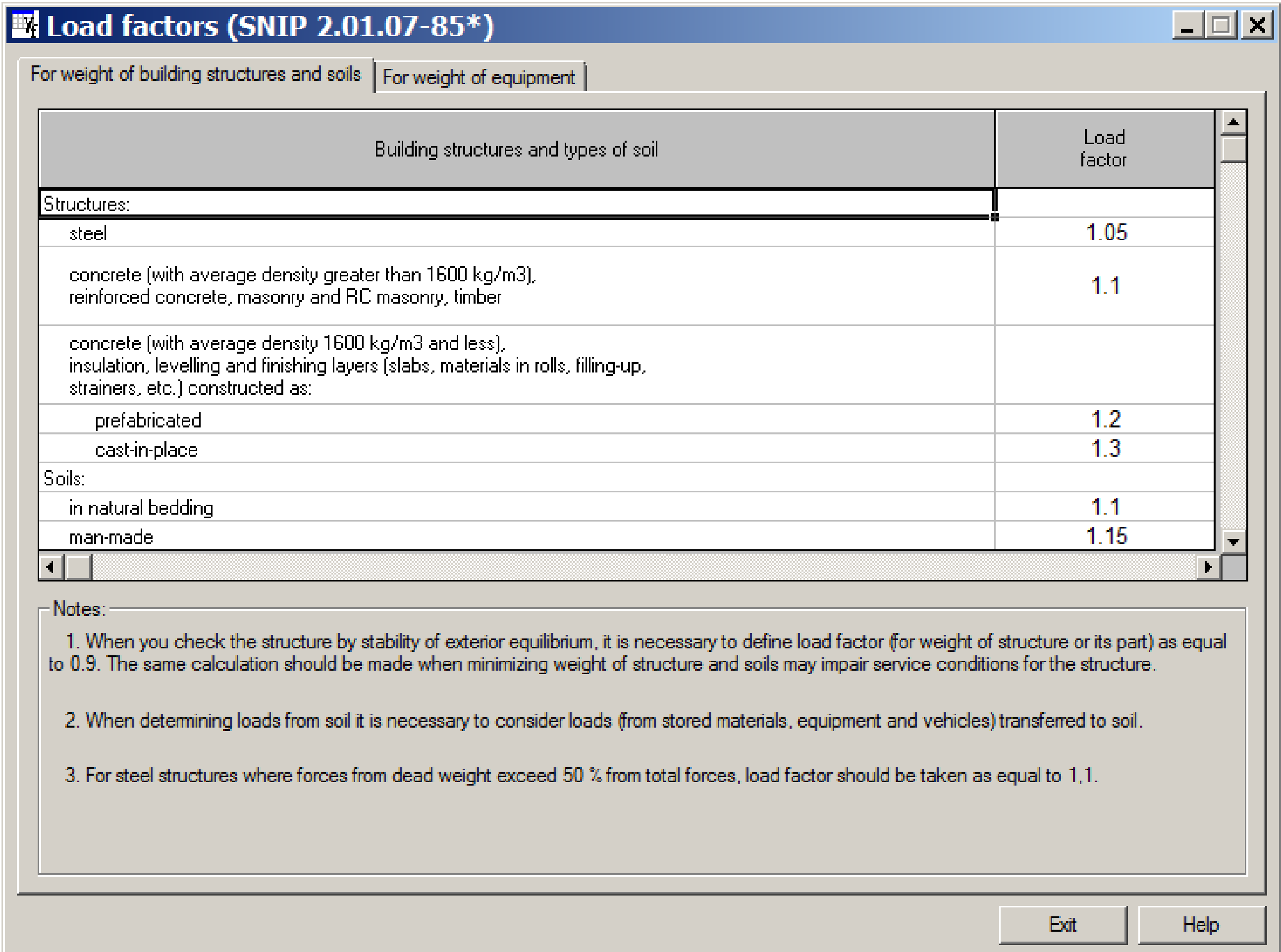
Load factors
Dead weight of multi-layer coating
Snow loads
Wind loads
Ice loads
Climatic thermal loads
Hazardous energy combinations of forces (EnergyCF)
Resonance check for wind turbulence.

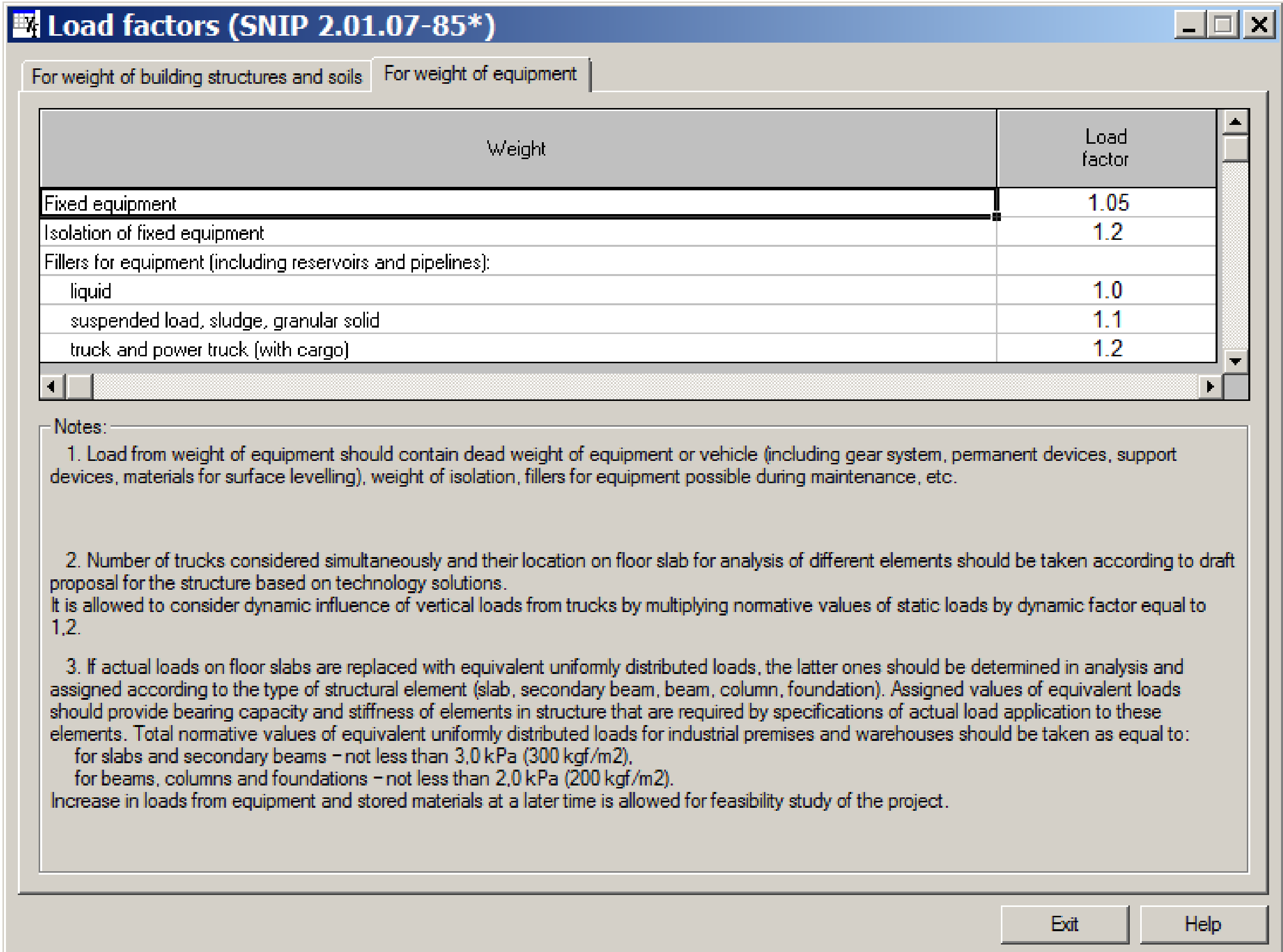
Figure 9.1

Load factors

The module contains reference tables for load factors:

– for weight of building structures and soils (table 1, SNIP 2.01.07-85* 'Loads and actions');
– for weight of equipment (table 2, SNIP 2.01.07-85* 'Loads and actions').





Dead weight of multi-layer coating

The module is mentioned for computation of normative and design loads from dead weight of the coat consisting of several layers. Resistance to heat transfer in multi-layer coat may be computed, if required. Calculation is made according to SNIP 2.01.07-85 'Loads and Actions' and SNIP II-3-79* 'Building Heat Engineering'.

When you define input data, the coating of user-defined materials is generated.

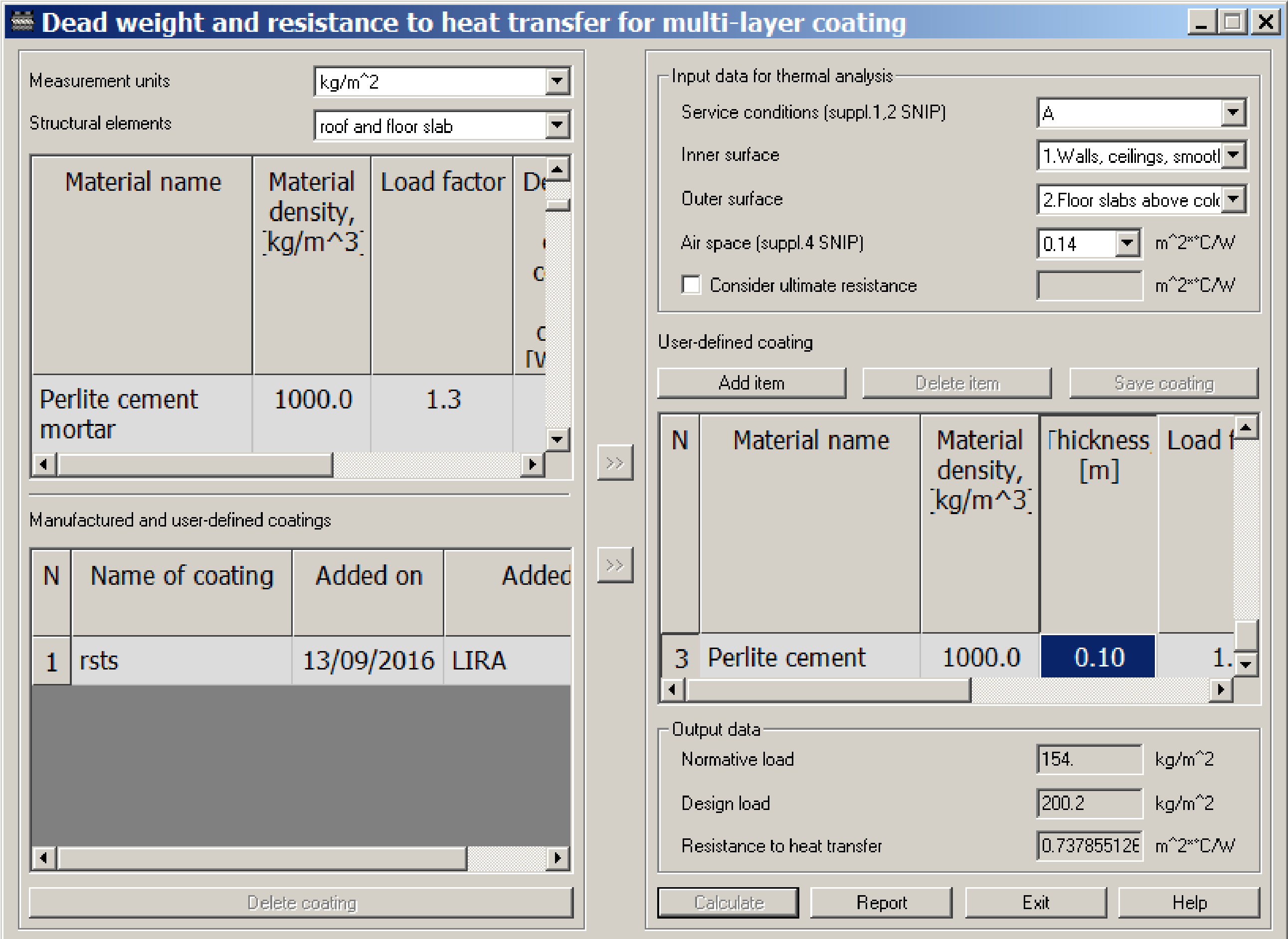


Figure 9.2

Input data

Select the **Measurement units** for the output data in the appropriate box.
In the **Structural elements** box, select tables of structural elements for materials: roofs and floor slabs, walls, thermal insulation or other.
Then, select with the pointer necessary material in the table.

To add selected material to user-defined coating, click **Add**.
You could select materials simply by changing the tables by turns. The next layer is added at the end of user-defined coating.
To add manufactured or previously defined coating into the current user-defined coating, select appropriate coating and add it with the **Add** button ">>". This button becomes available when you select name of coating in the table.
You could add several manufacture or previously defined coatings into the current user-defined coating. The next coating will be added at the end of user-defined coating.

To add material that is not available in any of structural element tables, it is necessary to add empty row to the user-defined coating. To do this, click the row above which you should add empty row. The **Add item** button becomes available. Click this button to add the empty row above the selected one.

 www.liraland.com

To edit cells of the added row (just as any other cell), double-click appropriate cell. In this case, the background of the cell in the edit mode will be of system colour **Window text** (for standard Windows, it is white).

To delete item from the user-defined coating, click the row that should be deleted. Then **Delete item** button becomes available. Click this button to delete the row.

Define input data for thermal analysis.

To take account of (or to ignore) coefficients for heat transfer of inner and outer surfaces of enclosing structures, in the **Input data for thermal analysis** area, select (or remain clear) appropriate items from the lists.
These are values of heat transfer coefficients for inner and outer surfaces of enclosing structures. These values are taken from tables 4 and 6 SNIP II–3–79* respectively.
To consider the **air space** in the user-defined coating, define the value of resistance to heat transfer of the air space in the appropriate box.
To save the user-defined coating, click **Save coating**. In another dialog box that appear, define **Name of coating** and **Additional data about coating**. Name of coating - short name for the coating (up to 40 characters) that has certain meaning and allows you to identify the coating in future. Additional data about coating - it may be the surname of the customer who created the coating, company name, etc. (up to 50 characters).
When you click **Calculate**, if the coating is not saved, the program automatically suggests that you save it.

Output data

Output data contains normative load, design load and resistance to heat transfer.
To present a report document in HTML format, click **Report**.

Dead weight of multi-layer coating "rsts"

Material name	Normative load, [kg/m^2]	Load factor	Design load, [kg/m^2]
Cement and sand mortar 1800.000[kg/m^3]*0.010[m]	18.000	1.300	23.400
Cement and sand mortar 1800.000[kg/m^3]*0.020[m]	36.000	1.300	46.800
Perlite cement mortar 1000.000[kg/m^3]*0.100[m]	100.000	1.300	130.000
TOTAL	154.000	-	200.200

Resistance to heat transfer for multi-layer coating "rsts"

Service conditions (suppl.1,2 SNIP): A

Material name	Resistance to heat transfer for layer, [m^2*°C/W]
Cement and sand mortar 0.010[m]/0.760[W/m*°C]	0.013
Cement and sand mortar 0.020[m]/0.760[W/m*°C]	0.026
Perlite cement mortar 0.100[m]/0.260[W/m*°C]	0.385
Inner surface 8.700[W/m*°C]	0.115
Outer surface 17.000[W/m*°C]	0.059
Air space (suppl.4 SNIP)	0.140
TOTAL	0.738

Figure 9.3

Snow loads

This module is meant to compute snow loads on buildings and structures according to SNIP 2.01.07-85* (1987, 2003) and DBN V.1.2-2:2006. The program enables you to compute normative and design values of snow load on horizontal projection of roof according to schematic presentations mentioned in the table of mandatory Application 3 SNIP. For calculation by DBN V.1.2-2:2006 it is possible to compute design values of ultimate, service and quasi-permanent snow load.

The following types of load are realized: 1, 2, 2', 3, 4, 5, 6, 7, 10, 11, 12.

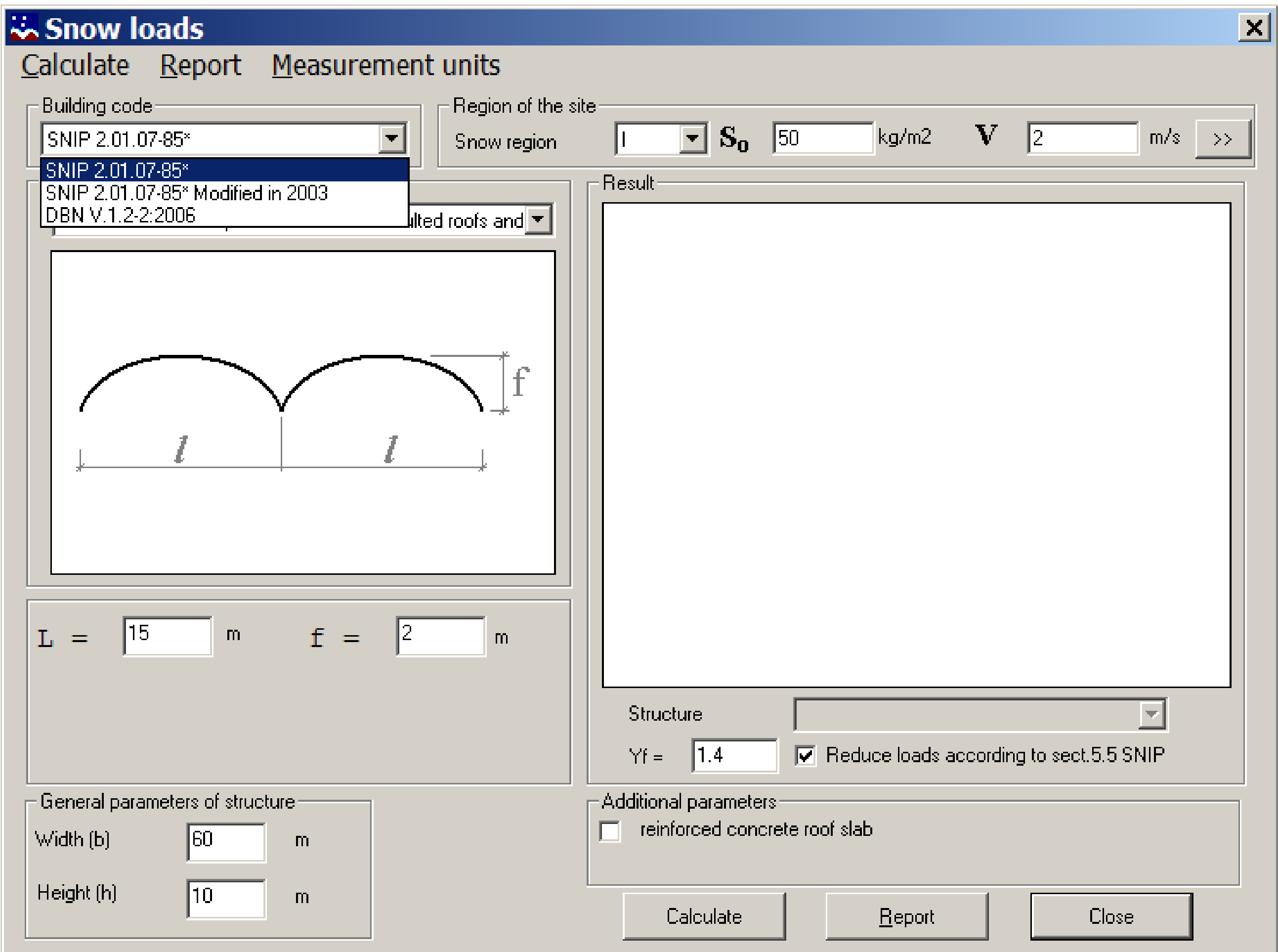


Figure 9.4

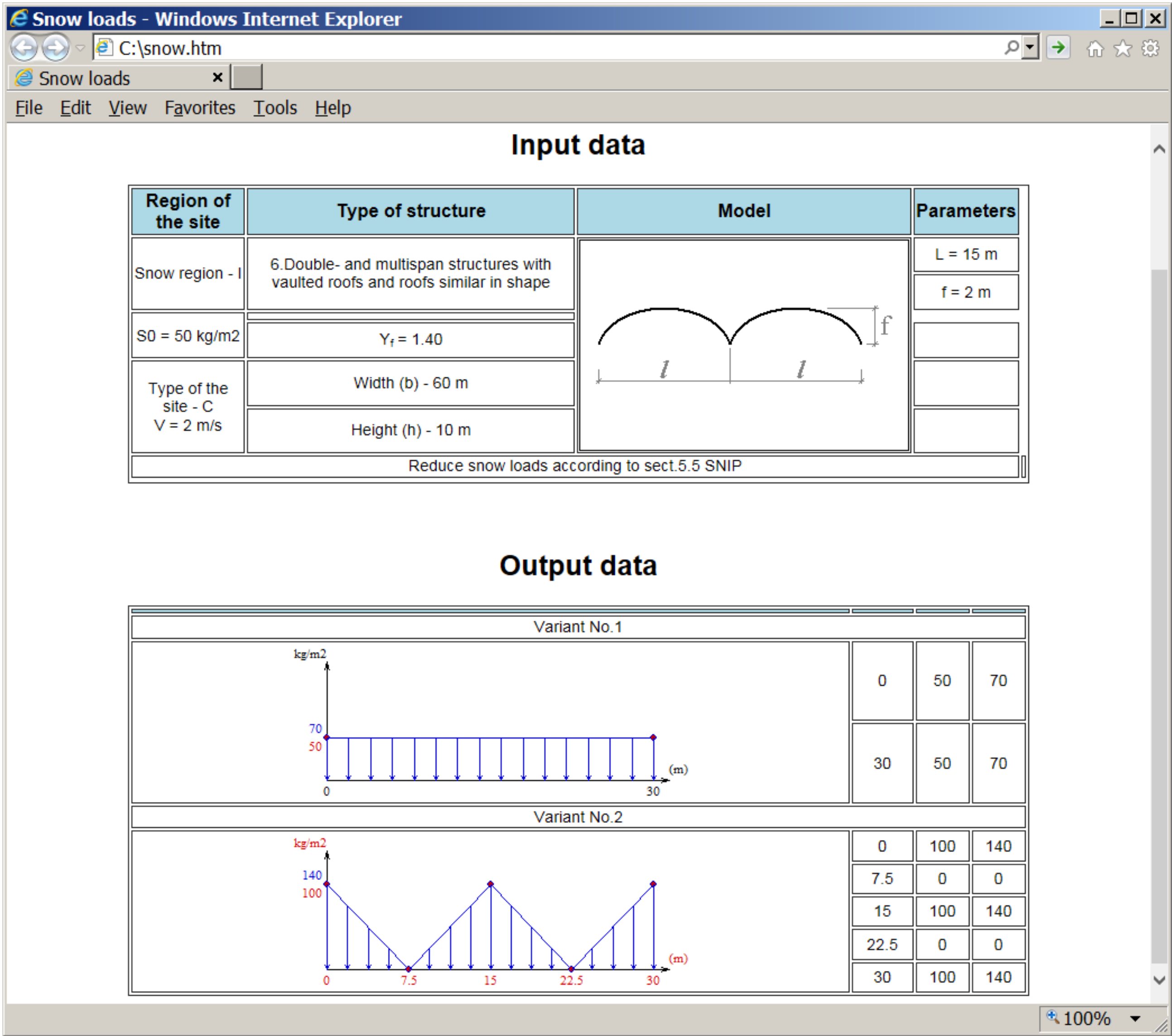
Input data
Select necessary building code in the appropriate list.

In the **Region of the site** area, select the **Snow region** in the list, the values for $\mathbf{S_0}$ and **V** will be displayed automatically according to selected building code.

To calculate the $\mathbf{S_0}$ and **V** values manually, in the **Snow region** list, select **User** (which means User-defined).
Define **Type of structure** in the appropriate list.
According to the type of structure, define (or edit) parameters of the structure.
For certain structures, in the **Structure** box it is necessary to select zone where the load is applied and select **Additional data**.
Input load factor $\boldsymbol{\gamma_f}$ or leave the default value (for calculation by SNIP).
Define appropriate data in the edit boxes.
To start calculation, click **Calculate**.

Output data

Output data is presented graphically on the screen at the right part of the dialog box. When you move the pointer along schematic presentation (graph), you could see values of snow load along the width of structure.
Output data may be presented as report that contains model of the structure and table of snow loads along the width of structure.
To generate and preview report file in HTML format, click **Report**.



Input data

Region of the site	Type of structure	Model	Parameters
Snow region - I	6.Double- and multispan structures with vaulted roofs and roofs similar in shape		L = 15 m
			f = 2 m
S0 = 50 kg/m2	Y_f = 1.40		
Type of the site - C V = 2 m/s	Width (b) - 60 m		
	Height (h) - 10 m		
Reduce snow loads according to sect.5.5 SNIP			

Output data

Variant No.1			
	0	50	70
	30	50	70
Variant No.2			
	0	100	140
	7.5	0	0
	15	100	140
	22.5	0	0
	30	100	140

Figure 9.5

Wind loads

This module is meant to compute wind loads on buildings and structures according to SNIP 2.01.07-85* and DBN V.1.2-2:2006.

The program enables you to compute normative and design values of average component of wind load with account of aerodynamic coefficient by schematic presentations mentioned in corresponding Appendix to building code.

The following types of load are realized:
for SNIP – 1, 2, 3, 4, 9, 10, 11, 12–a, 12–b, 13–17.
for DBN – 1, 2, 3, 4, 9, 10, 11, 12–a, 12–b, 13, 14, 16-18.

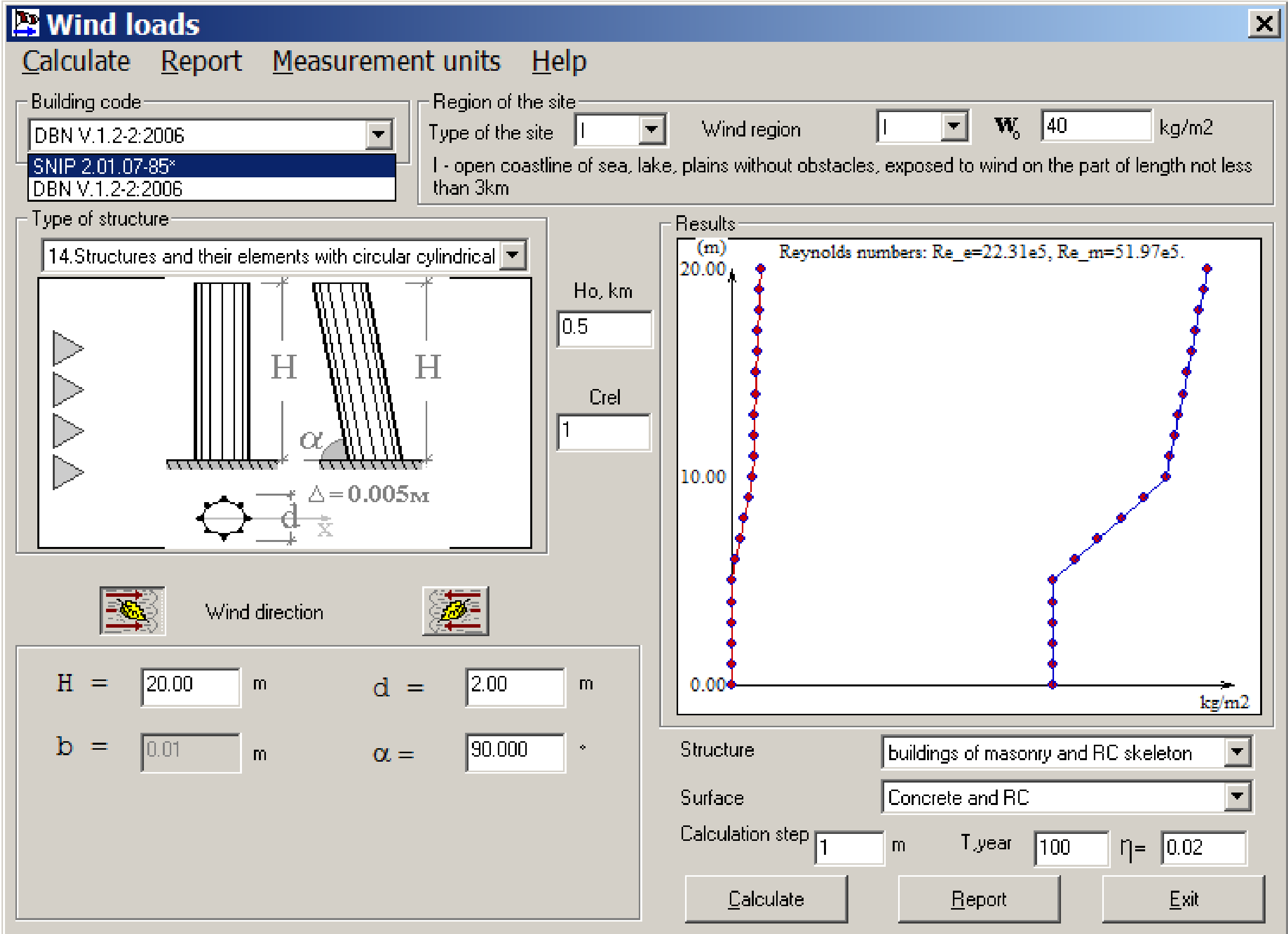
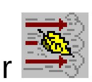
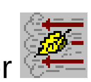


Figure 9.6

Input data

– In the **Region of the site** area, select **Type of the site** and **Wind region** in the appropriate lists and define normative value of wind force $\mathbf{W_0}$.

– Define **Type of structure** in the appropriate list.

– According to the type of structure, define wind direction by clicking either  or  button, define parameters of the structure.

– In the **Surface** list box, select the surface of wind load: left wall of the structure, right wall, etc.

– Define **Calculation step** and load factor γ_f .

– Define appropriate data in the edit boxes.

– To start calculation, click **Calculate**.

Output data

Output data is presented graphically on the screen at the right part of the dialog box. When you move the pointer along schematic presentation (graph), you could see values of wind load along the height of structure.

Output data may be presented as report that contains model of the structure and table of wind loads and their location relative to the ground surface.

To generate and preview report file in HTML format, click **Report**.

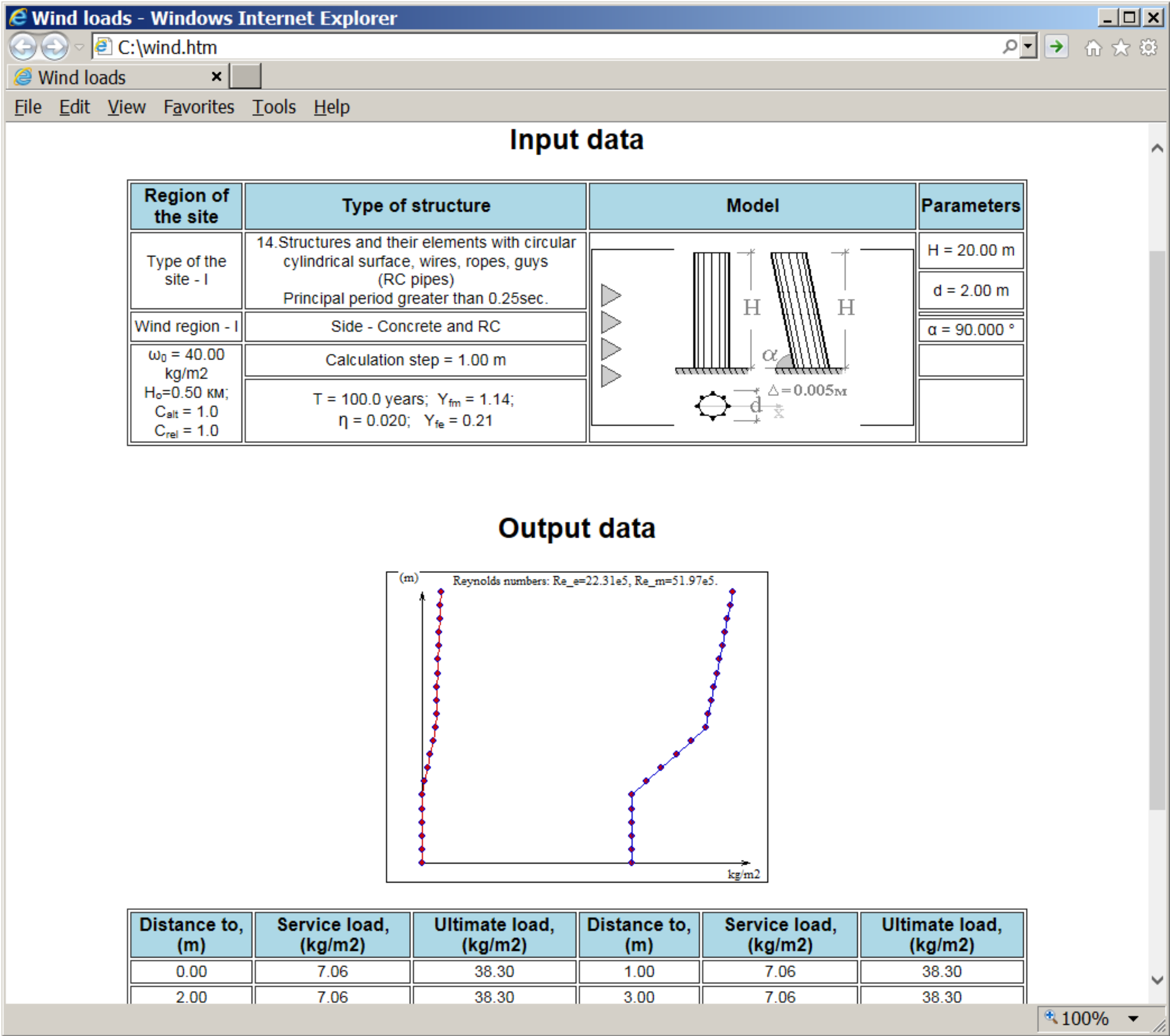


Figure 9.7

Ice loads

The module is meant to compute ice loads according to SNIP 2.01.07-85* and DBN V.1.2-2:2006.

The program enables you to compute normative and design values of linear and surface ice load.

Input data

Select the **Measurement units** in the appropriate box.

Select the building code in the appropriate box.
– In the **Element type** area of the dialog box, select **Circular section** option and define diameter.
– Select the **Ice region** in the list; in this case the values **b** (thickness of ice layer) will be displayed automatically. If you define **b** manually and leave this edit box, the ice region will be modified to **User** (user-defined).
– For calculation by DBN, in the same way select the **Ice region according to values of wind pressure**. In this case the value $\mathbf{W_b}$ (characteristic value of wind pressure for ice load) will be displayed automatically. In the appropriate box define **T** - average repetition period for ultimate design value.
– Define **h** – height above ground level where the ice load should be calculated.
– To start calculation, click **Calculate**.

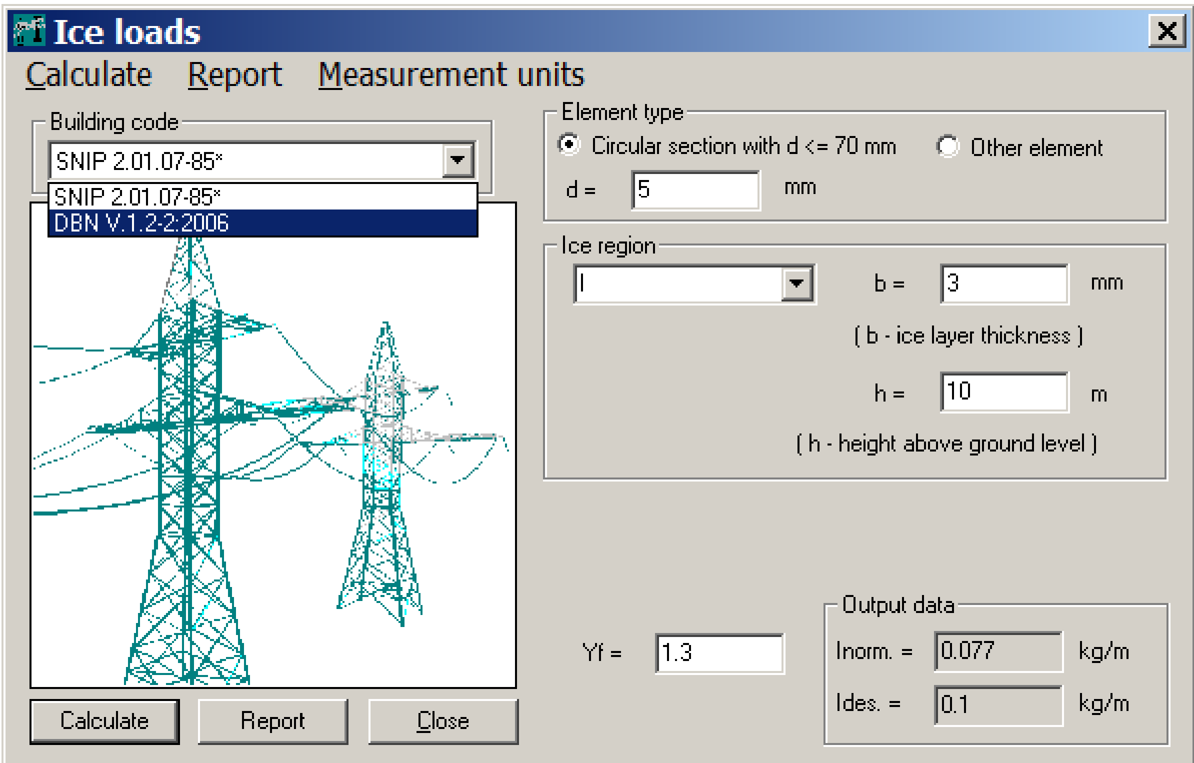
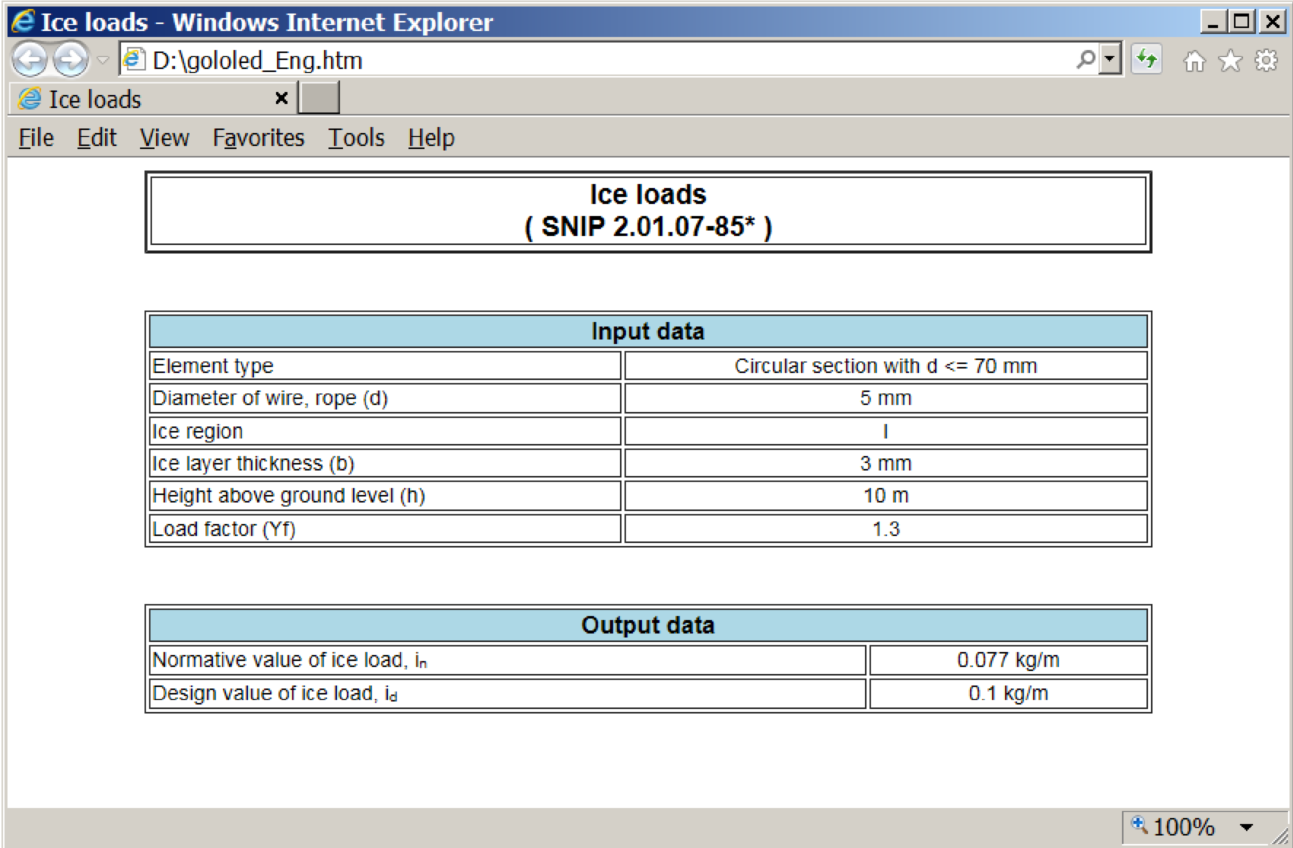


Figure 9.8

Output data

Output data is presented graphically on the screen. It also may be presented as report that contains input data and table with loads values.
To generate and preview report file in HTML format, click **Report**.



Ice loads
(SNIP 2.01.07-85*)

Input data	
Element type	Circular section with d <= 70 mm
Diameter of wire, rope (d)	5 mm
Ice region	I
Ice layer thickness (b)	3 mm
Height above ground level (h)	10 m
Load factor (Yf)	1.3

Output data	
Normative value of ice load, i_n	0.077 kg/m
Design value of ice load, i_d	0.1 kg/m

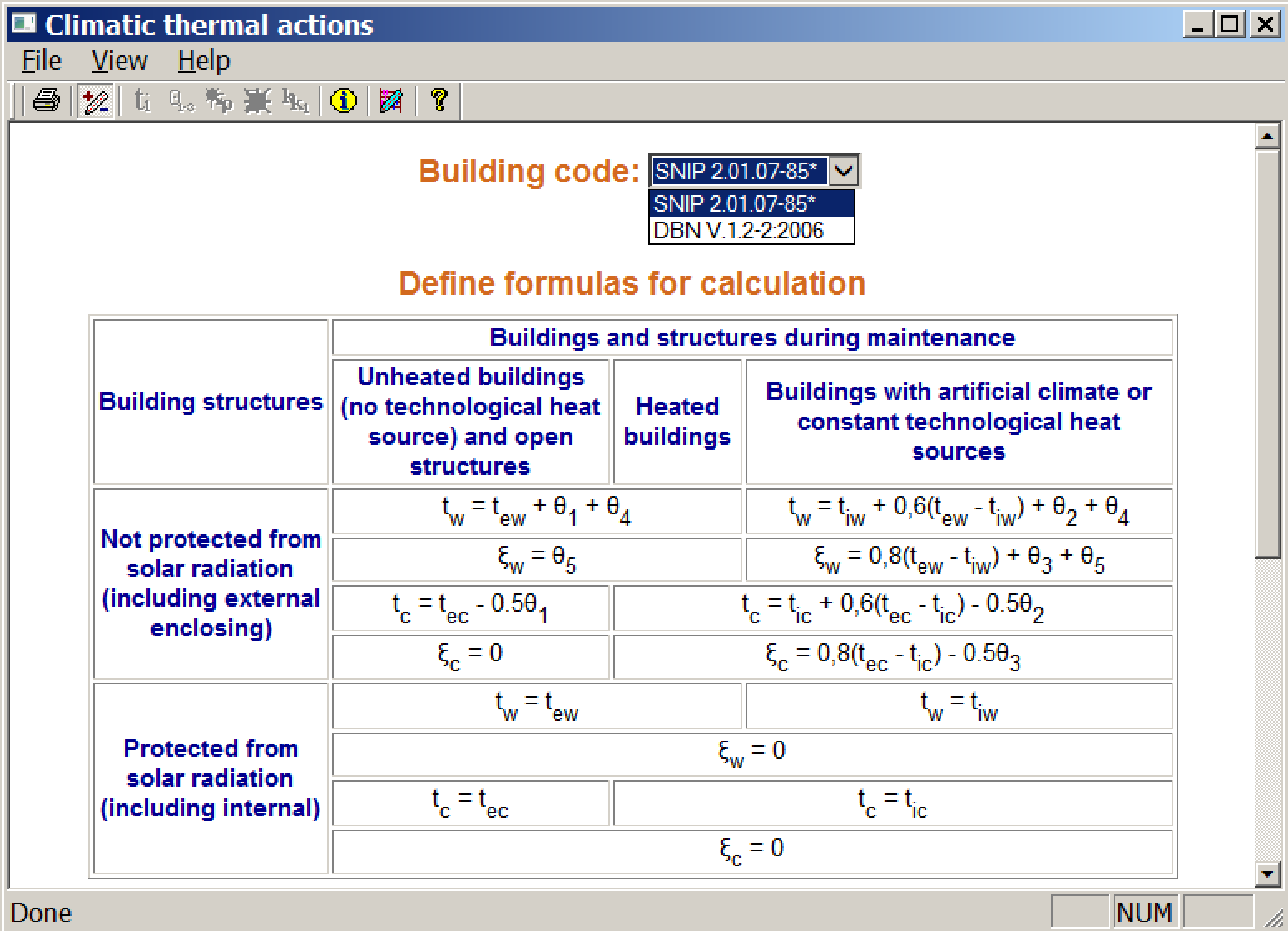
Climatic thermal loads

The module is meant for computation of climatic thermal loads on building structures according to SNIP 2.01.07-85* and DBN V.1.2-2:2006 'Loads and Actions'.

The data from the following documents is also considered in calculation:
– appendix 7 SNIP II-3-79* that contains coefficients for absorption of solar radiation by material of external surface of enclosing structure;
– appendix 5, 6, 7 SNIP 2.01.01-82 that contains maximum values of total (direct and diffuse) solar radiation.

Normative and design values of changes in mean temperatures are computed. It is also possible to compute mean daily temperatures of external air and initial temperature when the structure is closed to complete system.

 www.liraland.com



Building structures	Buildings and structures during maintenance: Unheated buildings (no technological heat source) and open structures	Heated buildings	Buildings with artificial climate or constant technological heat sources
Not protected from solar radiation (including external enclosing)	$t_w = t_{ew} + \theta_1 + \theta_4$		$t_w = t_{iw} + 0{,}6(t_{ew} - t_{iw}) + \theta_2 + \theta_4$
	$\xi_w = \theta_5$		$\xi_w = 0{,}8(t_{ew} - t_{iw}) + \theta_3 + \theta_5$
	$t_c = t_{ec} - 0.5\theta_1$	$t_c = t_{ic} + 0{,}6(t_{ec} - t_{ic}) - 0.5\theta_2$	
	$\xi_c = 0$	$\xi_c = 0{,}8(t_{ec} - t_{ic}) - 0.5\theta_3$	
Protected from solar radiation (including internal)	$t_w = t_{ew}$		$t_w = t_{iw}$
	$\xi_w = 0$		
	$t_c = t_{ec}$	$t_c = t_{ic}$	
	$\xi_c = 0$		

Figure 9.9

Input data

In the **Define formulas for calculation** table, click appropriate rows (columns) and select formulas for calculation of thermal loads by type of buildings during maintenance (**Unheated buildings (no technological heat source) and open structures**, **Buildings with artificial climate or constant technological heat sources**) and building structures (**Not protected from solar radiation (including external enclosing)**; **Protected from solar radiation (including internal)**).

Define input data for calculation of parameters: $\mathbf{t_{ew}, t_{ec}, t_{0w}, t_{0c}}$.

Click the **Increments of Mean Temperatures and Temperature Drop** button on the toolbar.

Select with the pointer increment of temperature for building structures (**steel or reinforced concrete, concrete, masonry and masonry reinforcing with specific thickness**).

Click the **Absorption coefficient for solar radiation** button on the toolbar and select with the pointer appropriate **Material for external surface of enclosing structure** in the table.

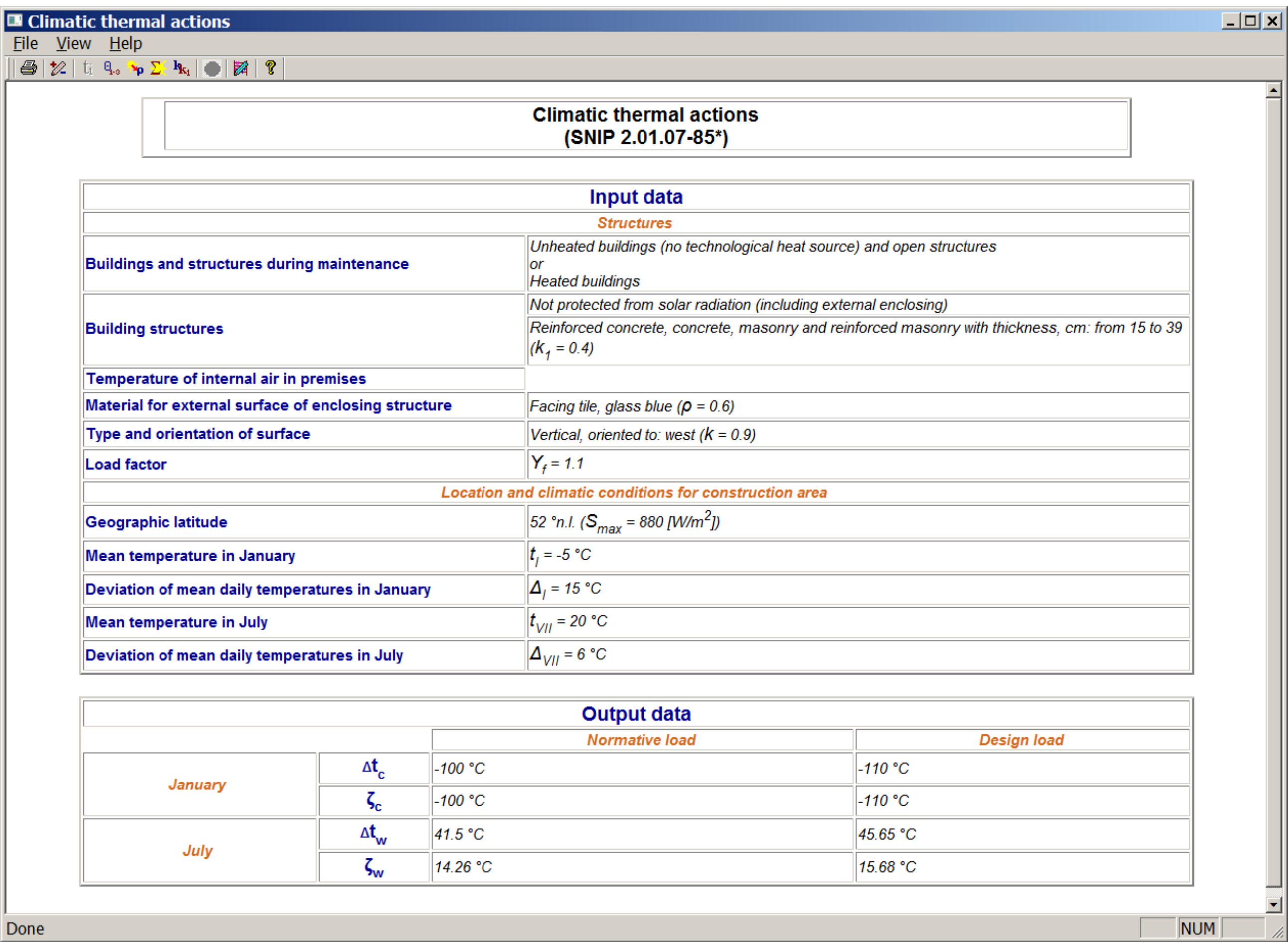
Click the **Total daily radiation, Smax** button on the toolbar.
In the appropriate list box, define the type and orientation of surface, select with the pointer necessary value of latitude.

Click the **Coefficients K and K1** button on the toolbar.
Select with the pointer the row with type and orientation of surface and in another table, the row with building structures.
To show or hide the input data on the screen (current state of the problem), use the **Show/hide input data** button on the toolbar.

Click the **Calculate** button on the toolbar.

Output data

After calculation, the program generates report that contains input and output data.
It is possible to save or print the report file (**FILE/Save report, FILE/Print**).



Climatic thermal actions
(SNIP 2.01.07-85*)

Input data	
Structures	
Buildings and structures during maintenance	*Unheated buildings (no technological heat source) and open structures or Heated buildings*
Building structures	*Not protected from solar radiation (including external enclosing)* *Reinforced concrete, concrete, masonry and reinforced masonry with thickness, cm: from 15 to 39 (k_1 = 0.4)*
Temperature of internal air in premises	
Material for external surface of enclosing structure	*Facing tile, glass blue (ρ = 0.6)*
Type and orientation of surface	*Vertical, oriented to: west (k = 0.9)*
Load factor	$Y_f = 1.1$
Location and climatic conditions for construction area	
Geographic latitude	*52 °n.l. (S_{max} = 880 [W/m^2])*
Mean temperature in January	t_I = -5 °C
Deviation of mean daily temperatures in January	Δ_I = 15 °C
Mean temperature in July	t_{VII} = 20 °C
Deviation of mean daily temperatures in July	Δ_{VII} = 6 °C

Output data			
		Normative load	*Design load*
January	Δt_c	-100 °C	-110 °C
	ζ_c	-100 °C	-110 °C
July	Δt_w	41.5 °C	45.65 °C
	ζ_w	14.26 °C	15.68 °C

Figure 9.10

Hazardous energy combinations of forces (EnergyCF)

The module enables you to determine hazardous combinations of forces in bar section (plane eccentric tension-compression) according to criterion of extreme energy in the section.

Energy of section in the *i*-th load case is described by formula:

$$Ui = \frac{1}{2E}\left(\frac{N_i^2}{F} + \frac{M_i^2}{J}\right),$$

where **i=1÷n**; **n** – number of independent load cases; **Ni** – axial force; **Mi** – bending moment; **E** – modulus of elasticity; **F** – area of section; **J** – moment of inertia for section. It is considered that forces in the section are computed in liner elastic analysis.

Energy of section in the *i*–th load case may be presented as

$$Ui = \frac{1}{2E}\left(X_i^2 + Y_i^2\right),$$

where $Xi = \frac{Ni}{\sqrt{F}}, Yi = \frac{Mi}{\sqrt{J}}$ components of vector $[R_i] = [X, Y]_i$. This makes it possible to manage the process of determining hazardous combinations of forces as the search for vectors with max length among all possible vector sums $[R_i]$.

Section energy from combinations of load cases reaches extreme values at the vertices of polygon (convex closed shell with discrete set of all vector sums $[R_i]$). Number of vertices in polygon-shell is equal to **2n**, while total number of vector sums equals to $2^n \left(2n << 2^n\right)$.

The program generates polygon-shell with **2n** vertices. For every vertex, the program generates the sum of components **X, Y** and the list (combination) with numbers of load cases mentioned in this sum. For vertices, the following values are also computed: energy of section **U**, axial force **N**, moment **M** and stress $\sigma = \frac{N}{F} + \frac{M}{W}$.
Theoretical base and algorithm for calculation are described in detail in [6, 7].

Input data

The program window is presented in the figure below.

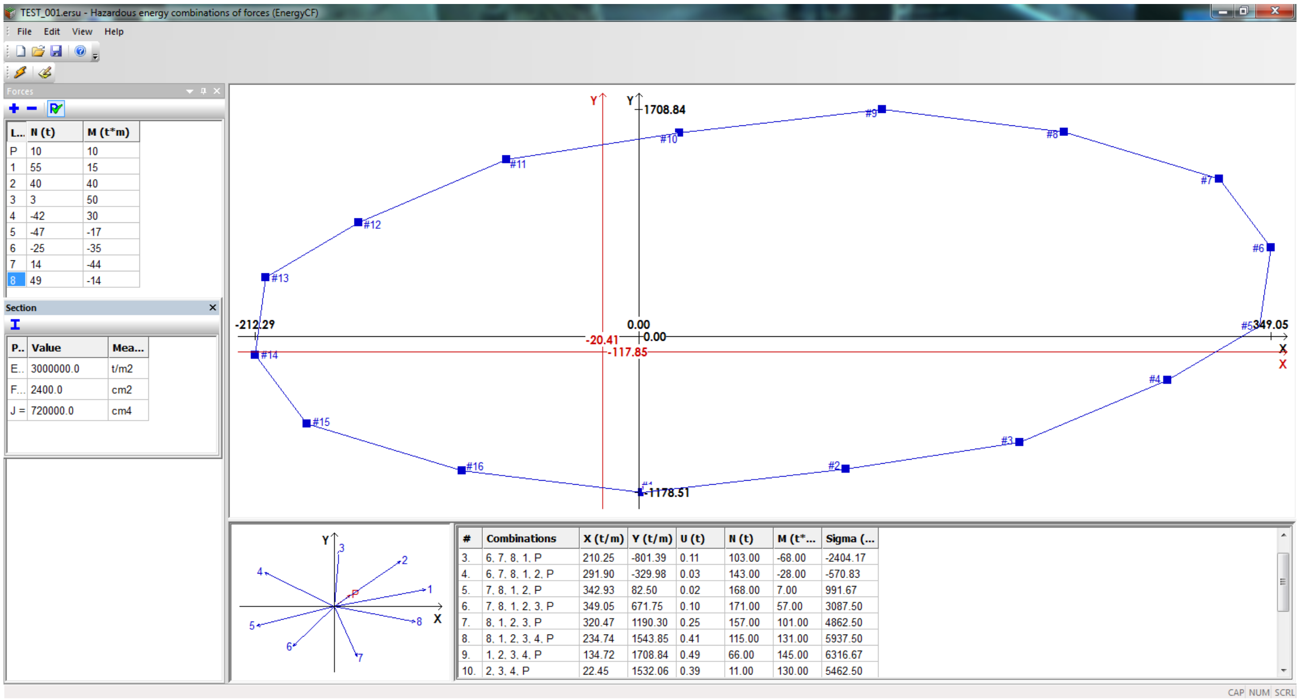
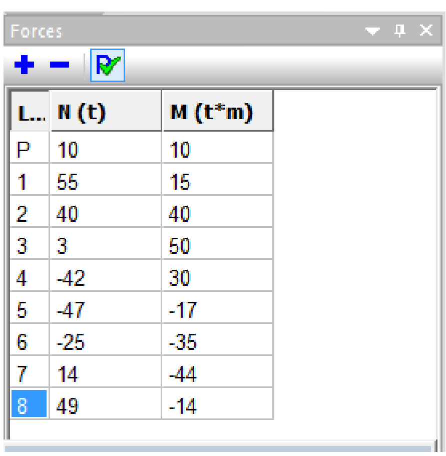


Figure 9.11

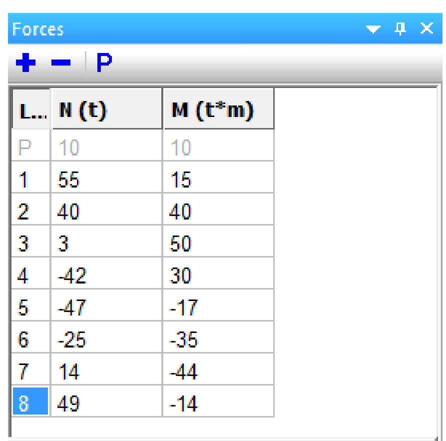
Forces

In the first row of the **Forces** table, define forces **N** and **M** from dead load case. All components of the dead load case are denoted with index **P**. To define forces from dead load case, click the **Account of dead load case** button on the toolbar of this table. If this button is not active, then the row of forces from the dead load case remains unavailable and calculation will be made without account of dead load case.

From the second row, define forces from other load cases in ascending order of their numbers beginning from the first one. To add or delete rows in the table, use the **Add** and **Delete** buttons on the standard toolbar.



a)



b)

Figure 9.12

Section properties

In the **Section** table, define modulus of elasticity for material **E**, area of section **F** and moment of inertia for the section **J**. When you click the **Section** button , the **Parametric sections**, program appears on the screen. There you could compute values **F** and **J**. Computed values will be automatically displayed in the appropriate boxes. Section modulus is computed by formula $W = \sqrt{\frac{F * J}{3}}$.

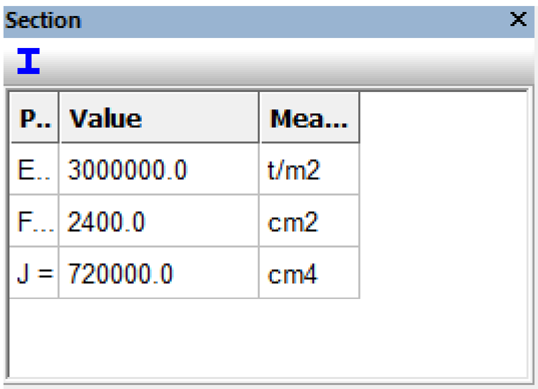


Figure 9.13

To start calculation, click the **Calculate** button on the standard toolbar.

Output data

After calculation you will see convex shell with hazardous combinations of forces computed at vertices of the shell. Shell and axes **X, Y** are coloured blue.

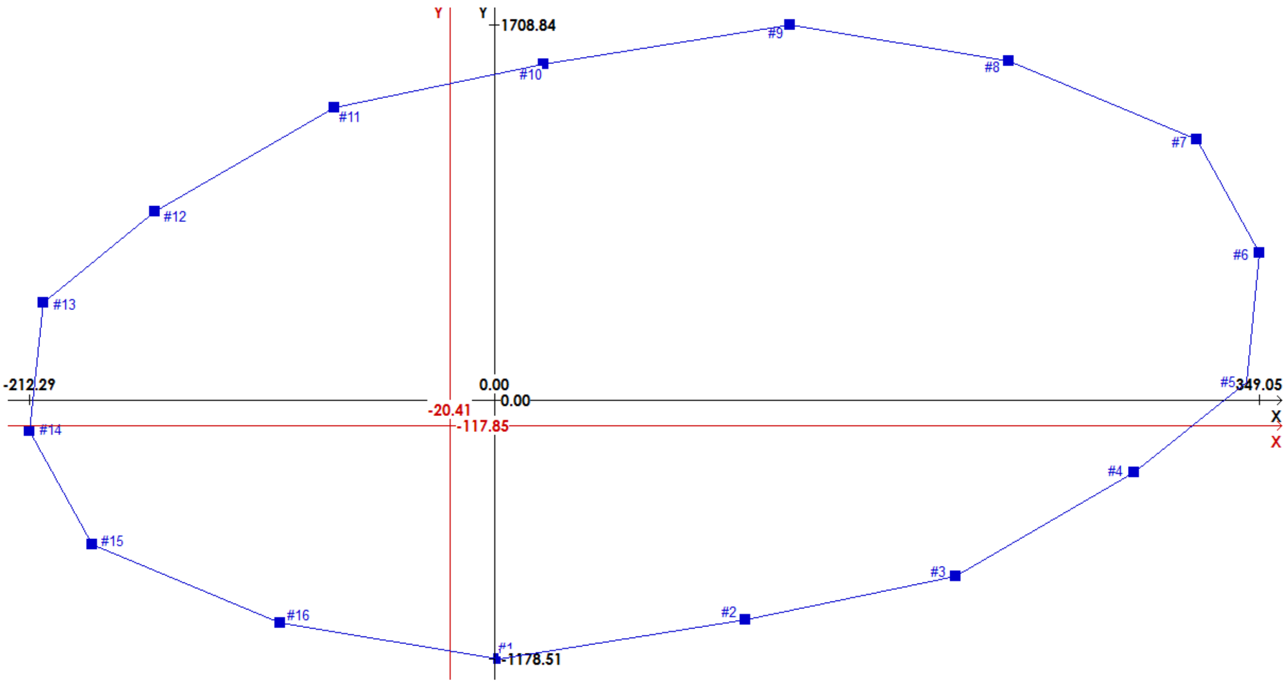
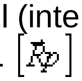


Figure 9.14

If the dead load case is taken into account, then centre of the shell (intersection of blue axes **X, Y**) is moved in parallel direction against direction of vector  by the value of its length. It is displayed with the second pair of axes **X, Y** coloured red.

The table of results contains: numbers of load cases included into appropriate combination, sums of components **X** and **Y** at the vertices of shell, appropriate forces **N** and **M**, stresses **Sigma** and energy **U**.

#	Combinations	X (t/m)	Y (t/m)	U (t)	N (t)	M (t*...	Sigma (...
1.	5, 6, 7, 8, P	2.04	-1178.51	0.23	1.00	-100.00	-4162.50
2.	5, 6, 7, 8, 1, P	114.31	-1001.73	0.17	56.00	-85.00	-3308.33
3.	6, 7, 8, 1, P	210.25	-801.39	0.11	103.00	-68.00	-2404.17
4.	6, 7, 8, 1, 2, P	291.90	-329.98	0.03	143.00	-28.00	-570.83
5.	7, 8, 1, 2, P	342.93	82.50	0.02	168.00	7.00	991.67
6.	7, 8, 1, 2, 3, P	349.05	671.75	0.10	171.00	57.00	3087.50
7.	8, 1, 2, 3, P	320.47	1190.30	0.25	157.00	101.00	4862.50
8.	8, 1, 2, 3, 4, P	234.74	1543.85	0.41	115.00	131.00	5937.50
9.	1, 2, 3, 4, P	134.72	1708.84	0.49	66.00	145.00	6316.67
10.	2, 3, 4, P	22.45	1532.06	0.39	11.00	130.00	5462.50
11.	2, 3, 4, 5, P	-73.48	1331.72	0.30	-36.00	113.00	4558.33
12.	3, 4, 5, P	-155.13	860.31	0.13	-76.00	73.00	2725.00
13.	3, 4, 5, 6, P	-206.17	447.83	0.04	-101.00	38.00	1162.50
14.	4, 5, 6, P	-212.29	-141.42	0.01	-104.00	-12.00	-933.33
15.	4, 5, 6, 7, P	-183.71	-659.97	0.08	-90.00	-56.00	-2708.33
16.	5, 6, 7, P	-97.98	-1013.52	0.17	-48.00	-86.00	-3783.33

Figure 9.15

When you click any vertex on the shell, the vertex will be coloured red and the correspondent row in the table will be coloured blue.
When you click any row in the table, appropriate vertex of a shell will be coloured red.

Input vectors $[\vec{R}_i]$ are displayed next to the table.

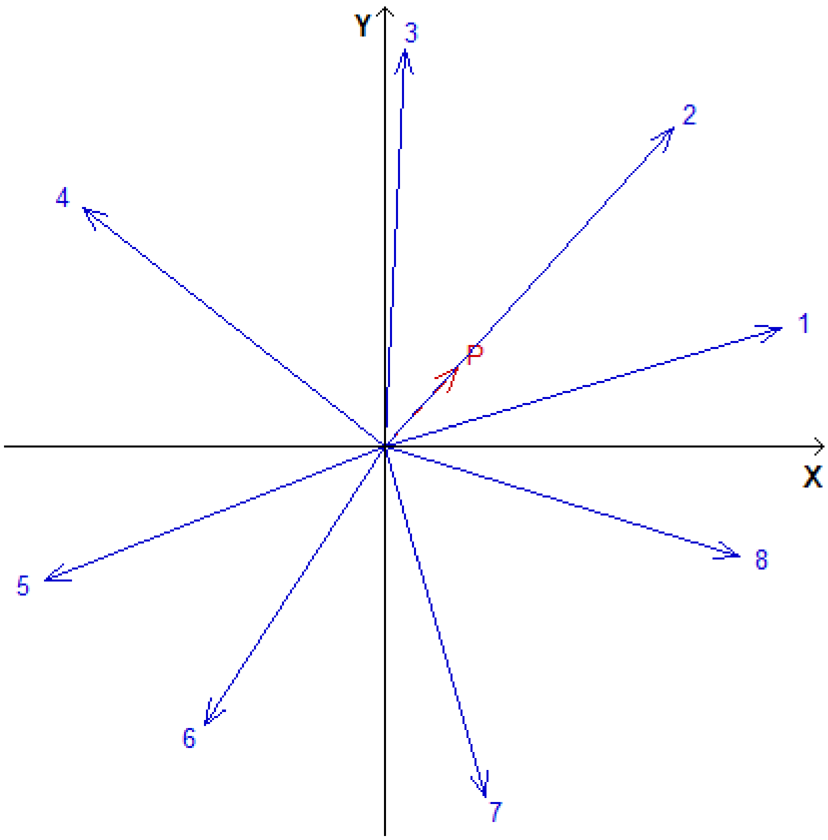


Figure 9.16

To generate report file in HTML format, click **Report** button  on the toolbar.

Resonance check for wind turbulence

The module enables you to check buildings and structures for resonance wind turbulence according to SP 20.13330.2011 and DBN V.1.2-2:201X.

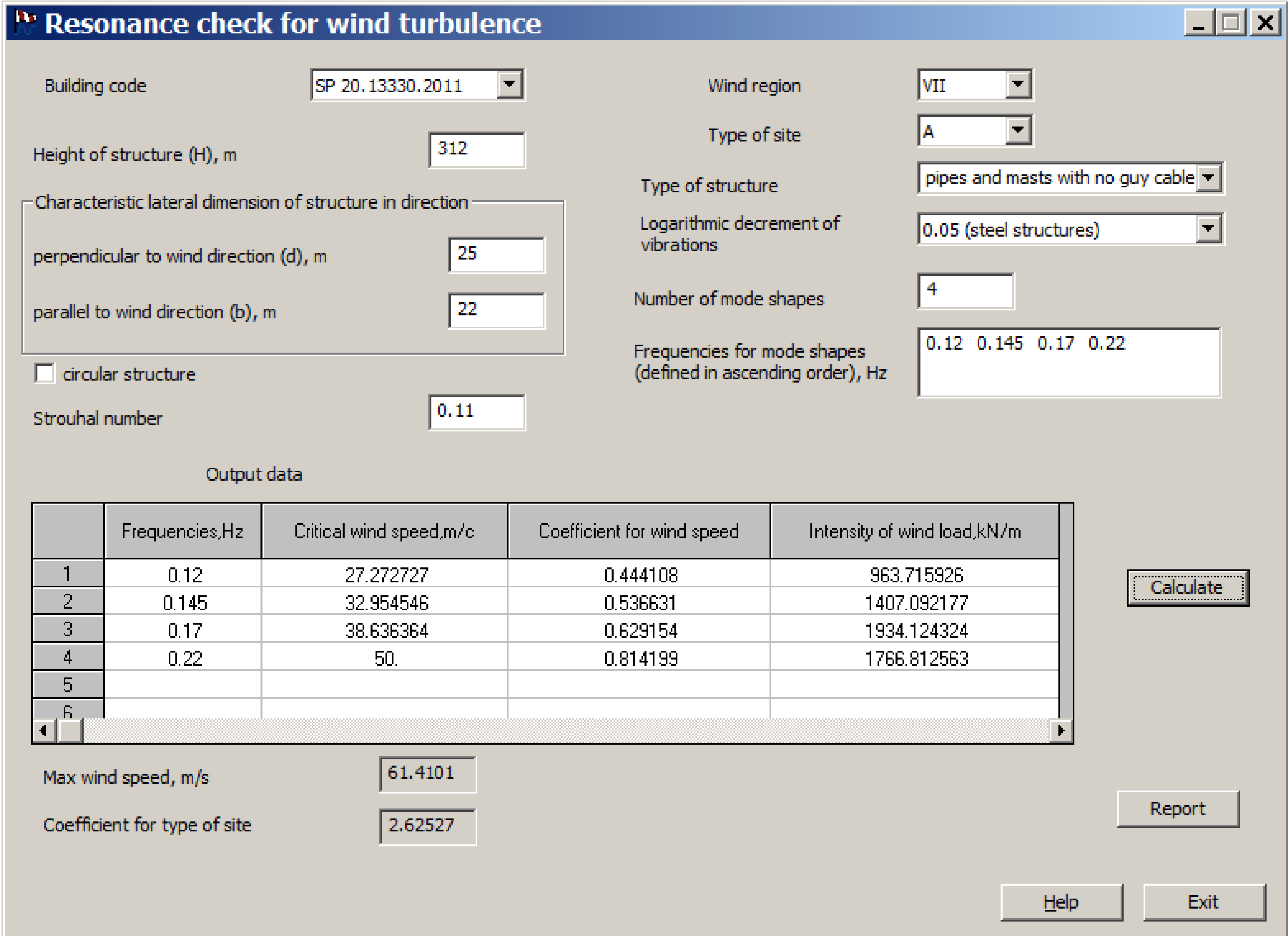


Figure 9.17

Input data

Select necessary building code from the list.

Define the following data:
– **H** – height of structure;
– **d** – characteristic lateral dimension of structure in direction ***perpendicular*** to wind direction;
– **b** – characteristic lateral dimension of structure in direction ***parallel*** to wind direction;
– shape of structure – circular or not;
– wind region: Ia, I, II, III, IV, V, VI, VII (for SP) or 1, 2, 3, 4, 5 (for DBN);
– Strouhal number St:
  – for circular cross-sections St = 0.2;
  – for sections with sharp edges St = 0.11;
  – it is allowed to define arbitrary value.
– number of mode shapes (by default 1);
– frequencies $\mathbf{f_i}$ in Hz for $\mathbf{1 \le i \le KF}$ mode shapes (defined in ascending order);
– type of structure; equivalent height $\mathbf{Z_e}$ is determined according to this type:
  – pipes and masts with no guy cables – $\mathbf{Z_e = 0.8*H}$;
  – buildings and structures with smoothly varying shape of cross section – $\mathbf{Z_e = 0.8*H}$;
  – other buildings and structures – $\mathbf{Z_e = H}$.
– type of site: A, B, C (for SP) or I, II, III, IV (for DBN);

– logarithmic decrement of vibrations:
  – for steel structures δ = 0.05;
  – for reinforced concrete structures δ = 0.10.

Output data

The following data is computed and displayed on the screen:

– **Vmax(Z_e)** – max wind speed, m/s;
– **Vcr,i** – critical wind speed for every mode shape, m/s;
– **kvi = Vcr,i / Vmax(Z_e)** – coefficient for wind speed for every mode shape;
– **Fi(z)** – intensity of wind load for resonance wind turbulence for every mode shape in direction perpendicular to average wind speed, kN/m;
– **Cy,cr** – aerodynamic coefficient (in the report file);
– **k(Z_e)** – coefficient for type of site;
– Ratio **H/d**;
– Ratio **b/d**.

Important. Speed coefficient **kvi** is required to determine average (**Wm,cr**) and pulsation (**Wp,cr**) components of resonance in direction parallel to wind speed: **Wm,cr = kvi^2 * Wm; Wp,cr = kvi^2* Wp**.

To present a report document in HTML format, click **Report**.

Deflections

Deflections

This chapter contains module that enables you to calculate inelastic deflections in multispan continuous beam for arbitrary live and short-term loads.

Analysis of inelastic deflections

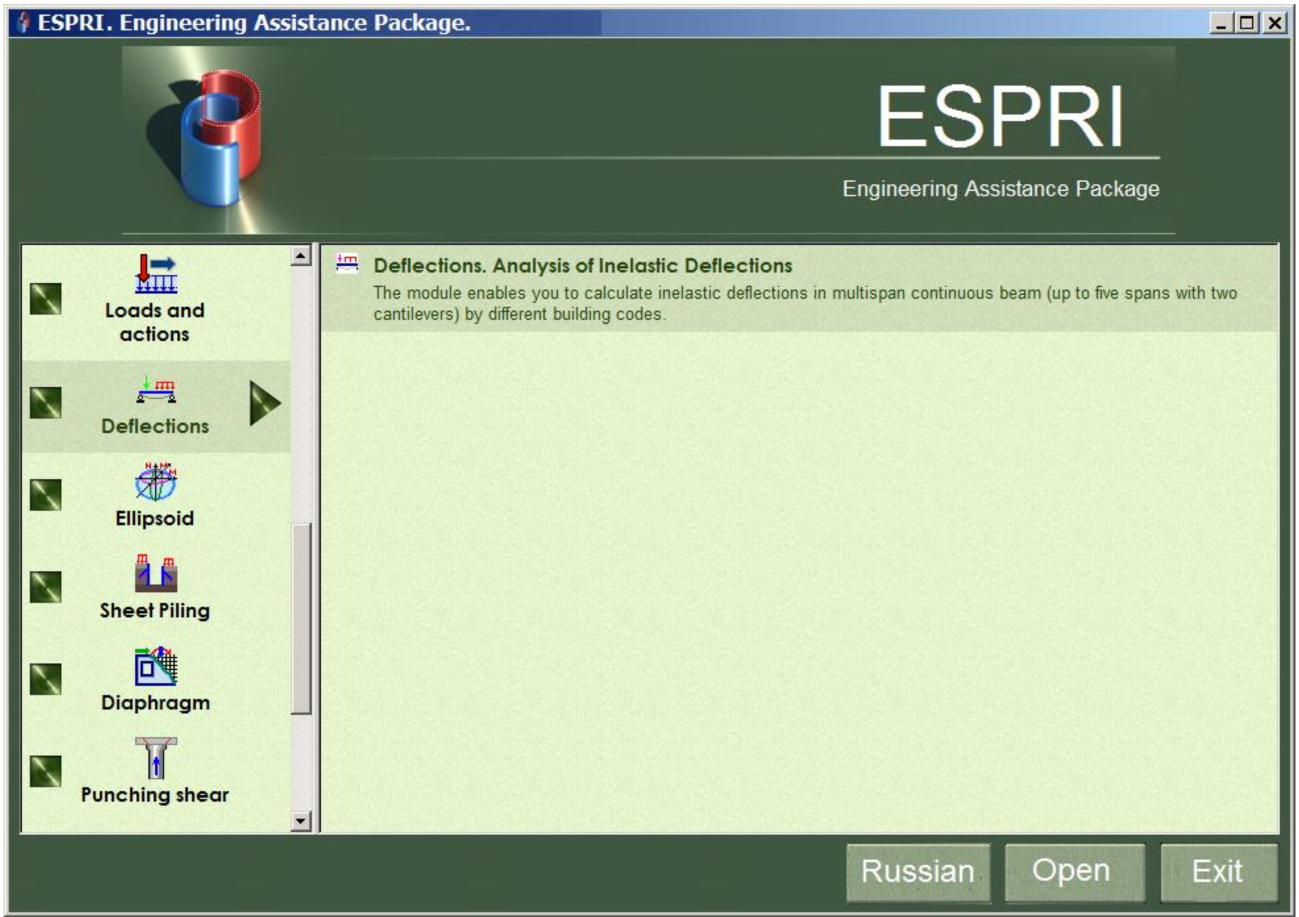


Figure 10.1

Analysis of inelastic deflections

The module enables you to calculate inelastic deflections in multispan continuous beam (up to five spans with two cantilevers) for arbitrary live and short-term loads.

The following building codes are supported: SNIP 2.03.01-84*, SNIP 52-01-2003, Eurocode 2, DSTU 3760-98, TSN-102-00*.

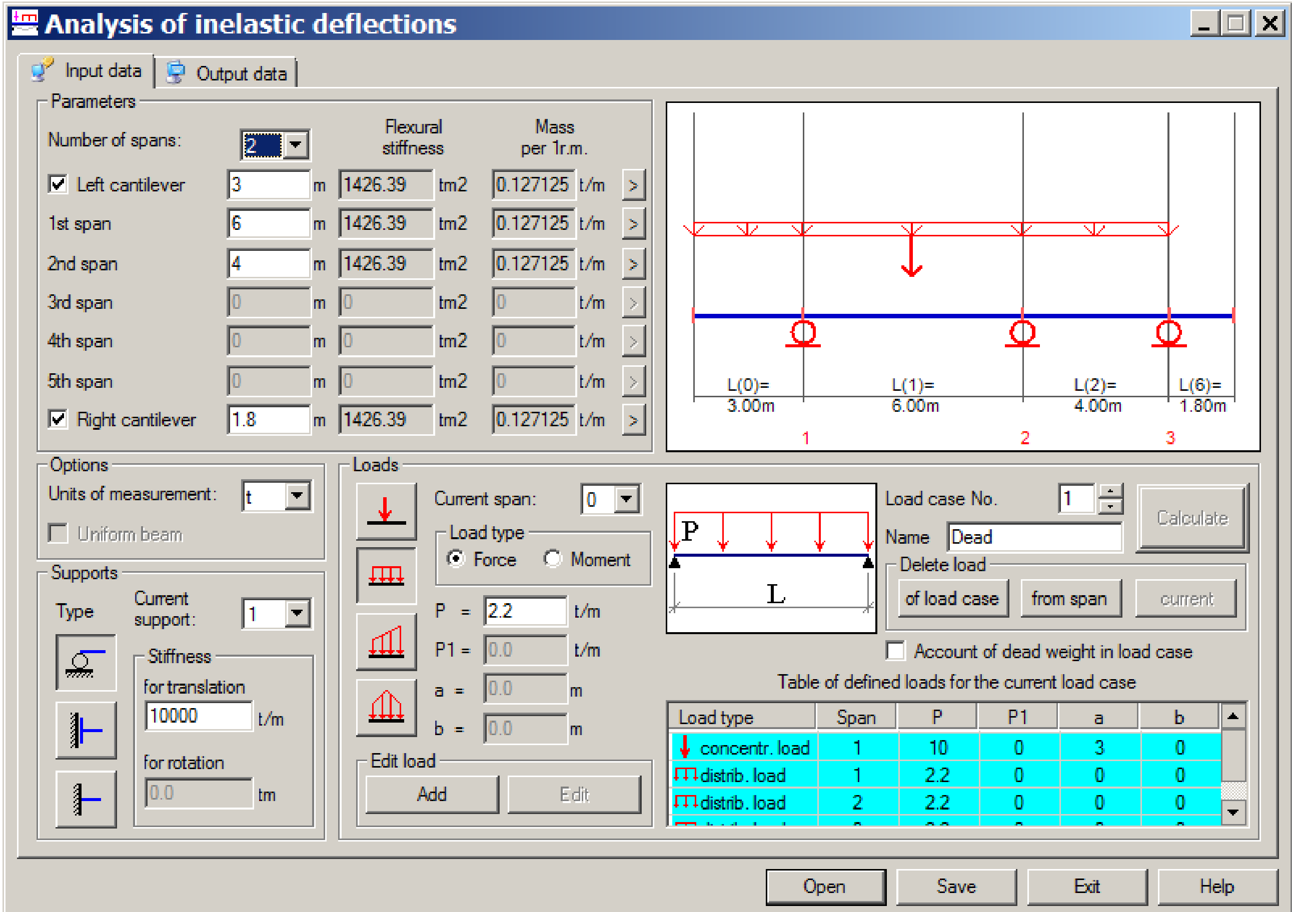


Figure 10.2

Input data

– In the **Parameters** area, in the **Number of spans** box, select number of spans and, if required, select appropriate check boxes to define the presence of the left or right cantilever.

– In the **Options** area, define units of measurement for loads and stiffness – **t** or **kN**.

– In the appropriate boxes, define span (cantilever) values in **m**.

– In the **Parameters** area, when you click the > button, the **Stiffness types** dialog box appears on the screen. In this dialog box, define type of section and its parameters, class of concrete and class of reinforcement, arrange reinforcement in sections (areas of upper and lower reinforcement with distances to them) for zones; length of zones may be defined either in absolute (**m**) or in relative (**%**) coordinates (e.g. length of the current zone 0,25 means that the section will be defined for the first quarter of the current span; for the second defined section, length of the current zone is equal to 0,5 – this section is assigned to the second quarter of the span, etc.)

Important. Option to select building code is available only for the ***first span***. Selected building code is automatically assigned to all spans of the beam. Building code should be selected before defining the data about reinforcement zones (for the new problem) or when all data about reinforcement zones is deleted (for analysed problem) as building code and section are defined there.

Click **Parameters >>** to define specific parameters according to selected building code. When necessary parameters are defined, click **Close**.

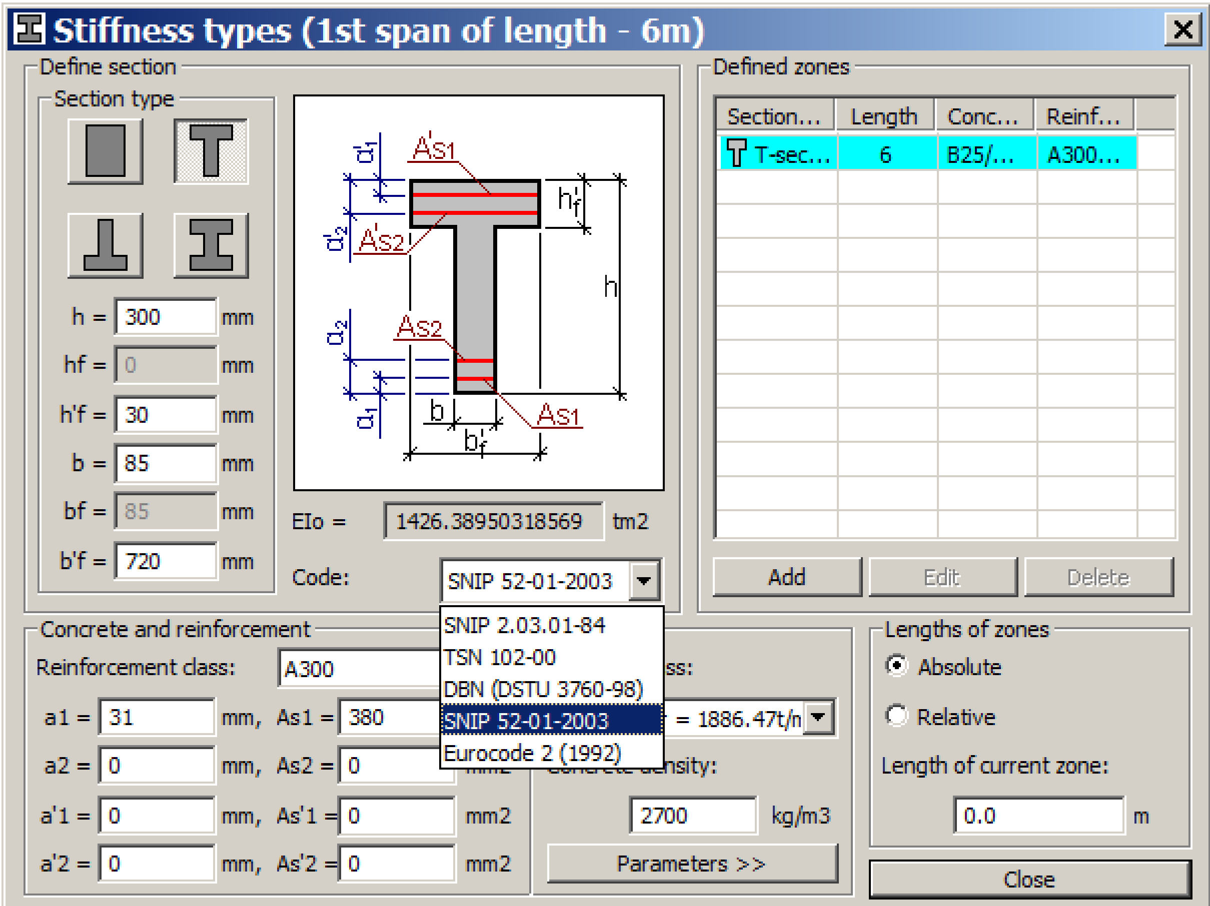


Figure 10.3

– In the **Supports** area, in the **Current support** box or by direct click on schematic presentation, select supports that boundary conditions should be applied to, define the type of boundary condition (it will be displayed on schematic presentation) and stiffness for translation or rotation, if required.

– Select number of current load case (up to 3 load cases). Dead and Live load cases will be treated as long-term load cases while short-term – as short-term ones. In this case Live and Short-term load cases will be applied span-by-span to beam to find out extreme values of deflections.

– In the **Loads** area, in the **Current span** box or by direct click on schematic presentation, select spans that the load should be applied to (it will be displayed on schematic presentation).

– In the **Loads** area, select the load type by clicking appropriate button, define necessary parameters and click **Add** under the **Edit load**.

– To modify parameters of load, select the load in the table of loads, input new parameters in the **Loads** area and click **Edit** under the **Edit load**.

– To delete load from the current span, select appropriate load in the table of loads. Then in the **Delete loads** area, click **current**. To delete all loads from the current span, in the **Delete loads** area, click **from span**.

– To add dead weight in the current load case, select appropriate check box.

– When the input data is defined, to start calculation, click **Calculate**.

Output data

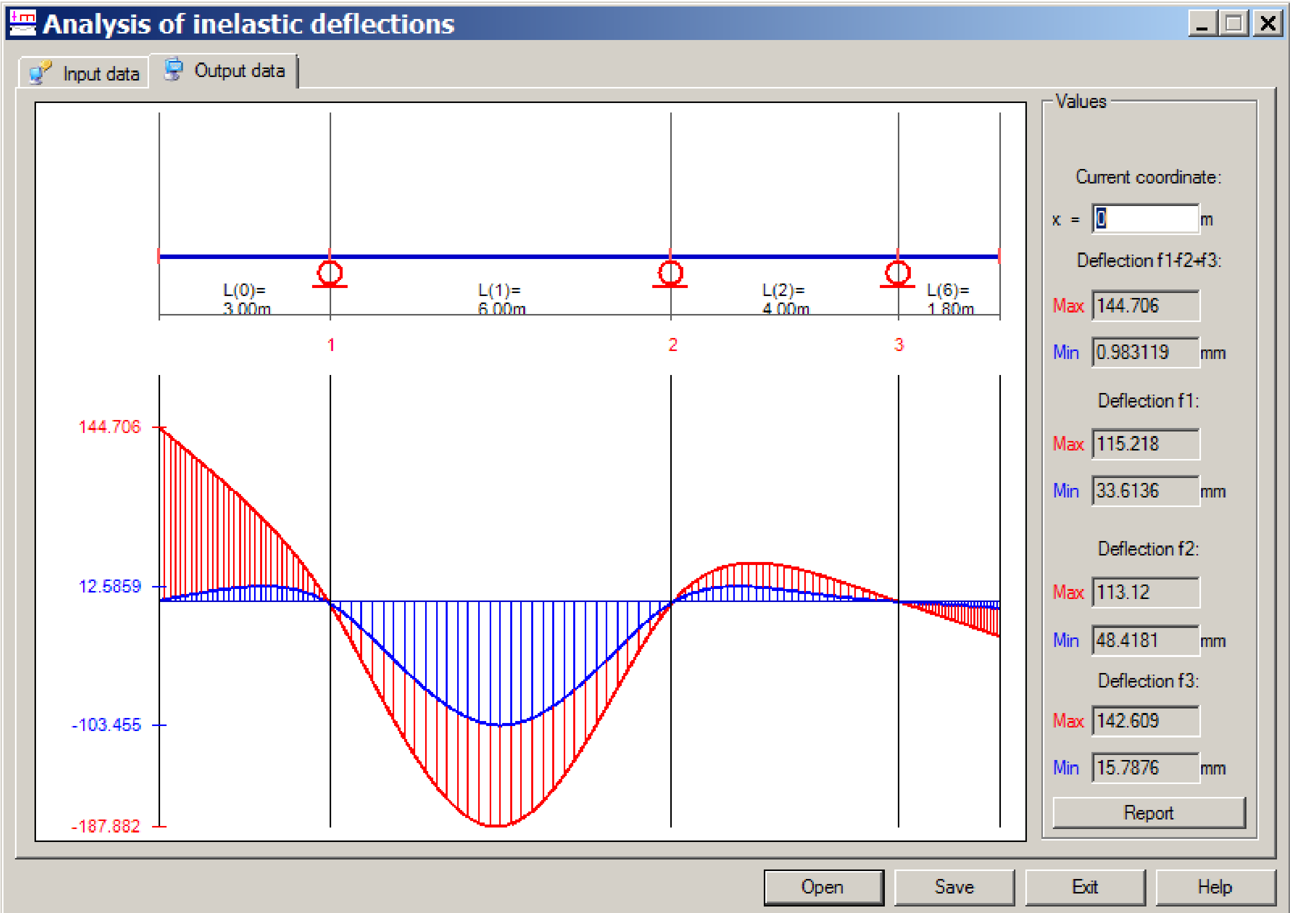


Figure 10.4

On the **Output data** tab you will see the diagram of deflections with extreme values.

To find out ordinates of diagrams at any point of beam, drag the pointer across the diagram and view the ordinate for the diagrams of: complete deflections (**f1-f2+f3**), deflections from short-term action of the whole load (**f1**), deflections from short-term action of dead and live loads (**f2**) and deflections from long-term action of dead and live loads (**f3**).

To present a report document in HTML format, click **Report**.

Ellipsoid

Ellipsoid

The chapter contains module that enables you to estimate strength of reinforced concrete (RC) sections in axial force and bending moments.

Ellipsoid. Bearing capacity of RC elements

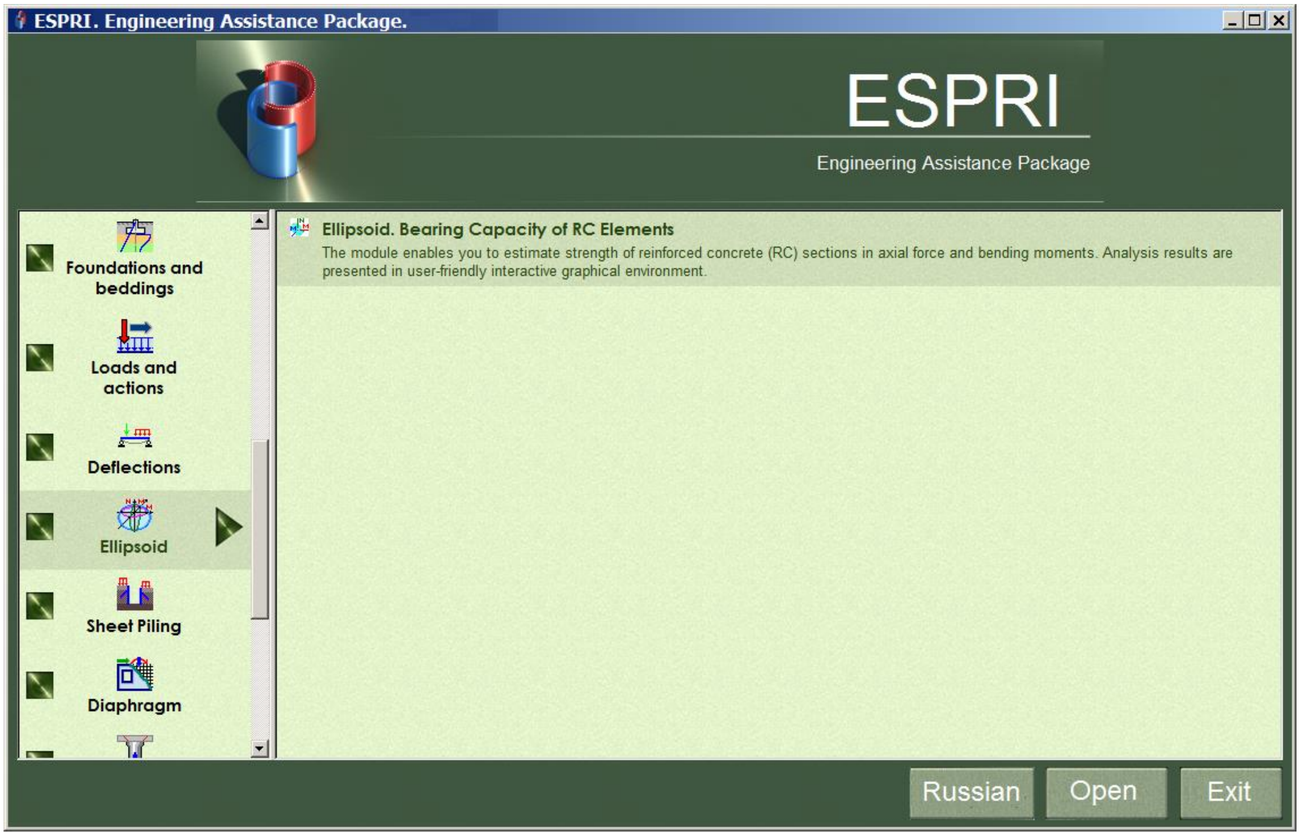


Figure 11.1

Ellipsoid. Bearing capacity of RC elements

The module enables you to estimate strength of reinforced concrete (RC) sections in axial force and bending moments according to SNIP 2.03.01-84* 'Concrete and Reinforced Concrete Structures'. The surface of bearing capacity is generated for the section with defined parameters.
In addition, for the loads specified in the table it is possible to calculate reserve factors by bearing capacity. Reserve factor is the ratio of bearing capacity of the section to defined forces. This factor indicates how many times the real forces may be increased until they exceed the bearing capacity of the section.

Input data

– In the **Column section** area, define the section for element of RC structure and define its geometric properties.

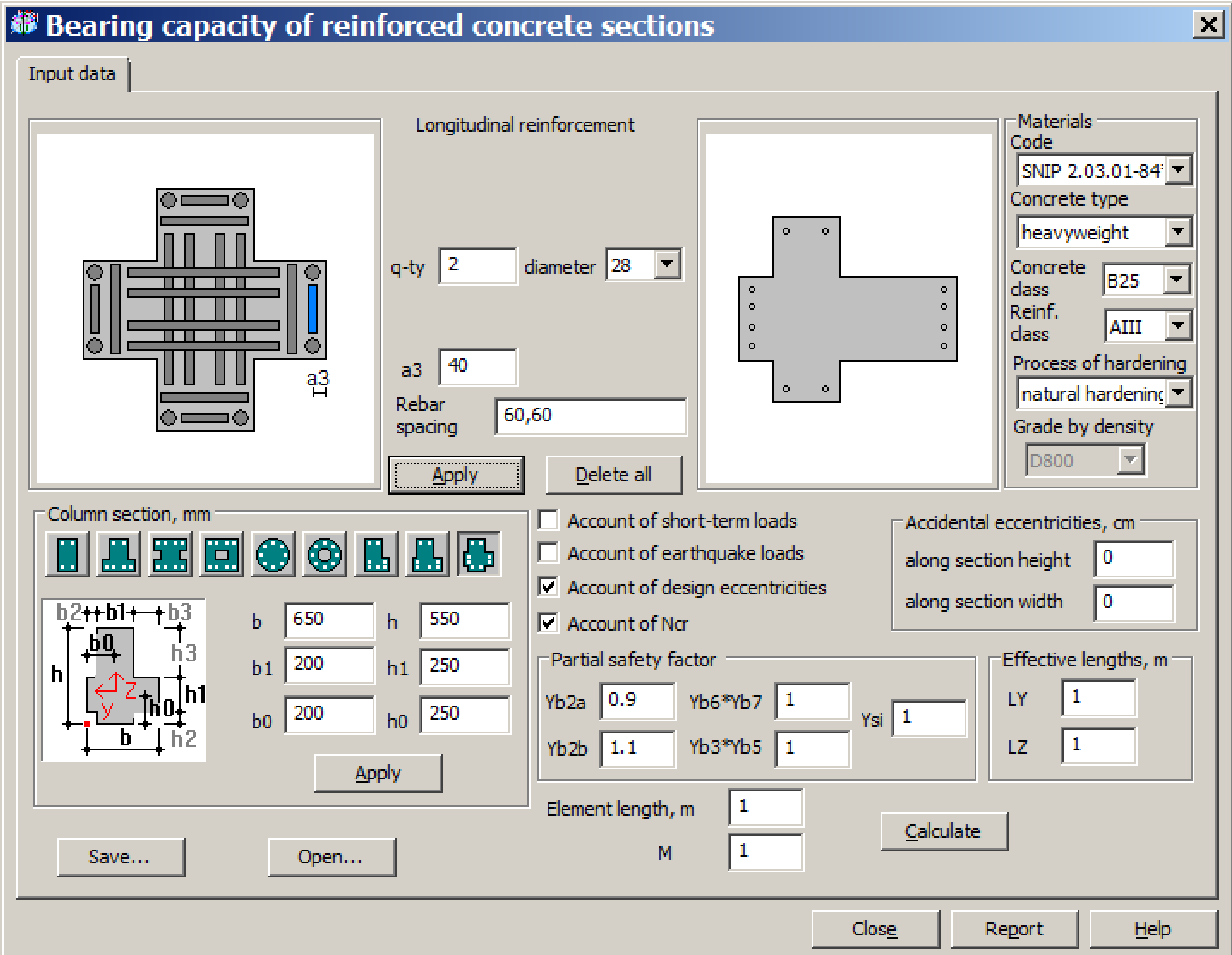


Figure 11.2

– To arrange reinforcement, select (with the pointer) the element on schematic presentation and then define diameter (diameters), location and spacing in the appropriate boxes. When you select certain elements, the **Arrange uniformly** check box becomes available. If this check box is selected, then defined number of rebars will be automatically arranged uniformly according to reinforcement pattern and defined location.

– In the appropriate boxes define additional data according to SNIP 2.03.01-84* : class of concrete and hardening process for concrete; if necessary, select appropriate check boxes to take account of short-term and/or earthquake loads; define partial safety factors, etc.

– When the input data is defined, click **Calculate**.

Output data

The output data is presented as the surface of bearing capacity for the specified section.

According to results of calculation it is possible to check bearing capacity of the section for the loads specified in the table.

To preview projections $\mathbf{M_yM_z}$, $\mathbf{M_yN}$, $\mathbf{M_zN}$, dimetric projection of the surface or all views simultaneously, click appropriate button. You could define certain point that should be checked (necessary values of forces) in the appropriate boxes. When the certain point (forces) is defined, to display projections, click **Redraw**. In this case the program generates three orthogonal projections that pass through the specified point. The point is presented on the ellipsoid and on its projections. If the specified point is located inside the ellipsoid surface, then the section is adequate for the specified forces.

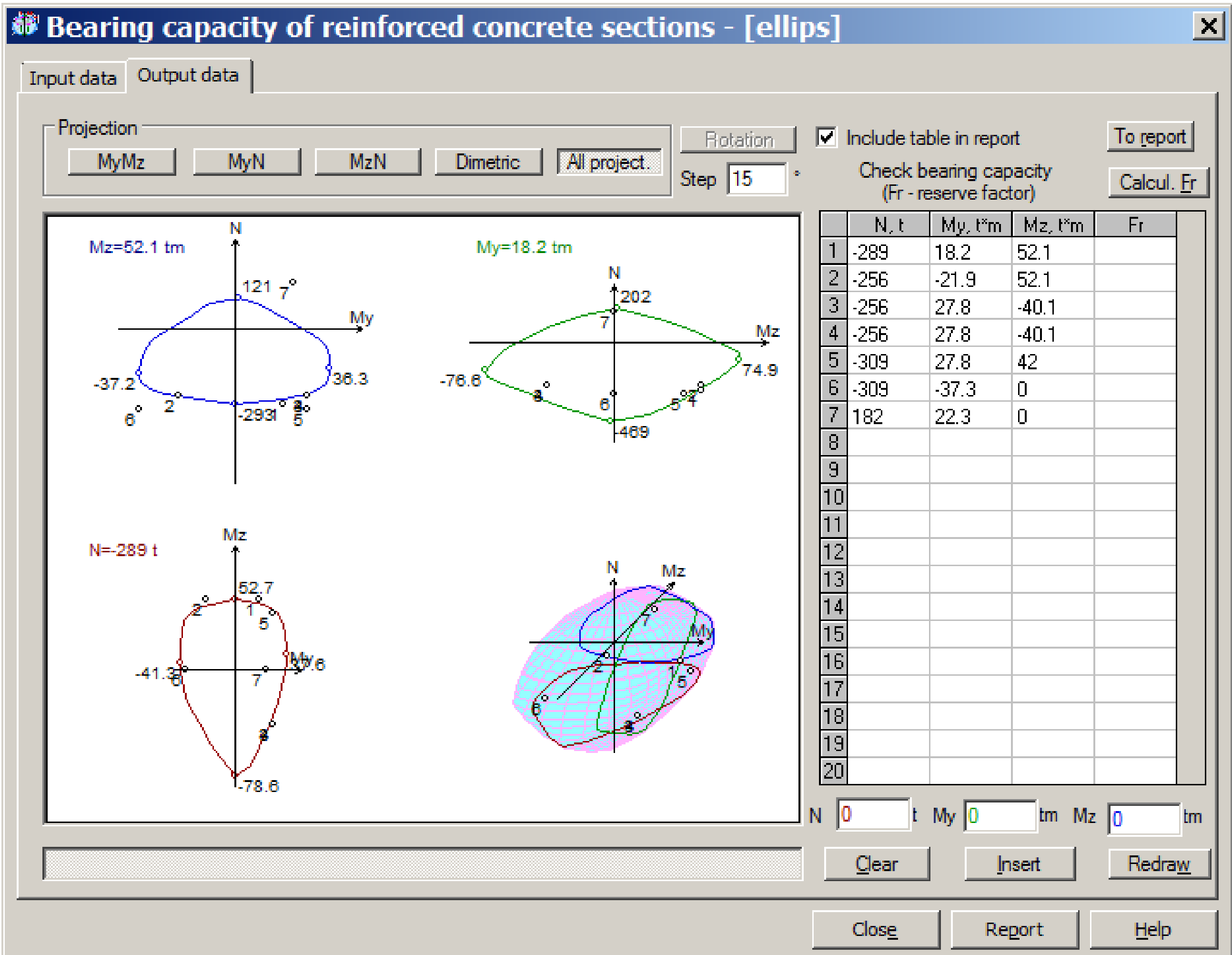


Figure 11.3

To add the specified forces to the table, click **Insert**.
To calculate reserve factors for loads specified in the table, click **Calculate Fr**.
If several rows of the table are filled in, then to display the point with certain forces and contours of planes that pass through it, select appropriate row in the table and click **Redraw**.

In the mode for preview of **Dimetric** projection, it is possible to rotate the image. To do this, click **Rotation** or click [F11]. To cancel this mode, just click **Rotation** once again (to make it not active) or click [Esc]. Step of rotation angle may be defined in appropriate box. The rotation is made in the following way:

- about vertical axis - click to the left (right) from the image (keys [←], [→]);
- about horizontal axis - click above (below) the image (keys [↑], [↓]);
- about axis that is orthogonal to the screen - [CTRL] key + click above (below) the image (keys [HOME], [END]).

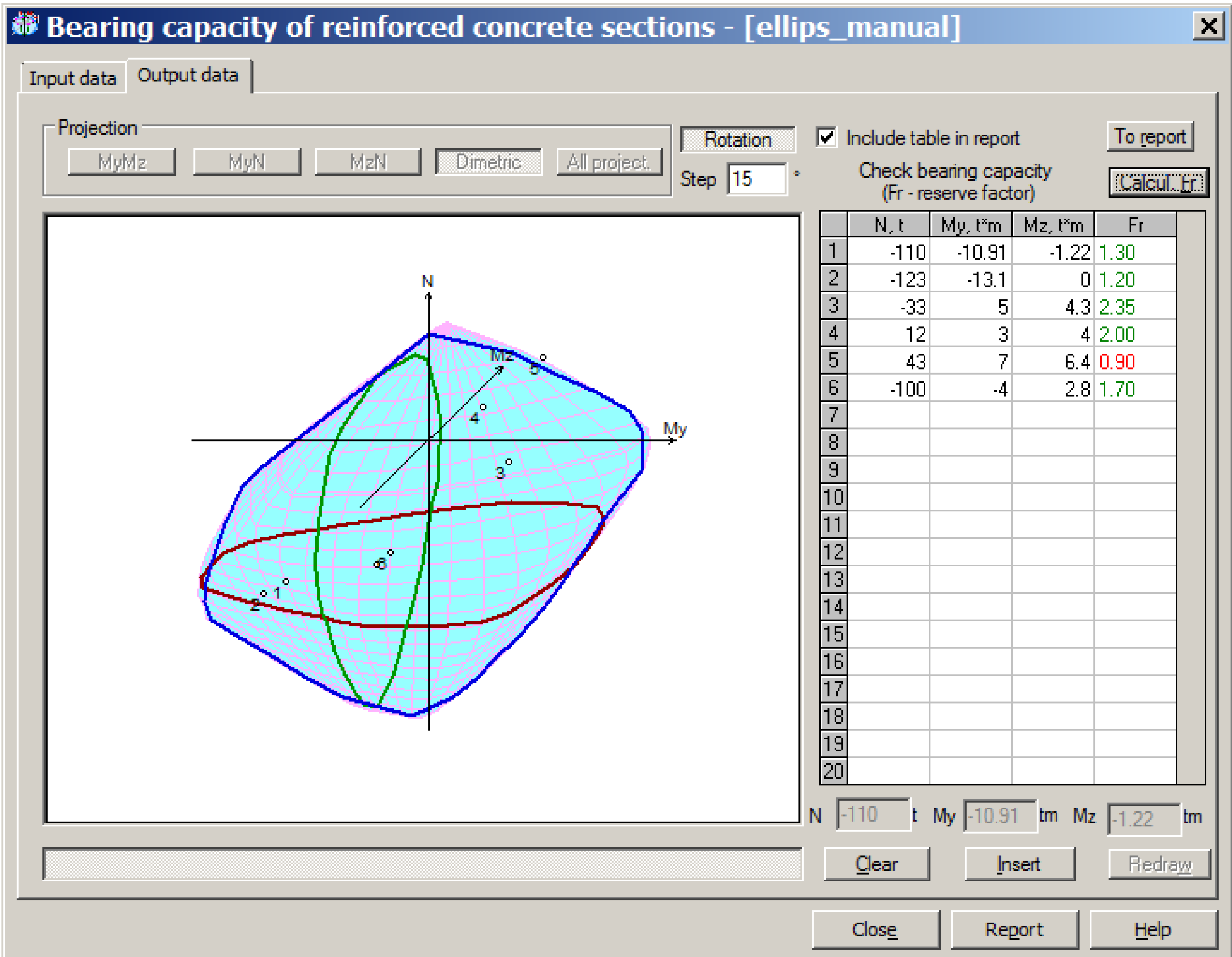


Figure 11.4

When you move the pointer across the projection image, the force values are displayed at the bottom of the dialog box. When you click certain point, the program redraws projections for this point and the values are displayed in the appropriate boxes below the table. To copy these values to the table, click **Insert**. The data will be inserted above the current row.

To delete certain rows or cells, select them with the pointer and click **Clear**. If there is no selected area, then the current cell is cleared. Then all rows that are not empty will be raised and reserve factors are recalculated.

When you change the row of the table (select another row), the values from all rows that are not empty will be displayed in the edit boxes below the table. To redraw projections for these values, click **Redraw**.

Report

To present a report document in HTML format, click **Report**.

To paste the current images of projections to the report file, click **To report**. In this case, if the **Include table in report** option is selected, then the table will be also included into report file. If parameters or type of section are modified, the report will be generated once again. You could save or print the report file.

Sheet piling

Sheet piling (chapter)

The module is meant to analysis of substructure erected according to the 'wall in soil' method. Design model is plane and contains soil, elements of sheet piling and anchorage. It is necessary to define dimensions and properties of soil, dimensions of pit and elevations of pit excavation, loads on soil, dimensions and material properties, sections of sheet piling and anchors as well as pretension forces in anchorage.

In the current version of the program it is allowed to define not more than 4 anchors from every side of sheet piling and not more than 4 elevations for pit excavation.

When the input data is defined, the soil is automatically triangulated with appropriate dividing of sheet piling elements and taking account of anchor locations. The soil is simulated with triangular FE of soil, sheet piling – with bar elements and anchors – with bar elements that take pretension.

Figure 12.1

Calculation is made sequentially by stages. At the first stage, the whole model (without anchors) is analysed on dead weight and defined load. Analysis results for the first stage correspond to initial stress state of design model (stresses in soil prior to commencement of the work).

Then number of stages is determined automatically and depends on defined elevations for pit excavation and elevations for anchors. That is, until the soil is not excavated (disassemblage of soil elements), the anchor could not be installed (assemblage of anchors).

During calculation the displacements are accumulated (by stages) at nodes, stresses – on elements of soil and forces – in elements of sheet piling and anchors.

Sheet piling (module)

Input data

In the ***Input data*** mode, the dialog box contains two tabs: **General model** and **Pit excavation**.

General model (see Fig. 12.2)
On this tab you define parameters of soil. Schematic presentation is modified automatically according to defined data.

In the **Dimensions** area of the dialog box, define dimensions of the pit and neighbouring soil that is considered in analysis:
– pit width;
– depth of pit (pit bottom) measured from ground elevation;
– horizontal dimension of soil to the left from pit;
– horizontal dimension of soil to the right from pit;
– bottom elevation of soil.

Important. Ground elevation is determined automatically when you input elevations for soil layers under **Soil properties for layer**.

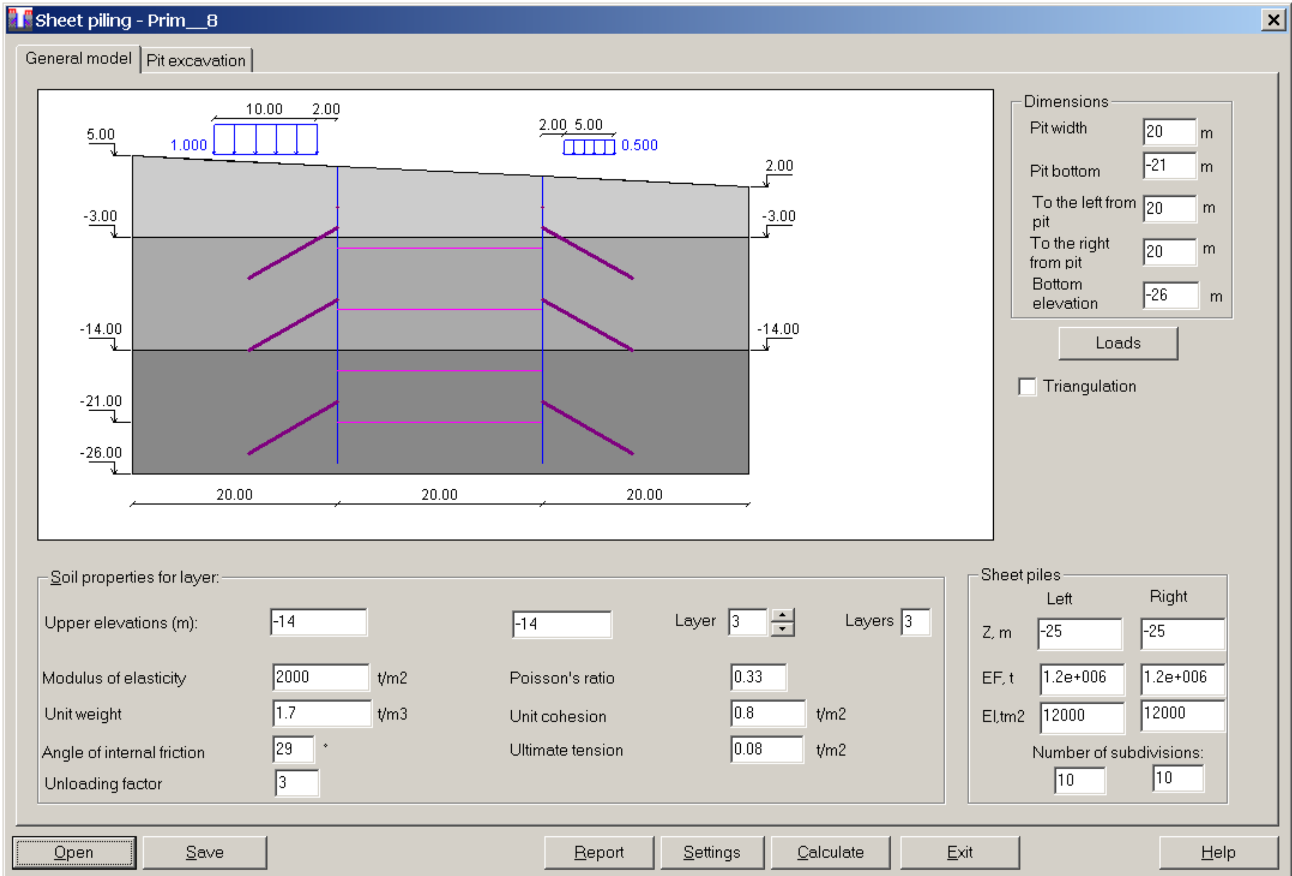


Figure 12.2

In the **Soil properties for layer** area, define total number of soil layers. Soil numbers are defined in the **Layer** box.
For every layer the following data should be defined:
– extreme left and extreme right upper elevations of soil layer;
– modulus of elasticity;
– Poisson's ratio;
– unit weight;
– angle of internal friction;
– unloading factor (correction factor to modulus of elasticity for soil along unloading path);
– unit cohesion;
– ultimate tension (ultimate strength of soil in tension).

In the **Sheet piles** area of the dialog box, define the following values:
– Z – depth of penetration;
– EF – axial stiffness of one running metre;
– EI – flexural stiffness of one running metre.

Division of model (at zones where finite elements of soil and sheet piling are joined) depends on the **number of subdivisions** in appropriate boxes for the left and right sheet piling (by default, it is equal to 10).

When you click **Loads** button, in another dialog box you could define parameters for uniformly distributed loads applied to the soil surface to the left of the left sheet piling and to the right of the right sheet piling (see Fig. 12.3).
In the corresponding boxes, define these values:
– A – distance from pit edge to the load;
– B – load application zone;
– q – uniformly distributed load.

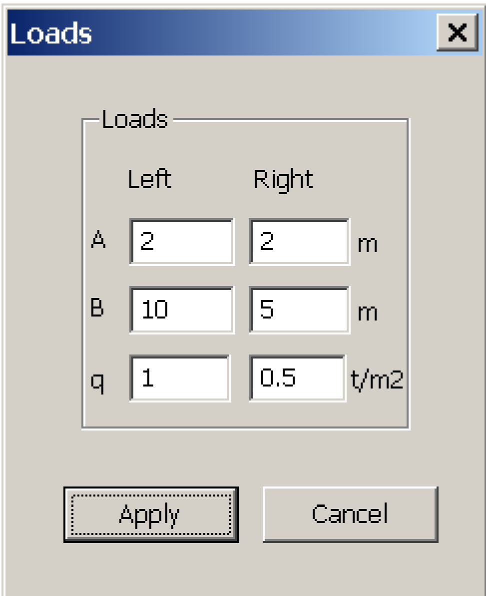


Figure 12.3

Important. To visualize FE design model (before calculation), select the **Triangulation** check box and click **Calculate**. In this case **Calculate** button becomes unavailable. To activate it again, on the **General model** tab, click to clear the **Triangulation** check box.

Pit excavation (see Fig. 12.4)
On this tab it is possible to define number of elevations (not more than 4) for pit excavation. When you define number of elevations, you could input values for the elevations in the appropriate boxes.

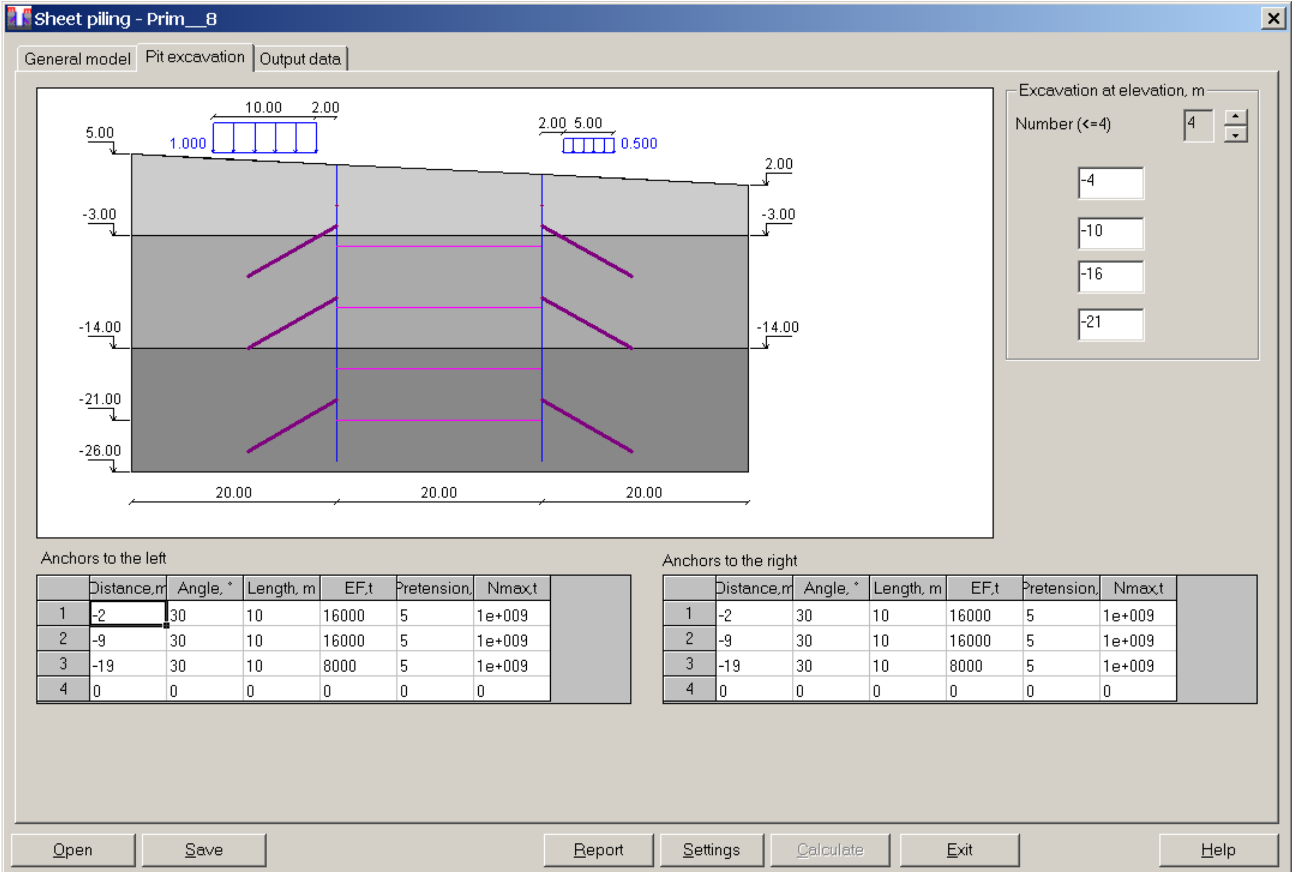


Figure 12.4

In the **Anchors to the left** and **Anchors to the right** tables, define these parameters:
– distance – elevation at which anchor connects the sheet piling;
– slope angle of anchor to horizontal line;
– length of anchor;
– EF – axial stiffness;
– pretension;
– Nmax – max allowed value for pretension.

Click the **Settings** button in order to define parameters for automatic triangulation – area and min allowed angle between triangle sides. By default, the area is taken as equal to 2.5m^2, and angle – 27°.

To save or open the existing problem, use appropriate buttons in the dialog box.

To start calculation, click **Calculate** (if the **Triangulation** check box is not selected).

To determine number of analysis stages

To illustrate the process, the algorithm for determining number of stages automatically is described for design model presented in Fig. 12.4.
Sheet piling of the soil is fixed with three anchors at the left and with two anchors at the right. Pit excavation is made with 4 excavations of soil.
For the specified problem, 8 stages of analysis will be determined automatically:

1 – analysis of whole model (without anchors) on dead weight and load;
2 – excavation of the 1st soil layer;
3 – installation of the upper left anchor;
4 – excavation of the 2nd soil layer;
5 – installation of the middle left and upper right anchors;
6 – excavation of the third soil layer;
7 – installation of the lower left and right anchors;
8 – excavation of the 4th soil layer.
When analysis is complete, the **Output data** tab will be available in the dialog box.

Output data

Output data (see Fig. 12.5) is presented graphically for every stage. When you click appropriate buttons in the **Soil** area, the following data is displayed on the screen:
– deformed shape of soil;
– contour plots of horizontal (X), vertical (Z) and combined (L) displacements;
– contour plots of normal (Nx, Nz) and shear (Txz) stresses;
– state of soil at the final stage.

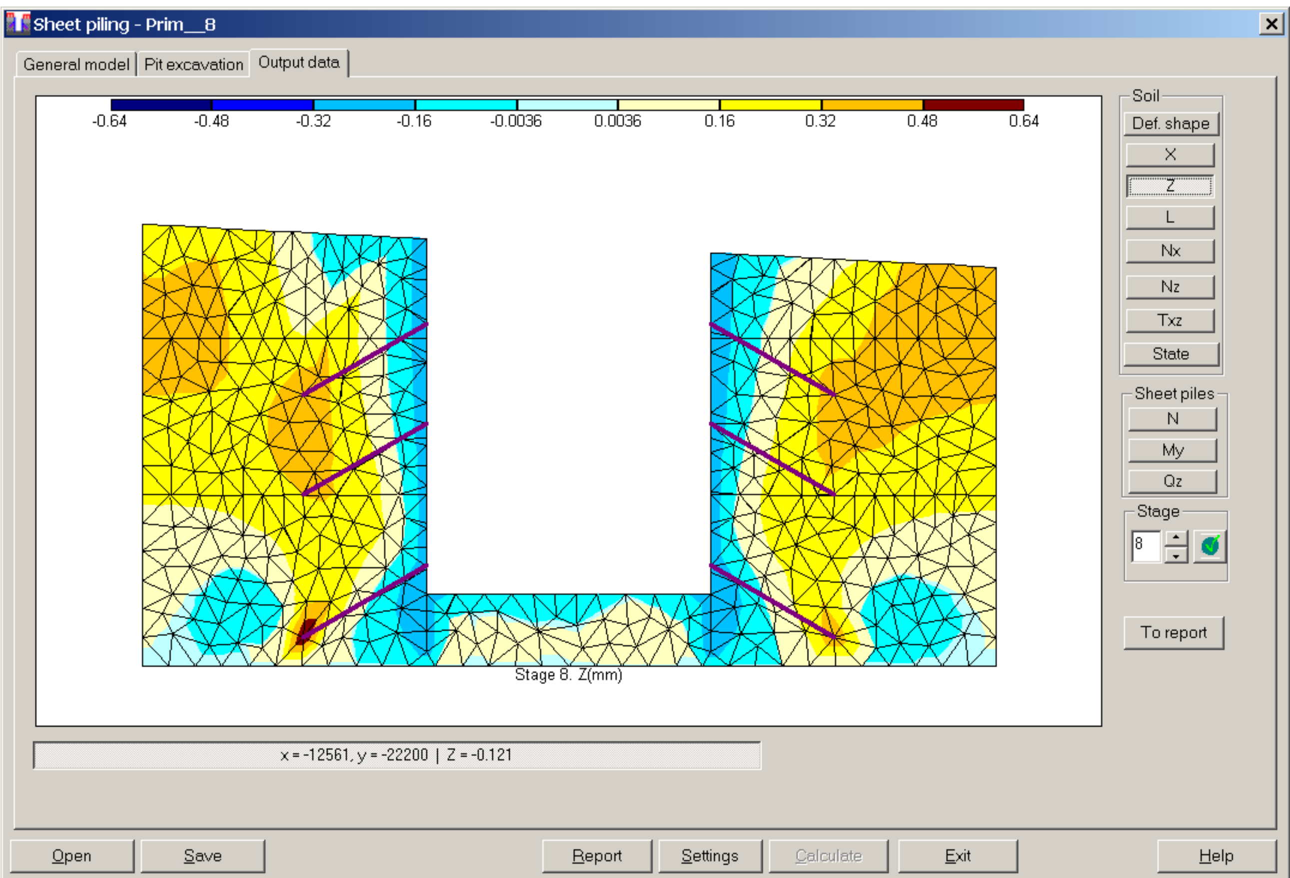


Figure 12.5

To display diagrams with moments, axial and shear forces in sheet piles, click appropriate buttons under the **Sheet piles**. When you click certain sheet pile, you will see separate window where all these diagrams are displayed together.

Forces in anchors are displayed in the box below schematic presentation when you locate the pointer over certain anchor in the mode for axial force diagrams (button **N** is active).

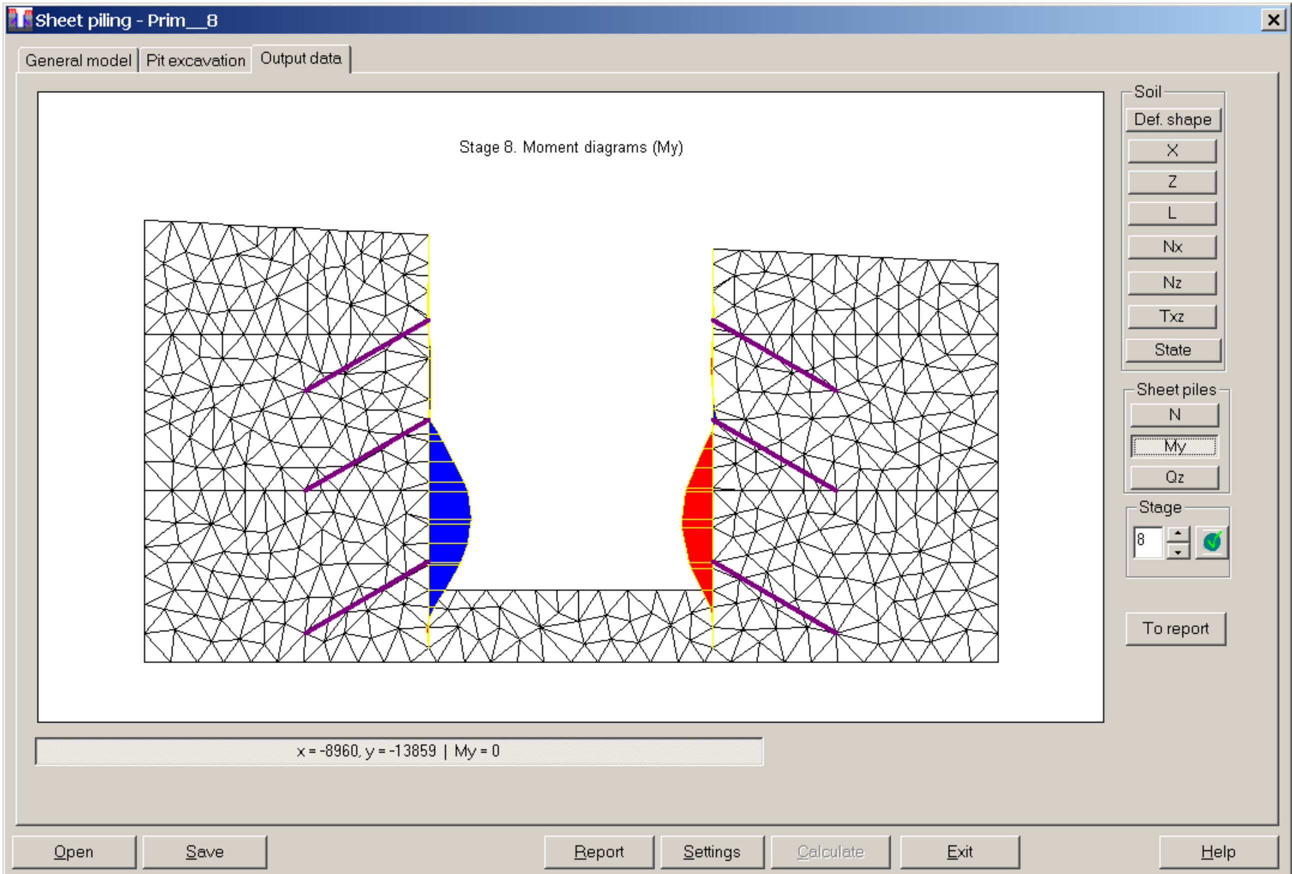


Figure 12.6

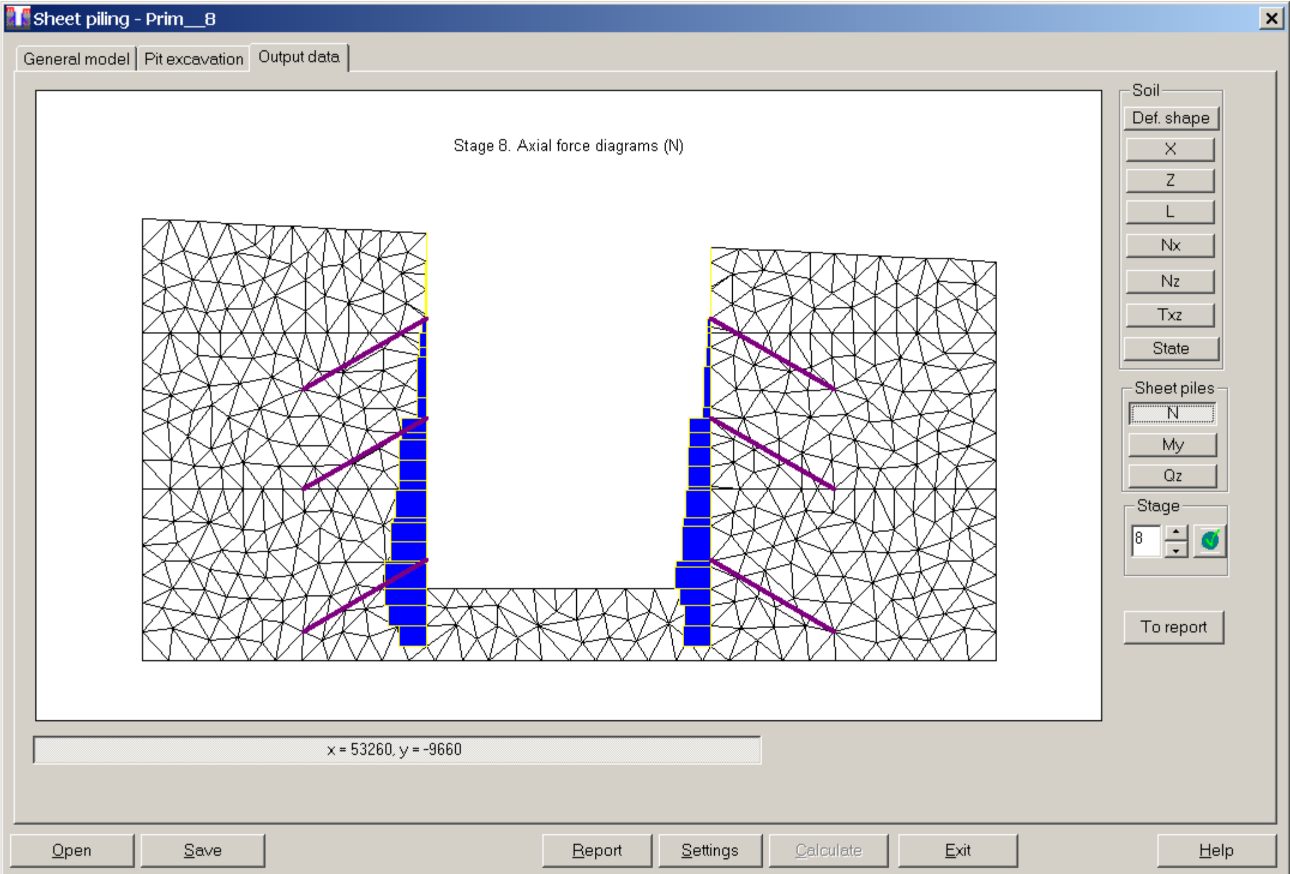


Figure 12.7

To present a report document in HTML format, click **Report**.

Diaphragm

Diaphragm

This chapter contains module that enables you to compute ultimate shear strength of RC diaphragms in static, earthquake and cyclic loads. The program is experimental one.

Strength of RC diaphragm in earthquake loads

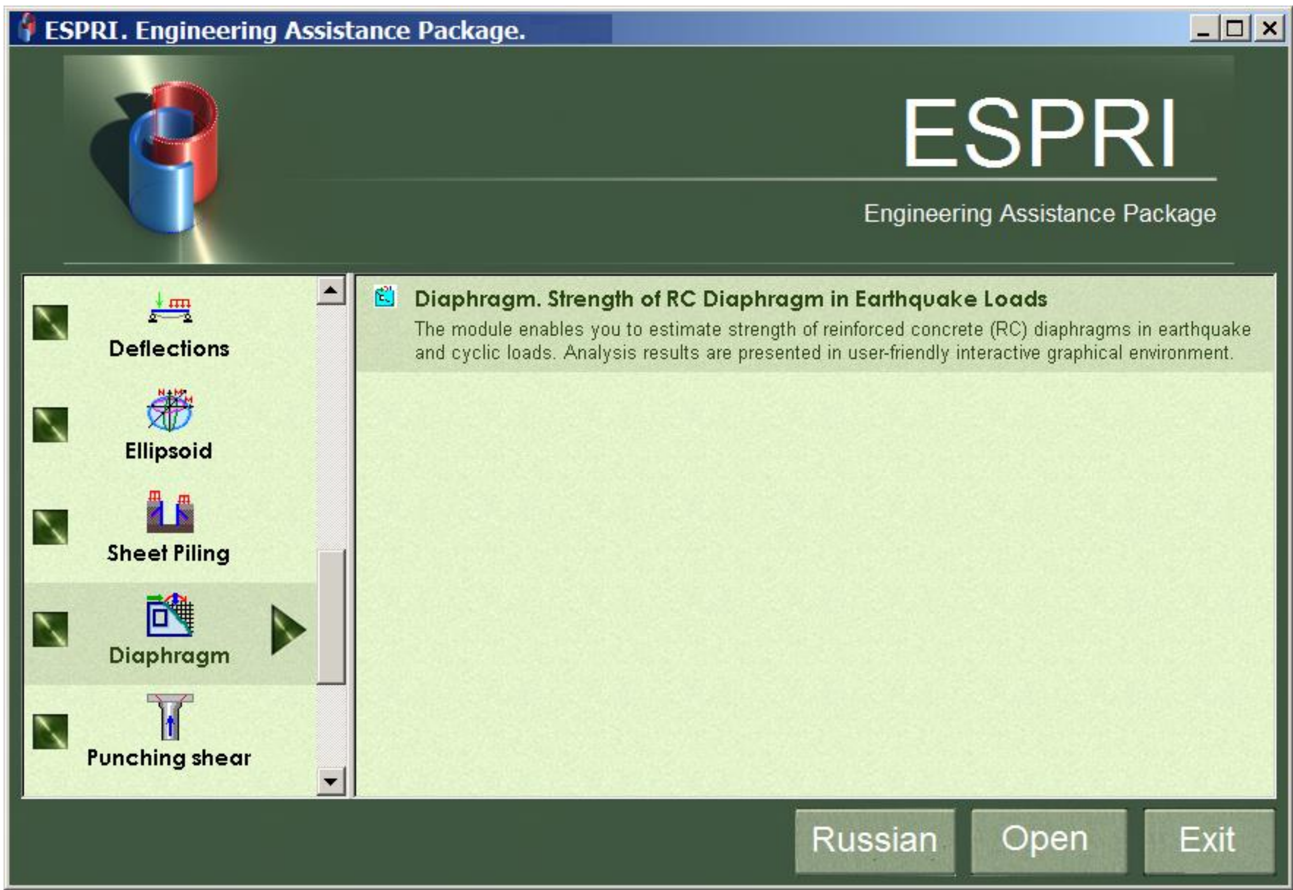


Figure 13.1

Strength of RC diaphragm in earthquake loads

The module enables you to compute ultimate shear strength of RC diaphragms in static, earthquake and cyclic loads. The program is experimental one.

In the output data you will obtain the bearing capacity curve (strength diagram) for the diaphragm. Reserve factor is computed for the specified forces N, Q, M and number of load cycles.

Input data (geometry, reinforcement parameters, partial safety factors, loads) is defined in the dialog box (see Figure 13.2).

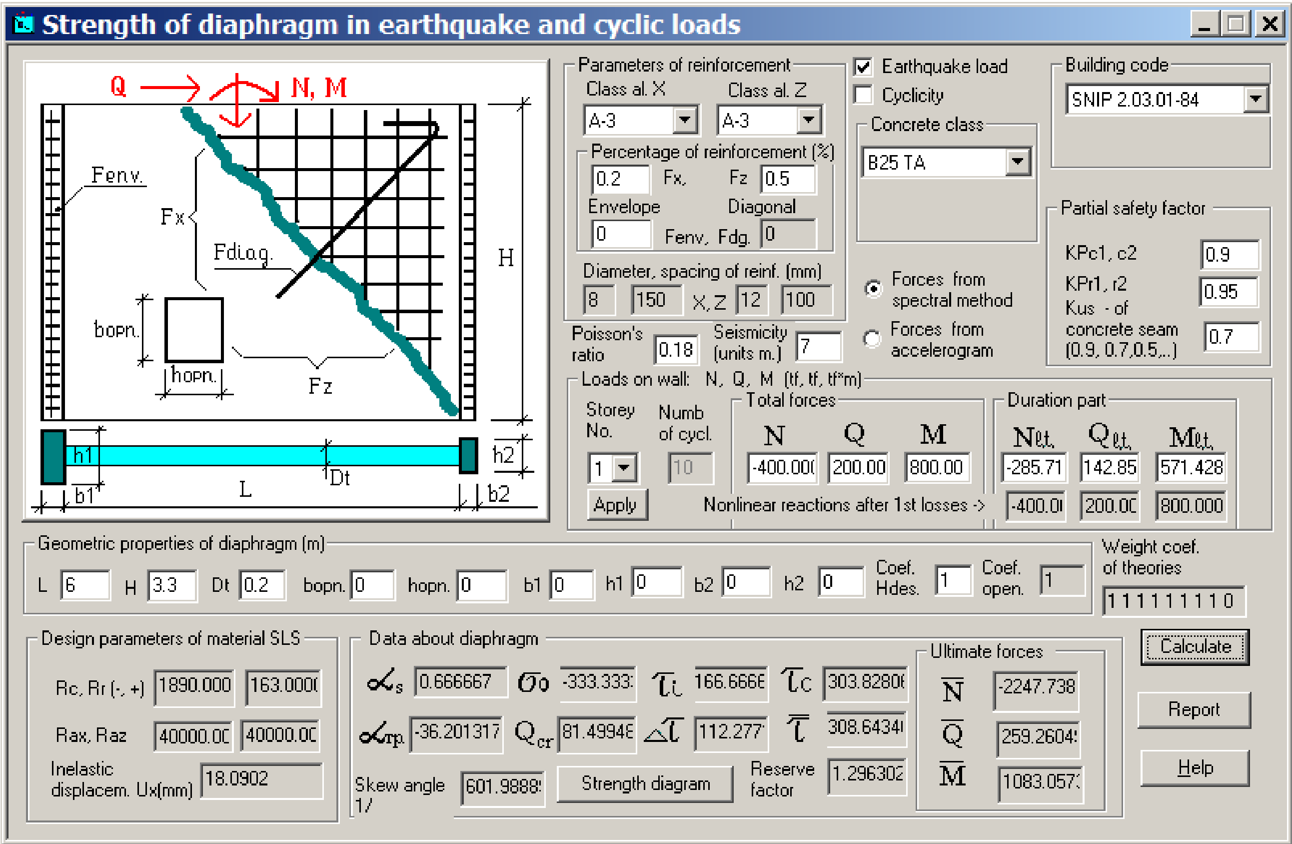


Figure 13.2

Assumptions and methods for calculation

Method of limit equilibrium is implemented in the program together with empirical methods for determining ultimate strength of RC diaphragms with cracks, such as:
– by dependencies of Uniform Building Code (UBC), USA;
– by dependencies of O. Hernandez, F. Barda, T. Tassios,M. Hirosawa, ATC-3, etc.;
– TsNIIEPzhilischa Goskomarhitectura;
– modified method RSN-13-87 (not realized in current version);
– certain provisions of Eurocode 8;
– certain provisions of SNIP 2.03.01-84* and building code CEB-FIP (Euro-International Concrete Committee – International Federation for Prestressing).

Database with bearing capacities for diaphragm in arbitrary combination of loads **Ni, Qi, Mi** is generated as an open system, so you could easily modify it and add new methods. It is also possible to ignore certain methods in calculation with the help of weight factors (not realized in current version).

Input data

Geometry for diaphragm

– **L, H, Dt** – length, height, thickness in m;
– $b_{opn.}$, $h_{opn.}$ – width and height of an opening (not realized in current version);
– **b1, h1; b2,h2** – thickness and length of adjacent columns, walls (when there are adjacent walls, it is recommended to include in calculation no more than **h1, h2 ≈ 3 ÷ 6*Dt**.).

Materials

Concrete – normative values of material strength corresponding to concrete classes B3.5 - B60 by SNIP 2.03.01-84*;
Heavyweight concrete of classes: B3.5 – B1000 by SP 52.03.01-2004 (Russian Federation) ;
Heavyweight concrete of classes: C8\10 – C50\60 by SNB 5.03.01-2004 (Republic of Belarus) ;
Heavyweight concrete of classes: C8\10 – C50\60 by DBN V.2.6.98-2009 (Ukraine) ;
Reinforcement – class A-I ÷ AV, B-I, B-II by SNIP 2.03.01-84* and A240, A400 by DSTU 3760-98 (A500 is not included in the database as this class of reinforcement is not allowed for structures subjected to earthquake loads).

Loads

Loads - **Ni, Qi, Mi** and **N long-term, Q long-term, M long-term** – total values of loads and their long-term part for the *i*-th storey (in this version analysis of one storey is allowed).
Loads – static Kw=1; earthquake (Kw=10 - by default for earthquake); low-cycle Kw <= 50.
If Ni=0, analysis is carried out for the dead weight of the wall.

Notation for loads in output data file:

– Nz – axial compression force for diaphragm (**«-» compression**); (tf)
– Qx – shear force (sign variable); (tf)
– Mxz – bending moment in the plane of diaphragm; (tf*m).
(If Mxz=0, then Mxz is taken as equal to Mxz= Qx* H)
– Sei – design value for seismicity of the site 0, 7, 8, 9 units of magnitude;
– Kw – number of cyclic loads (Kw = 1 - 50);
– KPc12 – partial safety factor for concrete according to serviceability limit state (SLS);
– KPr12 – partial safety factor for reinforcement according to serviceability limit state (SLS);
– KUS – partial safety factor for diaphragm, concrete joint, etc.

Recommendations: how to assign values

to take account of cyclicity of loads, reduce shear strength along construction joint according to [1] we recommend:
a) KUS_a=0.9 for special processing of joint, otherwise KUS_a=0.7;
b) for design seismicity: 7, 8, 9 units of magnitude - respectively:
KUS_b=0.9, 0.75, 0.6; without KUS_b=1.0;
c) in the presence of construction joint in the section - in seismicity 7, 8, 9
KUS_c = 0.7, 0.6, 0.5.
d) for hollow and ribbed elements: Kus_d=0.9;

In the presence of several above-mentioned factors in groups a), b), c), d),... the total coefficient KUS is computed as:

KUS = KUS_a * KUS_b * KUS_c * KUS _d * ....*KUS _i.

Output data (repeated for number of storeys):

Ni – axial force varied from 0 to Nlim (ultimate);
eo – eccentricity of longitudinal force Ni (m);
TUi – actual shear stress (tf/m2);
TUr – design value for strength of diaphragm in shear;
TU1 – strength by Uniform Building Code of USA (UBC) 1973.
TU2 – --/-- by method of F. Barda, J. Hanson, W. Corley, 1977.
TU3 – --/-- by method of O. Hernandez, M. Zermeno, 1980.
TU4 – --/-- by method of T. Tassios, J. Lefas, S. Lulurgas, 1982.
TU5 – --/-- by method of M. Hirosawa (ver. 1.05.0 and later), 1980.
TU6 – --/-- by method of ATC-3 (Seismic Evaluation and Retrofit of Concrete Buildings), 1978.
TU7 – --/-- by method of NIISP, NIISK (ver.4.0 or later), 2010.
TU10 – --/-- by method of Eurocode 8 (ver.4.0 or later), 2002.
Dtau – reduction of reaction in cyclic load $\Delta\tau_n$
TUC – mean value of diaphragm strength: TUc=(TU1+..+Tui8)/i;
TUR – design min strength of diaphragm in shear with account of cyclicity (reduction of reaction);
TUR = (TUc+TU1+..+TUi -Tмax-Tмin)/ (i-1)- Dtau;
ALFA – coefficient for shear influence α_s = Mi/(Qi*L) ;
Nmax – max axial force in uniaxial compression;
Nlim – ultimate axial compression force when α_s ;
Qlim – ultimate shear force with account of cyclicity, completeness of stress diagram in concrete and service conditions for diaphragm, construction joint, etc.;
Mlim – ultimate bending moment Mxz, tf*m;
Qcr – shear force at crack origin, t;
Acr – approximate width of crack propagation for the main crack, mm;
TETA – max skew angle of diaphragm, rad;
Ux – max displacement of diaphragm top along the X-axis, mm;
SO – actual average axial stress (NZ)
SGstr. – max compressive stress in concrete strip;
Psi_c – relative length of compressed section;
SG1 – principal tensile stress (1st principal plane);
SG2 – principal compressive stress (2nd principal plane);
ALFAT – slope angle of the main crack in degrees (positive value if clockwise);
SG_max,min – max and min fibre stress;
Кreserve. – reserve factor for strength of diaphragm;
Output data is presented in the text file *Stena_Sei.htm*.

Design strength of diaphragm in shear with account of cyclicity is determined according to dependencies below.

Main dependencies for strength of wall

Designation in output data	Dependence
TU1	1. by Uniform Building Code of USA (UBC) [2] : $\tau_u = \tau_c + \tau_h \tau_{sy};$ where $\tau_c = \min\begin{cases} 0{,}25\left(\sqrt{f_c} + \sigma_0\right) \\ 0{,}05\sqrt{f_c} + \left(0{,}10\sqrt{f_c} + 0{,}2\sigma_0\right)/(a_s - 0{,}5) \end{cases}$
TU2	2. by dependences of Barda F., Hanson J., Corley W. [2, 9]: $\tau_u = 0{,}2\sqrt{f_{cc}}(3{,}2 - a_s) + (1/4)\sigma_0 + \rho_v f_{sy};$
TU3	3. by dependences of Hernandez O.B., Zermeno M.E. [2, 10] : $\tau_u = \tau_c + \tau_s;$ $\tau_c = \sqrt{\tau_{oc}(\tau_{oc} + \sigma_0)};\ \ \sigma_0 \le 5\tau_{oc};$ $\tau_s = \rho_h f_{sy}$ *if* $a_s > 1{,}25;$ $\tau_{oc} = \left(0{,}5 - 0{,}1a^2_s\right)\sqrt{f_c} > 0{,}16\sqrt{f_c},$ $\tau_s = \rho_h\left(a_s - \frac{1}{4}\right)f_{sx} + \rho_v\left(\frac{5}{4} - a_s\right)f_{sy}$ *if* $0{,}25 \le a_s \le 1{,}25;$ $\tau_s = \rho_v f_{sy}$ *if* $a_s < 0{,}25;$
TU4	4. by dependences of Tassios T., Lefas J., Lulurgas S. [2, 11] : $\tau_u = 0{,}5\left(1 - 0{,}1a^2_s\right)\left[\sqrt{f_c} + \sigma_0 + \rho_h f_{sy}\right]$ *if* $a_s > 1.25;$ $\tau_u = 0{,}6\sqrt{f_c} + \rho_h f_{sy} + \frac{1}{4}\sigma_0$ *if* $a_s < 1.25$
TU5	5. by dependences of Hirosawa M. $\tau_u = 0{,}08\left[\frac{0{,}0679\,(100\rho l)^{(0,23)}(f_c + 180)}{\sqrt{a_s + 0{,}12}} + 2{,}7\sqrt{\rho_h f_{sy}} + 0{,}1\sigma_0\right];$
TU6	6. by dependences ATC-3 (Seismic Evaluation and Retrofit of Concrete Buildings) (USA)

	$\tau_u = 0{,}165\sqrt{f_c} + 0.25\sigma_0 + \rho_h f_{sy} \le 0{,}83\sqrt{f_c};$
TU7	7. by dependences of NIISK, NIISP (Ukraine) [12, 13] $\left(\frac{N_i}{N_{pr}}\right)^2 + \left(\frac{Q_i}{Q_{pr}}\right)^2 - 1 = 0;\quad \tau_u = \tau_c + \tau_s;$ *if* $a_s \ge \sqrt{2}, \sigma_0 \le 2f_c/3;$ $\tau_c = (1 - 0.2a^2{}_s)\left[\sqrt{(f_c} + \frac{1}{4}\sigma_0)\right] \ge 0.2\sqrt{f_c};$ $\tau_s = 0.195\rho_h f_{sx} + 0.8p_v f_{sy} + 0.05\rho_l f_{sy};$ *if* $\rho_h; \rho_v \le 0.005(0.5\%); \rho_l \le 0.035(3.5\%).$ *otherwise if* $a_s < \sqrt{2}$ $\tau_c = 0{,}25\sqrt{f_c} + \frac{1}{4}\sigma_0 \ge 0.25\sqrt{f_c}$ *if* $\sigma_0 \le 0.5f_c;$ $\tau_s = \rho_h\left(a_s - \frac{1}{3}\right)f_{sx} + \rho_v(1.25 - a_s)f_{sy} + 0.05\rho_l f_{sy};$ *if* $0.333 \ge a_s \le \sqrt{2};$ $\tau_s = \frac{1}{3}\rho_h f_{sx} + \frac{2}{3}\rho_v\ f_{sy};$ *if* $a_s < 0.333;$ $Q_{pr} = \gamma_b\,\overline{\tau_u}{*}\,\omega{*}\,A_b;$
TU8	8. by modified dependencies RSN-13-87 for Moldova [12, 19] $Q_u = Q_{bc} + \sum_1^n R_{swi}\ A_{swi} \le \gamma_{kus}\,Q_r$
TU10	10. by dependences of Eurocode 8 $V_R = \left[\begin{matrix}\left(\frac{L-x}{2a_s}\right)\min(N, 0.55A_c f_c) + (1 - 0.05\min(5, \mu^{pl})) * \\ [0.16\max(0.5, 100p_v)(1 - 0.16\min(5, \alpha_s/L))\sqrt{f_c}A_c + V_w)]\end{matrix}\right] / \gamma_{el};$

	$$V_{R\max} = \begin{bmatrix} (1-0.06\min(5,\mu^{pl})\left(1+1.8\min\left(0.15,\frac{N}{A_c f_c}\right)\right) \\ (1+0.25\max(1.75,100\,p_v)[1-0.2\min(2,\alpha_s/L)\sqrt{f_c}d\ \ Z] \end{bmatrix} / \gamma_{el};$$ *if* $\alpha_s/L \leq 2.0$ $$V_{R\max} = 4/7 \begin{bmatrix} (1-0.02\min(5,\mu^{pl})\left(1+1.35\min\left(0.15,\frac{N}{A_c f_c}\right)\right) \\ (1+0.45(1.75,100\,p_v))\sqrt{\min(40,f_c)}\delta\ Z\sin 2\alpha \end{bmatrix} / \gamma_{el}$$ $Q_{pr} = V_{R\max}$; α - *angle between diagonal of diaphragm and axis of column:* *tan*(α)= 0.5*L*/(*Mi*/*Qi*)
	where
	τ_c – strength of concrete strip between cracks, flexural and shear strength and dowel effect of concrete with cracks;
	τ_s – strength of reinforcement in diaphragm with cracks;
ALFA	α_s – coefficient for influence of shear α_s =M \ (Q * L) ;
RBC	f_c – concrete strength in compression according to serviceability limit state, (MPa) ;
RBR	f_r – concrete strength in tension according to serviceability limit state, (MPa);
RAX	f_{sx}, f_{sy} – normative yield strength for horizontal / vertical reinforcement;
RAZ	f_{tx}, f_{ty} – normative fracture limit for horizontal / vertical reinforcement;
FAX	ρ_h – coefficient for horizontal reinforcement (%Fx/100);
FAZ	ρ_v – coefficient for vertical reinforcement (%Fz/100);
FOK	ρl – coefficient for envelope reinforcement (%Fok/100);
SO	σ_0 – average stress;
TUR	τ_u – strength of wall in shear, (MPa);

	V_{Rmax} – max strength of wall in shear in cyclic loads according to Eurocode 8;
	V_w – portion of transverse reinforcement in strength in shear: Vw= ρh δZ fsx;
	γ_{us} – partial safety factor of concrete joint;
	Z – lever arm for rectangular section: Z =0.8L

According to dependences of Tassios T., Lefas J., Lulurgas S. [9, 18], stress at crack origin in (MPa) when α_s >1 will be:

$$\tau_{cr} = 0.3(2.6 - as^2)\,[0.1 + 20pv)\sqrt{f_c} + 2pv^2\,10^4]\,,\ \ \textit{otherwise}$$

$$\tau_{cr} = 0.5(1.5 - as^2)\,[0.3 + 20pv)\sqrt{f_c} + 3pv^2\,10^4].$$

Cyclicity of earthquake loads is taken into account by idealized model of hysteresis of reinforced concrete walls (see Figure 13.4).

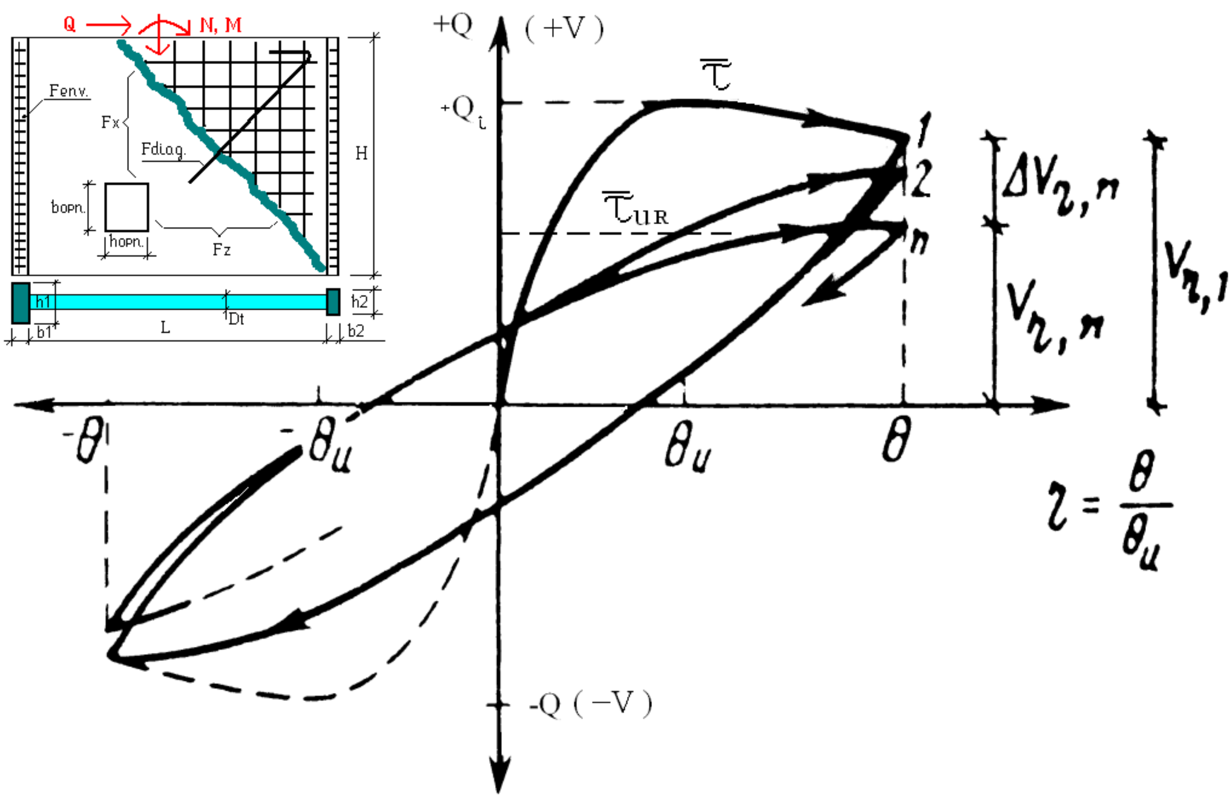


Fig. 13.3 Parameters of deformation in wall:

θ – angular deformation; $\bar{\tau}_{ur}$ – design min strength of diaphragm in shear with account of cyclicity;

τ_n / τ_1 – reduction of reaction after **n**–cycles with amplitude $\pm\theta$;

ΔV – degradation of reaction $(\Delta\tau_n)$.

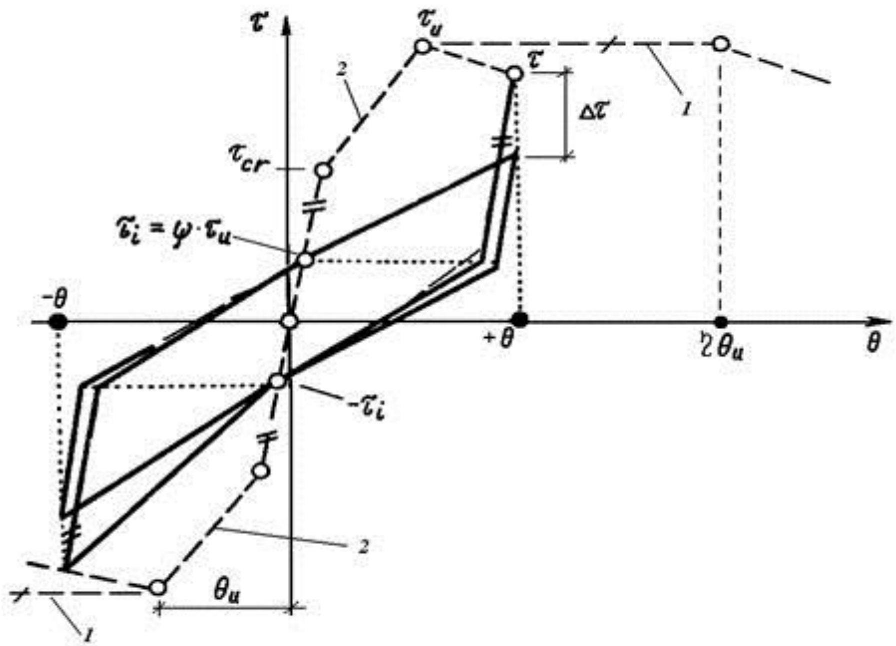


Fig. 13.4 Idealized model of hysteresis.
1 – skeleton curve of cyclic load in bending;
2 – skeleton curve of cyclic load in shear;
τ_{cr} – average shear stress at crack origin;
$\Delta\tau_n$ – degradation of shear stress [DTAU] for **n**-cycles of loading:
$\Delta\tau_n = \beta_d \cdot \sqrt[4]{n \cdot \tau_i}$, where β_d – coefficient of reaction degradation according to [9].

Strength diagram for the diaphragm along **N~Q** and **N~M** according to above-mentioned criteria is presented graphically (see Fig.13.5).

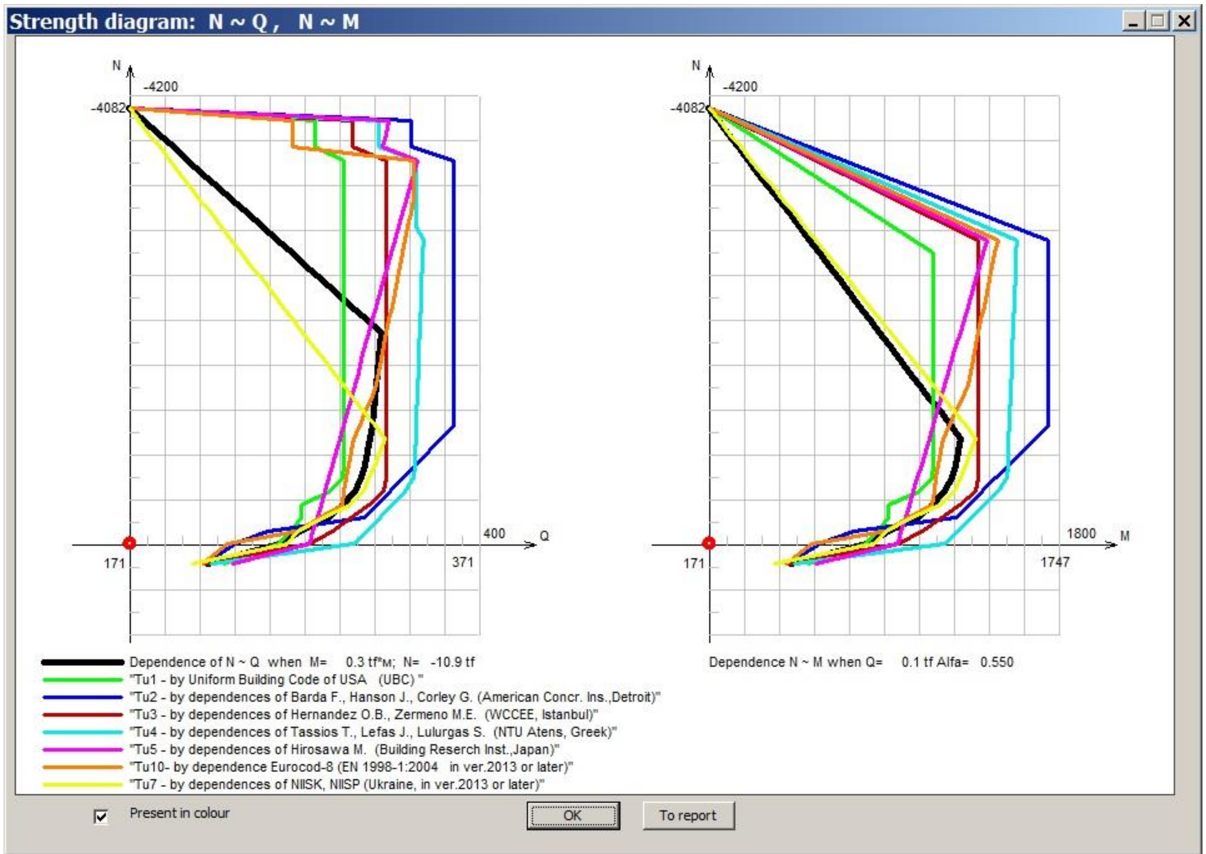


Fig. 13.5 Strength diagram for the diaphragm along **N~Q** and **N~M**.

Punching shear

Punching shear

This chapter contains modules for punching shear analysis of reinforced concrete slabs along arbitrary and rectangular contour.

The following building codes are supported:
 – SNIP 2.01.03-84*;
 – SNIP 52-01-03;
 – provisions of scientific and technical report made by GUP NIIZHB (contract No. 709 of October 01, 2002);
 – Eurocode 2.

The chapter contains the following modules:
Punching shear for arbitrary contour
Punching shear for rectangular contour
Punching shear analysis (Eurocode)
Punching shear analysis (Belarus)
Punching shear for circular contour (Eurocode)

Figure 14.1

Punching shear for arbitrary contour

This module enables you to carry out punching shear analysis of floor slabs and foundation slabs.

Shape of the punching shear contour is determined automatically depending on the type of column section. The following sections of columns are available: circular, rectangular, T-section, angle section and cross section. Dimensions of punching shear contour are computed depending on specified slope angle of punching shear pyramid (by default, this angle is equal to 45 degrees).

Open contour (there is an opening in the slab or the column is located at the slab edge) is generated by cutting off certain part from the contour. Cut-off part is simulated with straight line that passes through the point with specified coordinates and at specified slope angle to the horizontal X-axis.

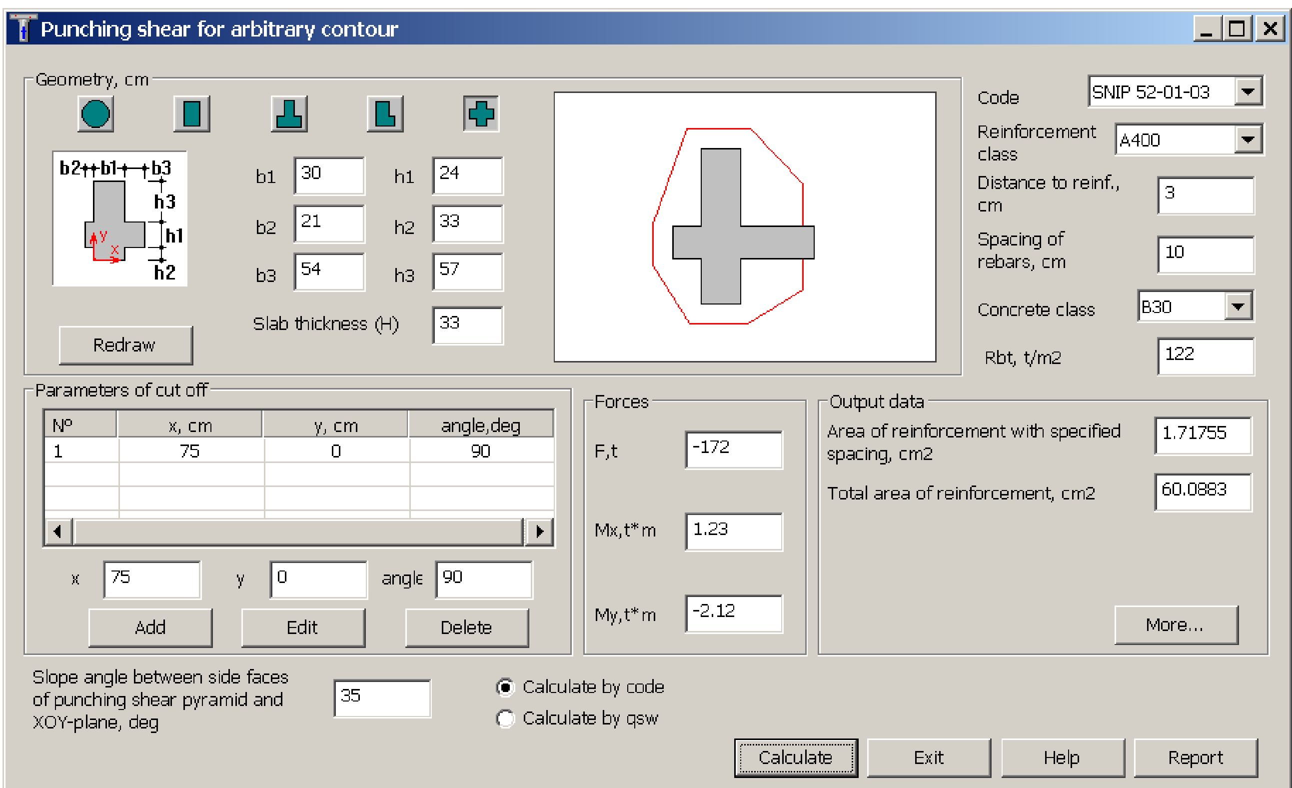


Figure 14.2

Punching shear contour is computed for concentrated load F and concentrated moments Mx and My applied at the gravity centre of column. The following ***sign convention*** is accepted:

– positive load F is directed upwards (along the Z-axis),

– positive moments Mx and My are located in the XoZ and YoZ-planes respectively and are directed anti-clockwise if you look from the end of the Y and X-axes.

 www.liraland.com

Section moduli of contour that should be considered in calculation are determined depending on direction of specified moments Mx and My.

The following provisions are considered in calculation:
1) all calculations are made relative to central (but in general case, not principal) axes of contour X and Y; these axes pass through its gravity centre and are parallel to faces of section for (non-circular) column;
2) specified concentrated load F is applied at the gravity centre of the column;
3) moment Mx acts along the X-axis (about the Y-axis) while moment My – along the Y-axis (about the X-axis);
4) concentrated load cannot be equal to zero;
5) both moments may be equal to zero;
6) positive eccentricities are located at positive values of appropriate coordinates;
7) if gravity centres of column and punching shear contour do not coincide, design momenta with account of sign convention are computed;
8) bearing capacity of concrete and reinforcement in punching shear zone is checked by absolute values of the specified load and design moments.

Reinforcement is computed if the following conditions are ***not*** satisfied:

1) $F \le F_{b,ult}$;

2) $\dfrac{F}{F_{b,ult}} + \dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} \le 1;$

3) $\dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} \le \dfrac{F}{F_{b,ult}}.$

The check below is made in calculation:

$$\frac{F}{F_{b,ult} + F_{sw,ult}} + \frac{M_x}{M_{bx,ult} + M_{sw,x,ult}} + \frac{M_y}{M_{by,ult} + M_{sw,y,ult}} \le 1$$

, where

$F_{b,ult}$ – ultimate axial force that concrete of design cross-section of slab may take;
$M_{bx,ult}$ – ultimate moment in concrete about the Y-axis (in plane that passes through the X-axis);
$M_{by,ult}$ – ultimate moment in concrete about the X-axis (in plane that passes through the Y-axis);
$F_{sw,ult}$ – ultimate axial force in reinforcement;
$M_{sw,ult}$ – ultimate moment in reinforcement about the Y-axis (in plane that passes through the X-axis);
$M_{sw,ult}$ – ultimate moment in reinforcement about the X-axis (in plane that passes through the Y-axis).

Slab thickness is not limited.

Check for moments that act on open contour is made with min values of Wx and Wy (section moduli for contour).

Area of transverse reinforcement with specified spacing is computed if the following condition is satisfied:

$0 \le F_{sw, ult} \le F_{b, ult}$.

If $F_{sw, ult} > F_{b, ult}$, then the message '*Ultimate force in reinforcement exceeds ultimate force in concrete'* is displayed and reinforcement is not computed.

To display all temporary and final data for the calculation, in the **Output data** area of the dialog box, click **More**.

To present a report document in HTML format, click **Report**.

Input data

To determine punching shear contour, in the **Geometry** area of the dialog box, define the type of column section, its dimensions and thickness of floor slab.

To convert closed contour into an open one, in the **Parameters of cut off** area, define coordinates **X, Y** and rotation **angle** for cut off line that passes through the contour and click **Add**. Specified parameters will be displayed in the table. Coordinates for the point are defined in the accepted coordinate system presented on schematic presentation of column section.

To edit parameters of cut off: select appropriate row in the **Parameters of cut off** table, then specify new parameters and click **Edit**.

To delete specified parameters of cut off: select appropriate row in the **Parameters of cut off** table and click **Delete**. In this case, if the row in the table is not selected, the last-defined row will be deleted.

In the upper right corner of the dialog box, specify materials: define class of concrete for the slab or **Rbt** design tensile strength of material of a slab. Define class of reinforcement or **Rsw** design tensile strength of transverse reinforcement.

In the appropriate boxes define concrete cover (distance to reinforcement) and spacing for tensile reinforcement in a slab.

In the **Forces** area, define loads on column - concentrated load **F**, moments **Mx** and **My**.

When input data is defined, click **Calculate**.

Output data

In the **Output data** area of the dialog box you will see values for area of transverse reinforcement with specified spacing and total area of reinforcement

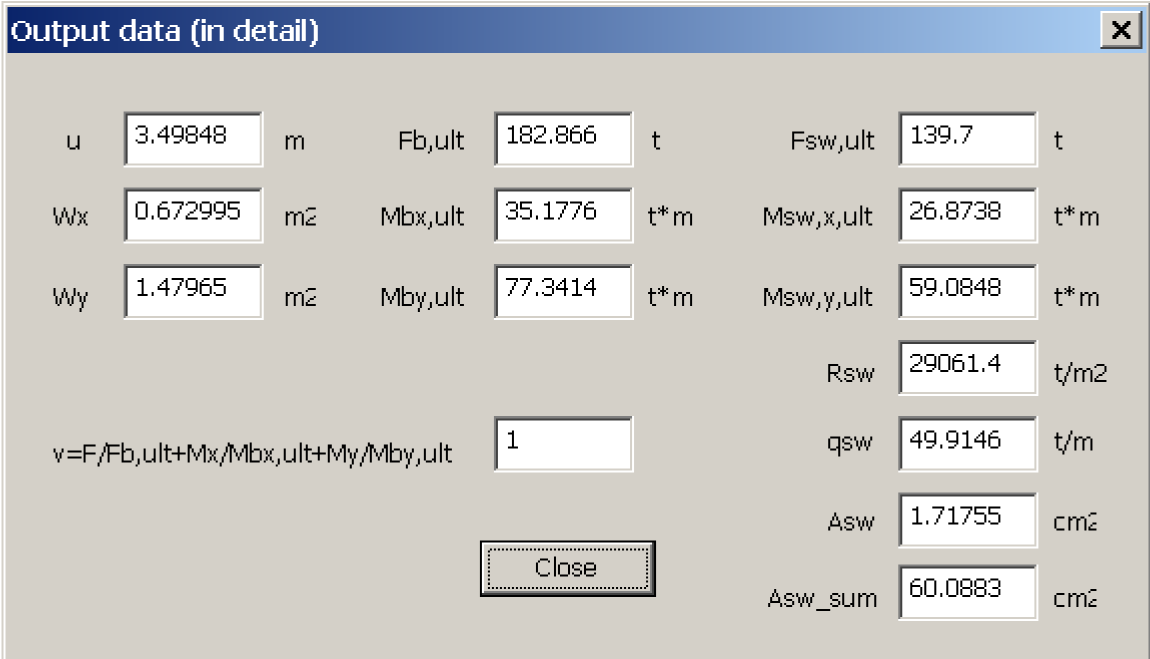


Figure 14.3

If $F_{sw,\,ult} > F_{b,ult}$, then the message '*Ultimate force in reinforcement exceeds ultimate force in concrete'* is displayed.

To display all temporary and final data for the calculation, in the **Output data** area of the dialog box, click **More**.

To present a report document in HTML format, click **Report**.

Punching shear for rectangular contour

This module enables you to carry out punching shear analysis of floor slabs and foundation slabs with concentrated load and concentrated moments.

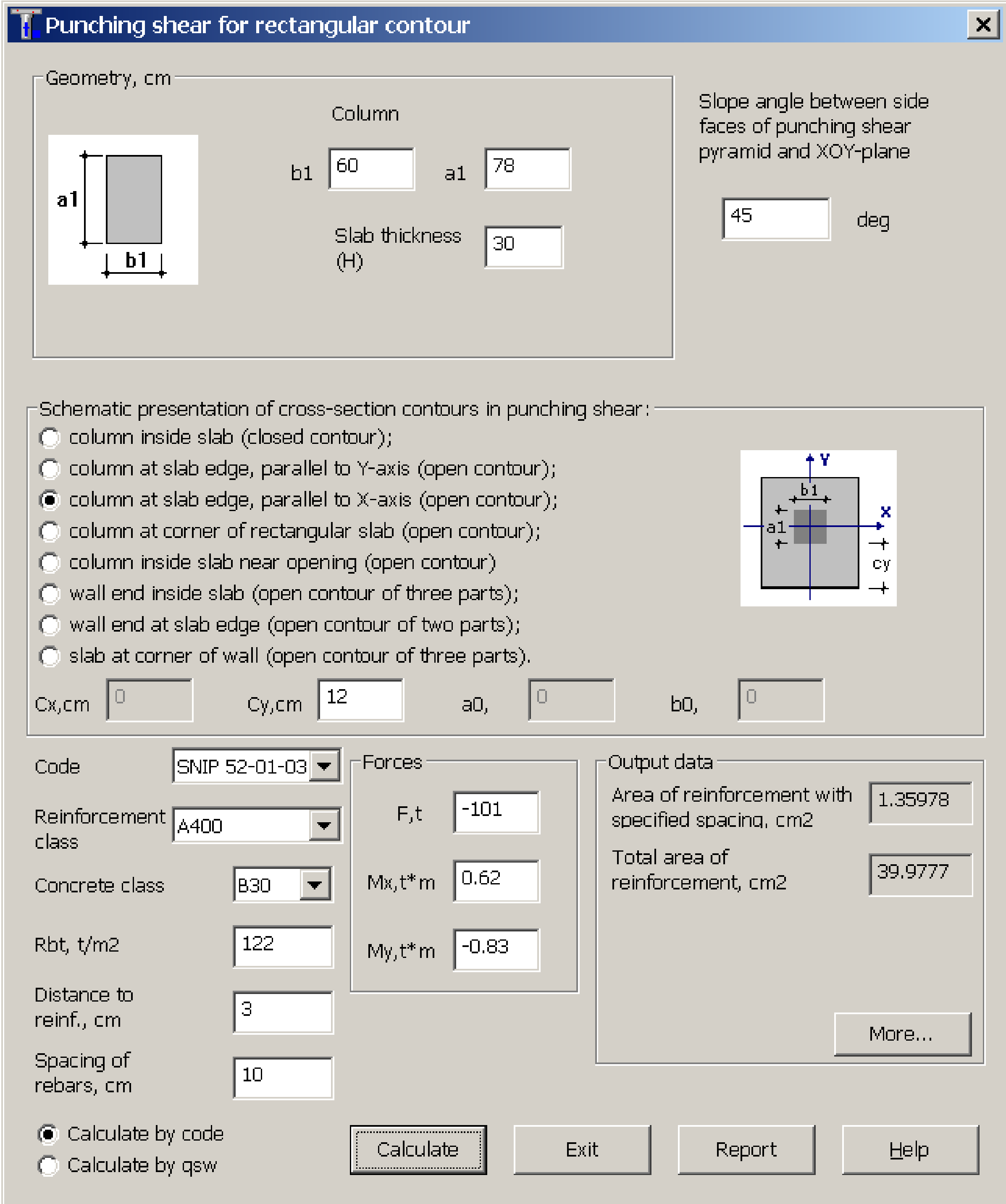


Figure 14.4

The following design cases are considered:

1. Interaction of slab with rectangular column:
 - column inside slab (closed contour);
 - column at slab edge that is parallel to the Y-axis (open contour);
 - column at slab edge that is parallel to the X-axis (open contour);
 - column at corner of rectangular slab (open contour);

- column inside slab near opening (open contour).

2. Interaction of slab with wall:
- wall end inside slab (open contour of three parts);
- wall end at slab edge (open contour of two parts);
- slab at corner of wall (open contour of three parts).

Parameters of punching shear contour are computed depending on specified slope angle of punching shear pyramid (by default, this angle is equal to 45 degrees).

Concentrated load ***cannot be*** equal to zero.
Both moments ***may be*** equal to zero.

Values of the specified moments are modified automatically depending on displacement of contour centre relative to the gravity centre of the column.

The following ***sign convention*** is accepted for loads:
– positive load F is directed upwards (along the Z-axis),
– positive moments Mx and My are located in the XoZ and YoZ-planes respectively and are directed anti-clockwise if you look from the end of the Y and X-axes.
Positive eccentricities are located at positive values of appropriate coordinates.

Section moduli of contour (Wx and Wy) that should be considered in calculation are determined depending on the sign of specified moments Mx and My.

In calculation of open contour, slab edges are located opposite to direction of the X and Y-axes.

Important. If column is located inside the slab near the opening, it is considered that the opening generates gaps a_0 and b_0 in punching shear contour. Gaps are always located on the right and upper sides of contour. Lengths of gaps and distances from gaps to column axes are defined by the user manually.
a_0 – gap length on right side of contour;
a_2 – distance (with appropriate sign) from the *lower end* of gap a_0 to the Y-axis;
b_0 – gap length on upper side of contour;
b_2 – distance (with appropriate sign) from the *left end* of gap b_0 to the X-axis.

For calculation by SNIP 2.03.01-84*, moments Mx and My are ignored.
Bearing capacity of concrete and reinforcement in punching shear zone is checked by absolute values of the specified load and moments.

There are two modes for calculation.

Mode 1. Calculation strictly by building code

If the **Calculation by code** option is selected, calculation will be made according to requirements of selected building code.

SNIP 52-01-2003

Reinforcement is computed if the following conditions are ***not*** satisfied:

1) $F_b \le F_{b,ult}$;

2) $V = \dfrac{F_b}{F_{b,ult}} + \dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} \le 1;$

3) $\dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} \le \dfrac{F_b}{F_{b,ult}}.$

In this case, if $\dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} > \dfrac{F_b}{F_{b,ult}}$, then it is considered that $\dfrac{M_x}{M_{bx,ult}} + \dfrac{M_y}{M_{by,ult}} = \dfrac{F_b}{F_{b,ult}}$.

The check below is made in calculation:

4) $V = \dfrac{F_b}{F_{b,ult} + F_{sw,ult}} + \dfrac{M_x}{M_{bx,ult} + M_{sw,x,ult}} + \dfrac{M_y}{M_{by,ult} + M_{sw,y,ult}} \le 1$.

Notations accepted in the formulas:
$F_{b,ult}$ – ultimate axial force that concrete of design cross-section of slab may take;
$M_{bx,ult}$ – ultimate moment in concrete about the Y-axis (in plane that passes through the X-axis);
$M_{by,ult}$ – ultimate moment in concrete about the X-axis (in plane that passes through the Y-axis);
$F_{sw,ult}$ – ultimate axial force in reinforcement;
$M_{sw,ult}$ – ultimate moment in reinforcement about the Y-axis (in plane that passes through the X-axis);
$M_{sw,ult}$ – ultimate moment in reinforcement about the X-axis (in plane that passes through the Y-axis).
v – check parameter.

SP 63.13330.2012

Reinforcement is computed if the following conditions are ***not*** satisfied:

1) $F_b \le F_{b,ult}$;

2) $V = \frac{F_b}{F_{b,ult}} + \frac{M_x}{M_{bx,ult}} + \frac{M_y}{M_{by,ult}} \le 1$;

In this case, if $\frac{M_x}{M_{bx,ult}} + \frac{M_y}{M_{by,ult}} > 0.5\frac{F_b}{F_{b,ult}}$,

then it is considered that $\frac{M_x}{M_{bx,ult}} + \frac{M_y}{M_{by,ult}} = 0.5\frac{F_b}{F_{b,ult}}$.

The check below is made in calculation:

3) $V = \frac{F_b}{F_{b,ult} + F_{sw,ult}} + \frac{M_x}{M_{bx,ult} + M_{sw,x,ult}} + \frac{M_y}{M_{by,ult} + M_{sw,y,ult}} \le 1$.

In this case, if

$$\frac{M_x}{M_{bx,ult} + M_{sw,x,ult}} + \frac{M_y}{M_{by,ult} + M_{sw,y,ult}} > 0.5\frac{F_b}{F_{b,ult} + F_{sw,ult}}$$

then it is considered that

$$\frac{M_x}{M_{bx,ult} + M_{sw,x,ult}} + \frac{M_y}{M_{by,ult} + M_{sw,y,ult}} = 0.5\frac{F_b}{F_{b,ult} + F_{sw,ult}}$$

If **V≤1**, then strength of slab in punching shear is provided with concrete.

Mode 2. Calculation by qsw

If the **Calculation by qsw** option is selected, calculation will be made by obtained value of force per r.m. in transverse reinforcement **qsw** with no limitations upwards.

In any mode of calculation, area of transverse reinforcement with specified spacing is computed if the following condition is satisfied:

$0.25F_{b,ult} \le F_{sw,ult} \le F_{b,ult}$.

If $F_{sw,ult} > F_{b,ult}$, then the message '*Ultimate force in reinforcement exceeds ultimate force in concrete'* is displayed and reinforcement is not computed.

If $F_{sw,ult} < 0.25 F_{b,ult}$, then it is considered that $F_{sw,ult} = 0.25 F_{b,ult}$.

Output data:
Asw – area of transverse reinforcement with specified spacing.
Asw_sum – total area of reinforcement.

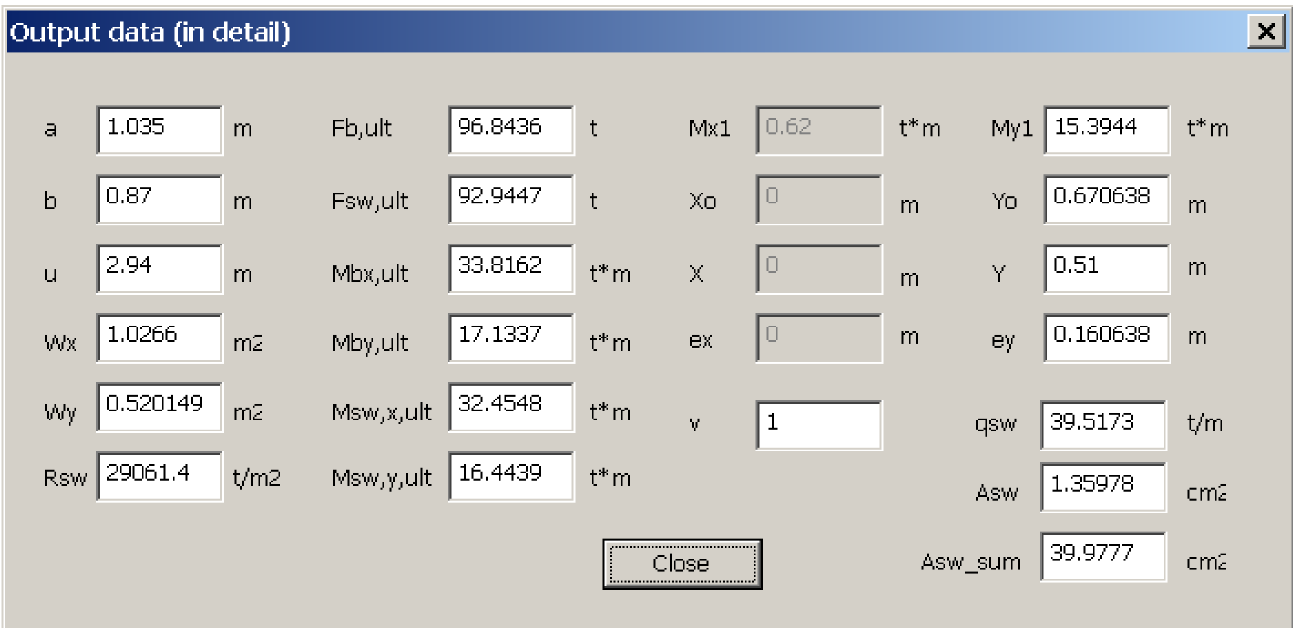


Figure 14.5

To display all temporary and final data for the calculation, in the **Output data** area of the dialog box, click **More**. If reinforcement is not required, then value for parameter **v** is displayed according to condition 2). Otherwise, value for parameter **v** is displayed according to condition 4).

To present a report document in HTML format, click **Report**.

Punching shear analysis (Eurocode)

The module enables you to carry out punching shear analysis of rectangular slabs and rectangular foundation slabs according to EN 1992-1-1 (Eurocode 2), DBN V.2.6-98:2009, DSTU B V.2.6-156:2010 (Ukraine) and CH PK EN 1992-1-1:2004/2011 (Kazakhstan).

The following cases (schematic presentations) may be defined:
1. Column inside slab.
2. Column inside slab: opening to the right of the column.

3. Column inside slab: opening above the column.
4. Column at the left edge (of slab) that is parallel to Y-axis.
5. Column at the left edge (of slab) that is parallel to Y-axis: opening to the right of the column.
6. Column at the bottom edge (of slab) that is parallel to X-axis
7. Column at the bottom edge (of slab) that is parallel to X-axis: opening above the column.
8. Column at corner of slab.

For every case, the following variants may be considered:
1. No capital.
2. Capital short – check only along control section outside the capital.
3. Capital long – check along control section within the capital limits.
4. Capital long – check along control section of slab outside the capital.

The dialog box contains two tabs (see the figure below): **Input data** and **Output data**.

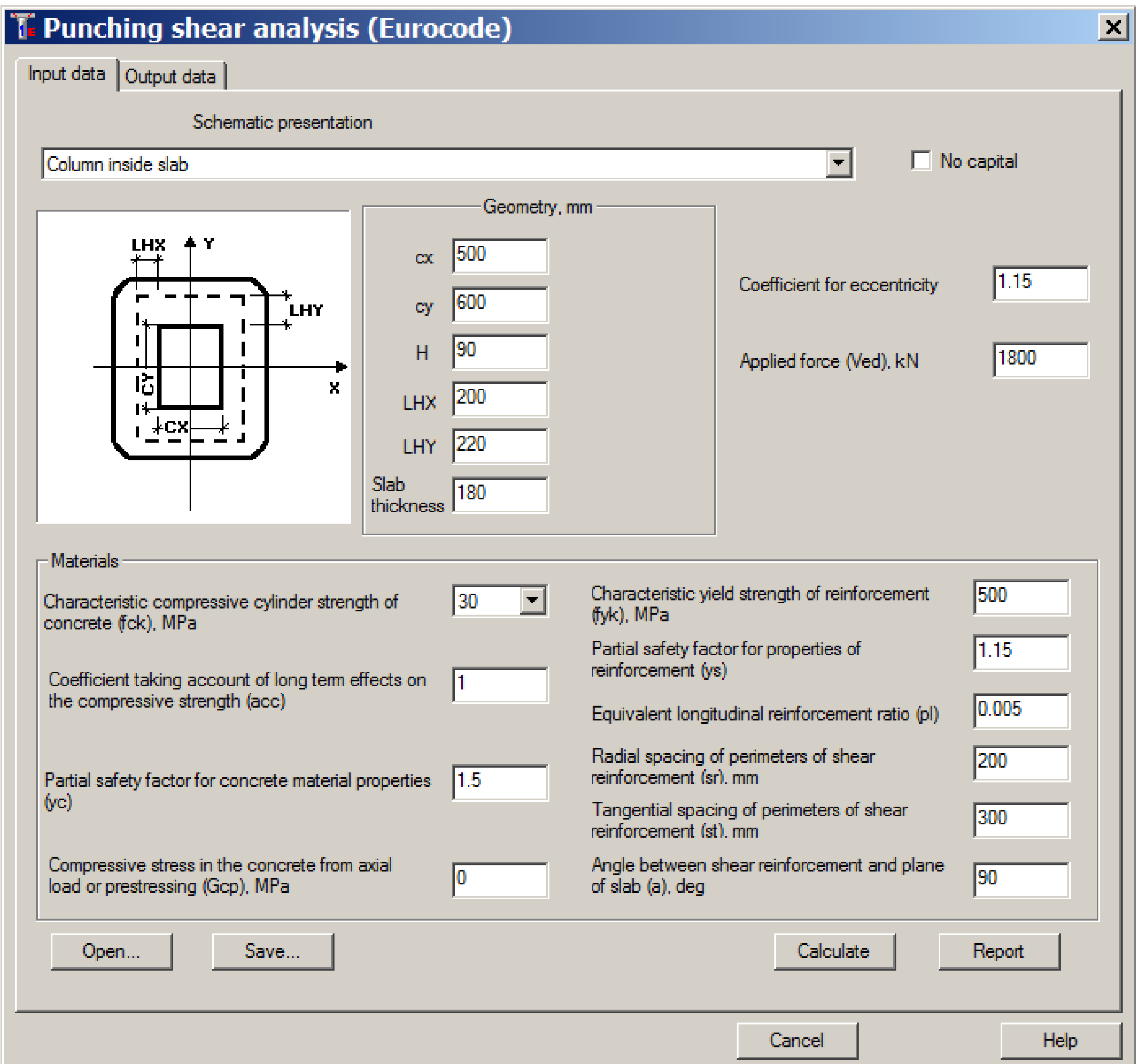


Figure 14.6

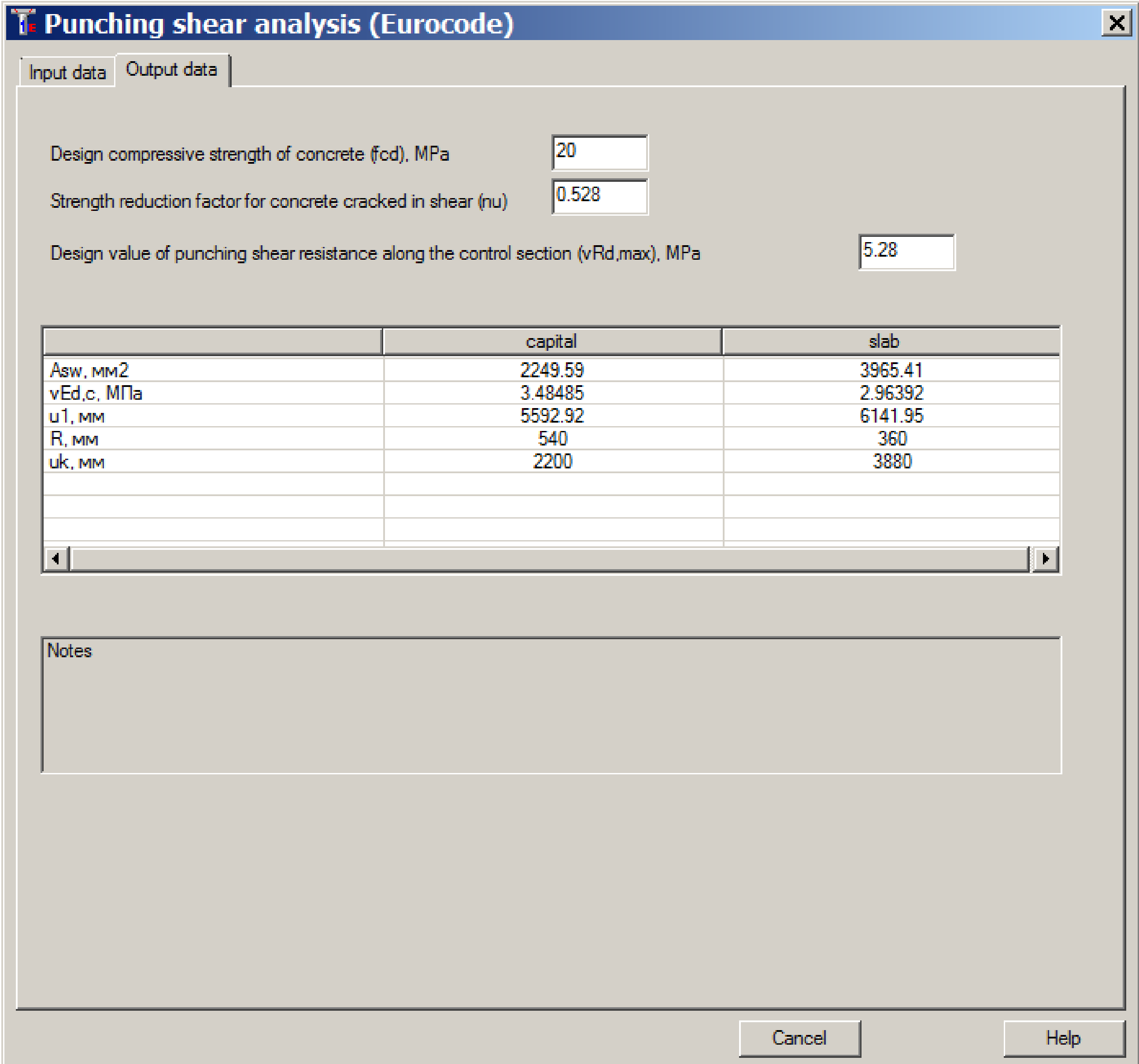


Figure 14.7

Input data

1. Geometry

If there is no capital, select the **No capital** check box.
cx – column dimension along the X-axis, mm;
cy – column dimension along the Y-axis, mm;
ax – distance between slab edge and left face of column along the X-axis, mm;
ay – distance between slab edge and bottom face of column along the Y-axis, mm;
d – effective thickness of slab, mm;

If there is a capital (the **No capital** check box should be clear)

H – height of capital;
LHX – distance from the column face to the edge of the column head along the X-axis, mm;
LHY – distance from the column face to the edge of the column head along the Y-axis, mm;

If there is an opening
Lx – dimension of opening along the X-axis, mm;
Ly – dimension of opening along the Y-axis, mm;
Sx – distance along the X-axis between right face of column and the nearest side of opening, mm; $cx/2 \le Sx \le 6*d$;
Sy – distance along the Y-axis between upper face of column and the nearest side of opening, mm; $cy/2 \le Sy \le 6*d$.

2. Load and materials

β – coefficient for eccentricity (at slab edge 1.4); may be defined manually;
V_{Ed} – applied force, kN.

Concrete
f_{ck} – characteristic compressive cylinder strength of concrete, MPa;
γ_c – partial safety factor for concrete material properties (table 2.1N); by default it is equal to 1.5;
α_{cc} – coefficient taking account of long term effects on the compressive strength; recommended value 1.0 (may be 0.8 – 1.0);
σ_{cp} – compressive stress in the concrete from axial load or prestressing; by default it is equal to 0.

Reinforcement
f_{yk} – characteristic yield strength of reinforcement, MPa;
γ_s – partial safety factor for properties of reinforcement ; by default it is equal to 1.15; may be = 1.0;
ρl – equivalent longitudinal reinforcement ratio ;
s_r – radial spacing of perimeters of shear reinforcement, mm;
s_t – tangential spacing of perimeters of shear reinforcement, mm;
α – angle between shear reinforcement and plane of slab, deg.

To start calculation, click **Calculate**.

If **LHX > ax** and/or **LHY > ay**, then calculation is terminated and the program displays the message: '*Capital edge is located outside the slab*'.
If **Sx < LHx** and/or **Sy < LHy**, then calculation is terminated and the program displays the message: '*Distance from the column face to the edge of the column head exceeds the distance from column face to opening*'.

Output data

On this tab you will see results of calculation.

In the appropriate boxes, the following data is displayed:
f_{cd} – design compressive strength of concrete, MPa;
nu – strength reduction factor for concrete cracked in shear;
$v_{Rd,max}$ – design value of punching shear resistance along the control section, MPa.

The table displays the following data:
A_{sw} – design area of reinforcement for basic control perimeter (u1), mm^2;
$v_{Ed,c}$ – stress along column face, MPa;
u1 – length of the basic control perimeter, mm;
R – curvature radius for basic control perimeter, mm;
uk – perimeter of column section, mm.

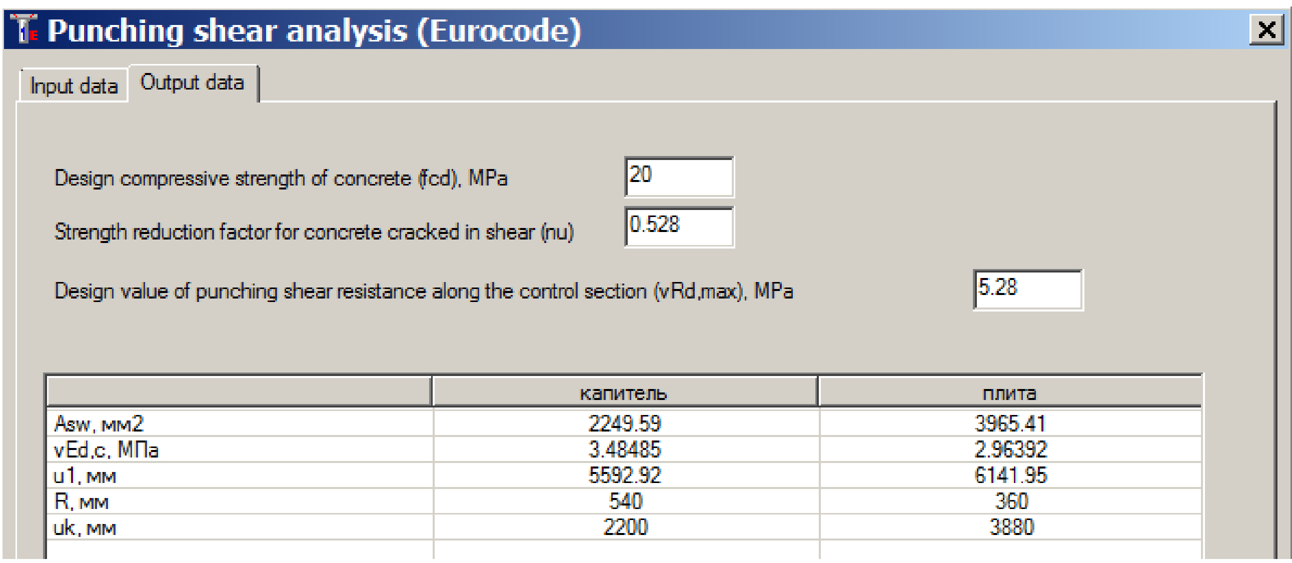


Figure 14.8

To present a report document in HTML format, click **Report**. The report file contains all intermediate and final results with necessary comments.

	Geometry, mm	
cx	Column dimension along X	500
cy	Column dimension along Y	600
d	Effective thickness of slab	180
LHX	Distance from the column face to the edge of the column head along X	200
LHY	Distance from the column face to the edge of the column head along Y	220
H	Height of capital	90
	Materials	
fck	Characteristic compressive cylinder strength of concrete, MPa	30
αcc	Coefficient taking account of long term effects on the compressive strength	1
γc	Partial safety factor for concrete material properties	1.5
σcp	Compressive stress in the concrete from axial load or prestressing, Mpa	0
fyk	Characteristic yield strength of reinforcement, MPa	500
γs	Partial safety factor for properties of reinforcement	1.15
ρl	Equivalent longitudinal reinforcement ratio	0.005
sr	Radial spacing of perimeters of shear reinforcement, mm	200
st	Tangential spacing of perimeters of shear reinforcement, mm	300
α	Angle between shear reinforcement and plane of slab, deg	90
	Load	
β	Coefficient for eccentricity	1.15
VEd	Applied force, kN	1800

Output data

fcd	Design compressive strength of concrete, MPa	20
nu	Strength reduction factor for concrete cracked in shear	0.528
vRd,max	Design value of punching shear resistance along the control section, MPa	5.28

		capital	slab
u1	Length of the basic control perimeter, mm	5592.92	6141.95
vEd,c	Stress along column face, MPa	3.48485	2.96392
vRd,c	Design value of punching shear resistance without reinforcement, MPa	0.550655	0.591891
vm	Min allowed resistance of concrete, MPa	0.486553	0.542218
vEd	Stress in basic contour, MPa	1.37078	1.87237
uo	Ultimate perimeter of contour in reinforcement zone, mm	13922.8	19429.3
Rp	Curvarure radius of ultimate perimeter, mm	1460.74	2204.74
fywd,ef	Effective design strength of punching shear reinforcement, MPa	317.5	295
Asw	Design area of reinforcement, mm2	2249.59	3965.41
Asw/u1	Area of reinforcement per 1r.m. of basic perimeter, mm2	0.402221	0.645628

Figure 14.9

Punching shear analysis (Belarus)

The module enables you to carry out punching shear analysis for rectangular floor slabs and rectangular mat foundations according to building code SNB 5.03.01-02 (Belarus).

The following cases (schematic presentations) may be defined:
1. Column inside slab.
2. Column inside slab: opening to the right of the column.
3. Column inside slab: opening above the column.
4. Column at the left edge (of slab) that is parallel to Y-axis.
5. Column at the left edge (of slab) that is parallel to Y-axis: opening to the right of the column.
6. Column at the bottom edge (of slab) that is parallel to X-axis
7. Column at the bottom edge (of slab) that is parallel to X-axis: opening above the column.
8. Column at corner of slab.

For every case, the following variants may be considered:
1. No capital.
2. Capital short – LH ≤ 1.5*H.
3. Capital medium – 1.5*H ≤ LH ≤ 1.5*(d+H).

4. Capital long – LH >1.5*(d+H). In this case, the program carries out a check of two control perimeters: within the capital limits and outside the capital.
Here d – effective height of slab; H – height of capital.
With a capital, the variant for computation is determined automatically.

The dialog box contains two tabs (see the figure below): **Input data** and **Output data**.

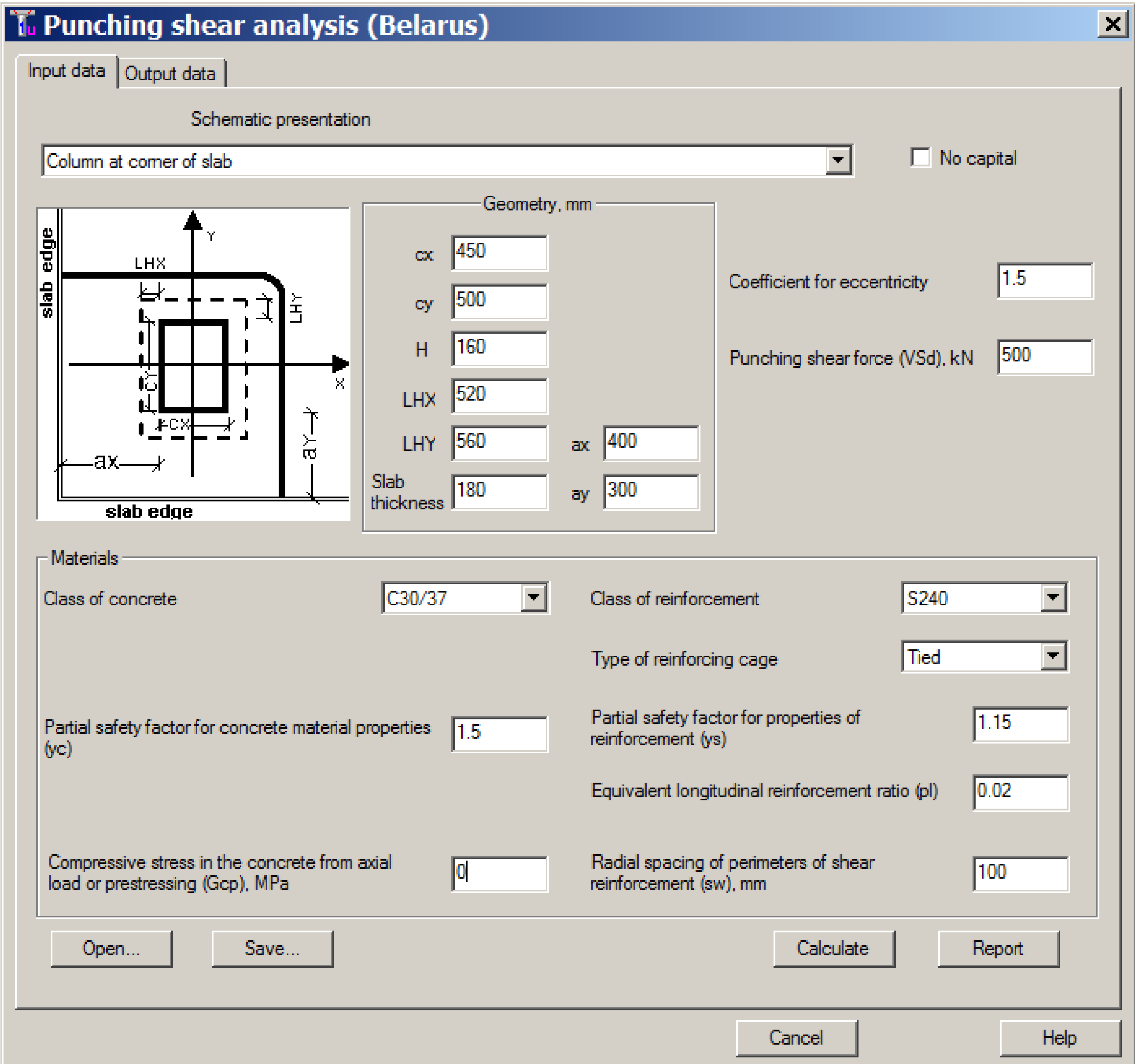


Figure 14.10

Input data

1. Geometry

Select the necessary schematic presentation from the list. If there is no capital, select the **No capital** check box.

cx – column dimension along the X-axis, mm;
cy – column dimension along the Y-axis, mm;
ax – distance between slab edge and left face of column along the X-axis, mm;
ay – distance between slab edge and bottom face of column along the Y-axis, mm;
d – effective thickness of slab, mm;

I*f there is a capital* (the **No capital** check box should be clear)
H – height of capital;
LHX – distance from the column face to the edge of the column head along the X-axis, mm;
LHY – distance from the column face to the edge of the column head along the Y-axis, mm;

If there is an opening
Lx – dimension of opening along the X-axis, mm;
Ly – dimension of opening along the Y-axis, mm;
Sx – distance along the X-axis between right face of column and the nearest side of opening, mm; **cx/2 ≤ Sx ≤ 6*d**;
Sy – distance along the Y-axis between upper face of column and the nearest side of opening, mm; **cy/2 ≤ Sy ≤ 6*d**.

2. Load and material properties

β – coefficient for eccentricity: for the column inside the slab – 1.15; for the column at slab edge – 1.4; for the column at corner of slab – 1.5; may be defined manually;
V_{Ed} – applied force, kN.

Concrete
Select class of concrete from the list. For example, C30/37.
γ_c – partial safety factor for concrete material properties; by default it is equal to 1.5;
σ_{cp} – compressive stress in the concrete from axial load or prestressing; by default it is equal to 0.

Reinforcement
Select class of reinforcement from the list. For example, S400.
Select the type of reinforcing cage: either welded or tied.
γ_s – partial safety factor for properties of reinforcement; by default it is equal to 1.15; may be = 1.0;
ρl – equivalent longitudinal reinforcement ratio;
sw – radial spacing of perimeters of shear reinforcement, mm.

To start calculation, click **Calculate**.

If **LHX > ax** and/or **LHY > ay**, then calculation is terminated and the program displays the message: '*Capital edge is located outside the slab*'.
If **Sx < LHx** and/or **Sy < LHy**, then calculation is terminated and the program displays the message: '*Distance from the column face to the edge of the column head exceeds the distance from column face to opening*'.

Output data

On this tab you will see results of calculation.

In the appropriate boxes, the following data is displayed:
A_{sw} – design area of reinforcement for basic control perimeter (U), mm^2;
vSd – force (per 1r.mm.) along control perimeter, N/mm;
vRd_c – design value of punching shear resistance (per 1r.mm.) without reinforcement, N/mm;
U – length of the basic control perimeter, mm;
R – curvature radius for basic control perimeter, mm.

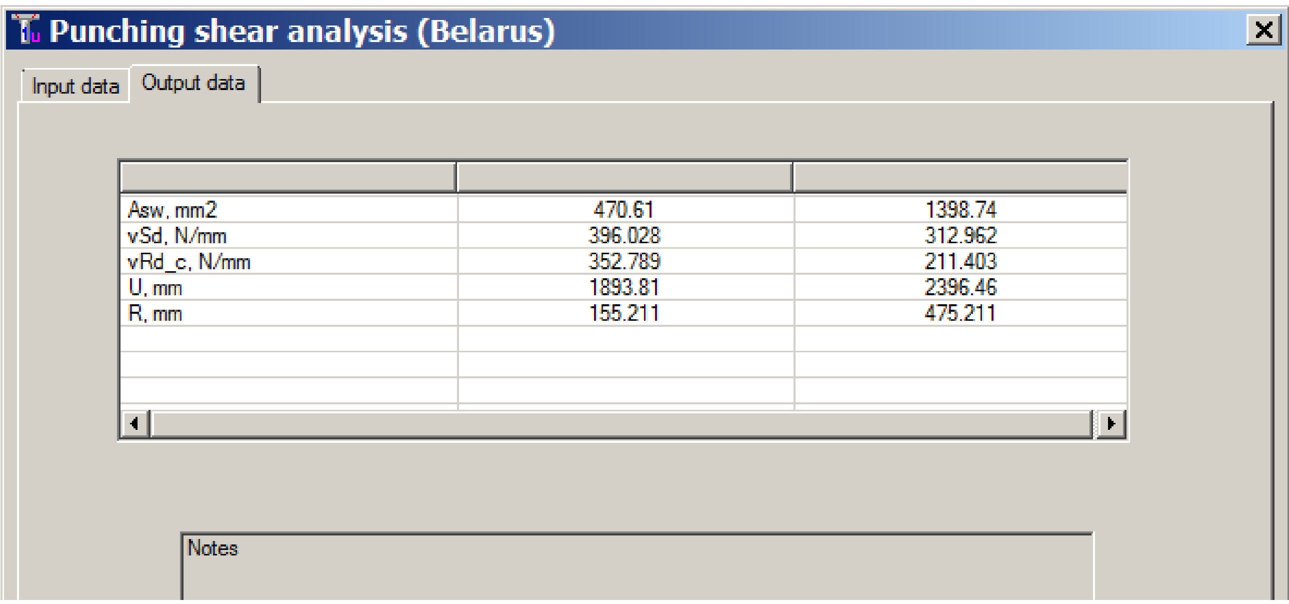


Figure 14.11

To present a report document in HTML format, click **Report**. The report file contains all intermediate and final results with necessary comments.

Input data

Column at corner of slab		
Long capital		
Geometry, mm		
cx	Column dimension along X	450
cy	Column dimension along Y	500
d	Effective thickness of slab	180
LHX	Distance from the column face to the edge of the column head along X	520
LHY	Distance from the column face to the edge of the column head along Y	560
H	Height of capital	160
ax	Distance between slab edge and left face of column along X	400
ay	Distance between slab edge and bottom face of column along Y	300
Materials		
Class of concrete		C30/37
Class of reinforcement		S240
Type of reinforcing cage		Tied
fck	Normative compressive strength of concrete, MPa	30
fctk	Normative tensile strength of concrete, MPa	2
fctm	Tensile strength of concrete (accepted for design procedure), MPa	2.9
fywd	Design strength of transverse reinforcement, N/mm2	174
γc	Partial safety factor for concrete material properties	1.5
Gcp	Compressive stress in the concrete from axial load or prestressing, MPa	0
γs	Partial safety factor for properties of reinforcement	1.15
ρl	Equivalent longitudinal reinforcement ratio	0.02
sw	Radial spacing of perimeters of shear reinforcement, mm	100
Load		
β	Coefficient for eccentricity	1.5
VSd	Applied force, kN	500

Output data

vSd,k	Force along column contour, N/mm	925.926
fcd	Design compressive strength of concrete, MPa	20
nu	Strength reduction factor for concrete cracked in shear	0.528
vmax	Design value of max force along the control section, N/mm	1795.2

U	Length of the basic control perimeter, mm	1893.81	2396.46
vSd	Force (per 1r.mm.) along control perimeter, N/mm	396.028	312.962
vRd,c	Design value of punching shear resistance (per 1r.mm.) without reinforcement, N/mm	352.789	211.403
vmin	Min allowed value of punching shear resistance (per 1r.mm.) without reinforcement, N/mm	295.652	156.522
ρw,min	Min reinforcement ratio	0.00193333	0.00193333
ρsw	Reinforcement ratio	0.002485	0.00583671
Asw1	Min design area of reinforcement, mm2	0	0
Asw	Design area of reinforcement, mm2	470.61	1398.74
Asw/U	Area of reinforcement per 1r.mm. of basic perimeter, mm2/r.mm	0.2485	0.583671

Figure 14.12

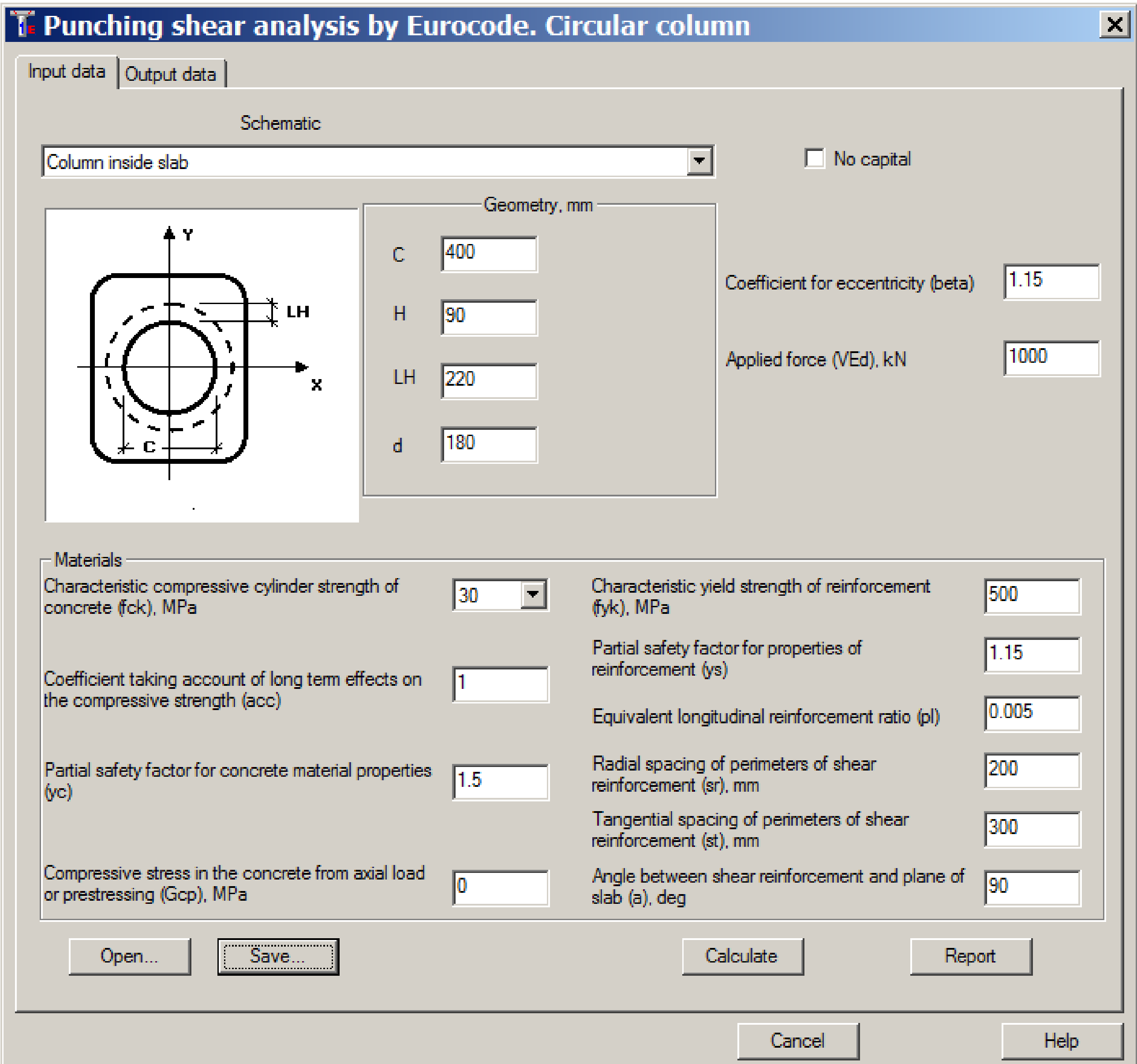
Punching shear for circular contour (Eurocode)

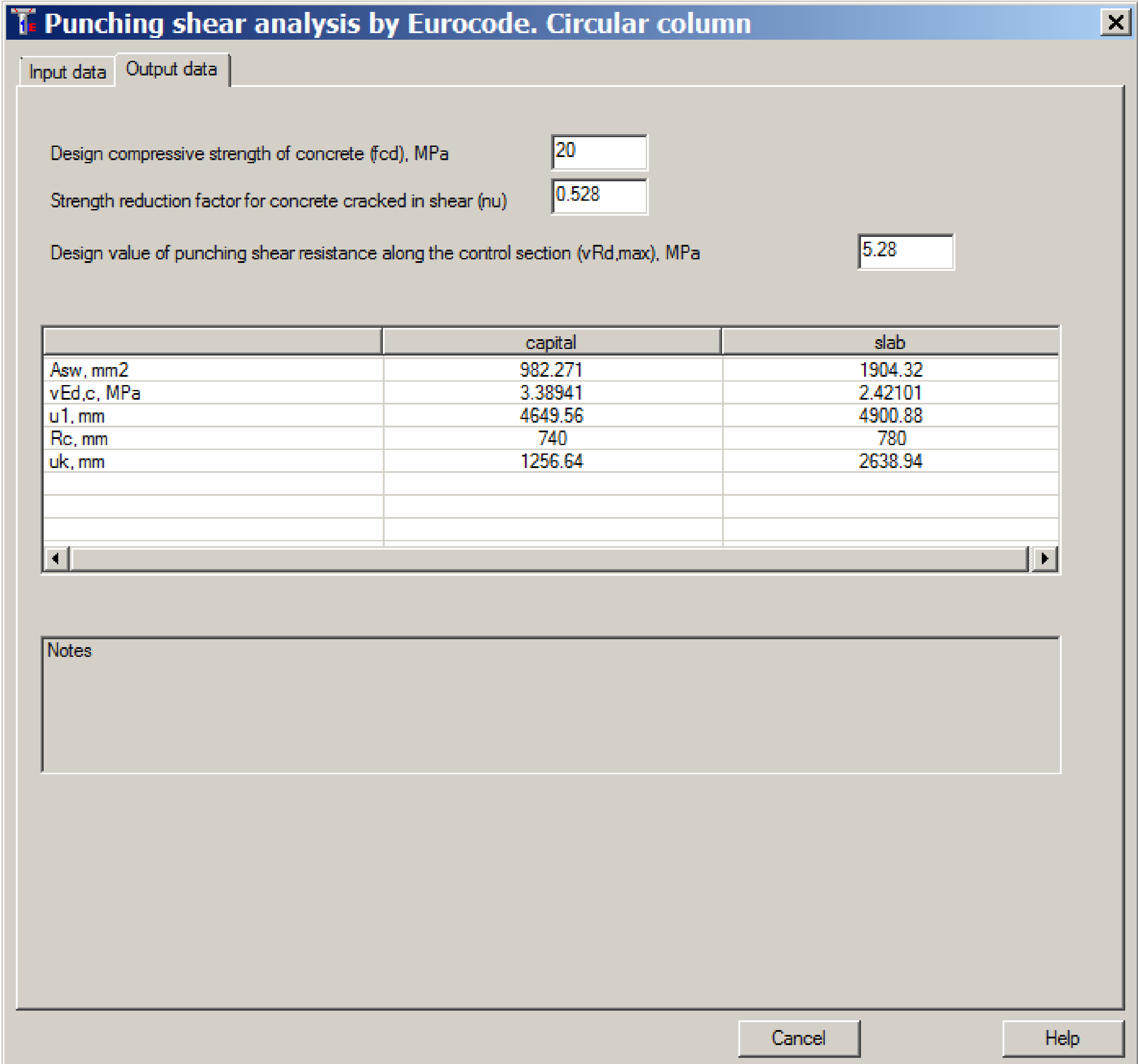
The module enables you to carry out punching shear analysis of RC floor slabs and RC foundation slabs with circular column. The following building codes are supported: EN 1992-1-1 (Eurocode 2), DBN V.2.6-98:2009, DSTU B V.2.6-156:2010 (Ukraine) and CH PK EN 1992-1-1:2004/2011 (Kazakhstan).

The following cases (schematic presentations) may be defined:
1. Column inside slab.
2. Column inside slab: opening to the right of the column.
3. Column at the edge of slab.
4. Column at the edge of slab: opening to the right of the column.
5. Column at corner of slab.

For every case, the following variants may be considered:
1. No capital.
2. Capital short – check only along control section outside the capital.
3. Capital long – check along control section within the capital limits.
4. Capital long – check along control section of slab outside the capital.

The dialog box contains two tabs (see the figure below): **Input data** and **Output data**.





Input data

1. Geometry

If there is no capital, select the **No capital** check box.
C – diameter of column, mm;
a – distance between slab edge and column centre, mm;
d – effective thickness of slab, mm.

If there is a capital (the **No capital** check box should be clear)
H – height of capital;
LH – distance from the column body to the edge of the column head, mm.

If there is an opening
Lx – dimension of opening along the X-axis, mm;

Ly – dimension of opening along the Y-axis, mm;
S – distance between the column centre and the nearest side of opening along the line that connects column centre and centre of the opening, mm; $C/2 \le S \le (6*d+C/2)$.

2. Load and materials

β – coefficient for eccentricity (at slab edge 1.4); may be defined manually;
V_{Ed} – applied force, kN.

Concrete
f_{ck} – characteristic compressive cylinder strength of concrete, MPa;
γ_c – partial safety factor for concrete material properties (table 2.1N); by default it is equal to 1.5;
α_{cc} – coefficient taking account of long term effects on the compressive strength; recommended value 1.0 (may be 0.8 – 1.0);
σ_{cp} – compressive stress in the concrete from axial load or prestressing; by default it is equal to 0.

Reinforcement
f_{yk} – characteristic yield strength of reinforcement, MPa;
γ_s – partial safety factor for properties of reinforcement ; by default it is equal to 1.15; may be = 1.0;
ρl – equivalent longitudinal reinforcement ratio ;
s_r – radial spacing of perimeters of shear reinforcement, mm;
s_t – tangential spacing of perimeters of shear reinforcement, mm;
α – angle between shear reinforcement and plane of slab, deg.

To start calculation, click **Calculate**.

If **LH > a**, then calculation is terminated and the program displays the message: '*Capital edge is located outside the slab*'.
If **(S-C/2) < LH**, then calculation is terminated and the program displays the message: '*Distance from the column body to the edge of the column head exceeds the distance from column face to opening*'.

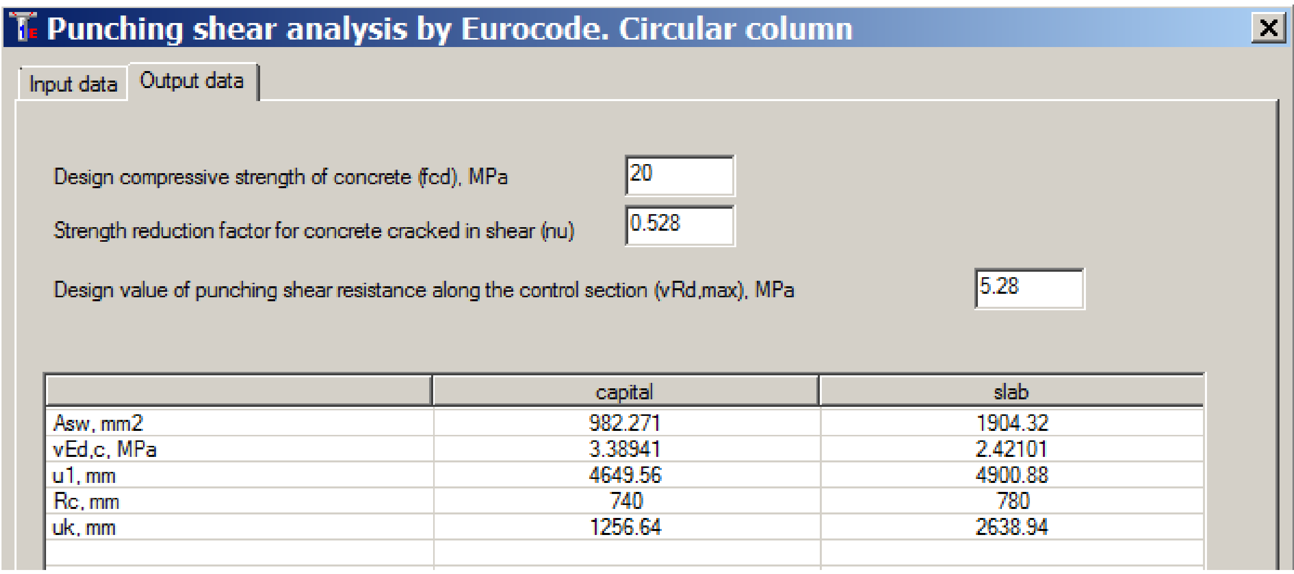
If computed radius of punching shear contour exceeds the slab limits, then analysis is terminated and appropriate warning is displayed on the screen.

Output data

On this tab you will see results of calculation.

In the appropriate boxes, the following data is displayed:
f_{cd} – design compressive strength of concrete, MPa;
nu – strength reduction factor for concrete cracked in shear;
$v_{Rd,max}$ – design value of punching shear resistance along the control section, MPa.

The table displays the following data:
A_{sw} – design area of reinforcement for basic control perimeter (u1), mm^2;
$v_{Ed,c}$ – stress along column face, MPa;
u1 – length of the basic control perimeter, mm;
R – curvature radius for basic control perimeter, mm;
uk – perimeter of column section, mm.



	capital	slab
Asw, mm2	982.271	1904.32
vEd,c, MPa	3.38941	2.42101
u1, mm	4649.56	4900.88
Rc, mm	740	780
uk, mm	1256.64	2638.94

To present a report document in HTML format, click **Report**. The report file contains all intermediate and final results with necessary comments.

Input data

Column inside slab		
Long capital		
Geometry, mm		
c	Diameter of column	400
d	Effective thickness of slab	180
LH	Distance from the column face to the edge of the column head along X	220
H	Height of capital	90
Materials		
fck	Characteristic compressive cylinder strength of concrete, MPa	30
αcc	Coefficient taking account of long term effects on the compressive strength	1
γc	Partial safety factor for concrete material properties	1.5
Gcp	Compressive stress in the concrete from axial load or prestressing, Mpa	0
fyk	Characteristic yield strength of reinforcement, MPa	500
γs	Partial safety factor for properties of reinforcement	1.15
ρl	Equivalent longitudinal reinforcement ratio	0.005
sr	Radial spacing of perimeters of shear reinforcement, mm	200
st	Tangential spacing of perimeters of shear reinforcement, mm	300
α	Angle between shear reinforcement and plane of slab, deg	90
Load		
β	Coefficient for eccentricity	1.15
VEd	Applied force, kN	1000

Output data

fcd	Design compressive strength of concrete, MPa	20
nu	Strength reduction factor for concrete cracked in shear	0.528
vRd,max	Design value of punching shear resistance along the control section, MPa	5.28

		capital	slab
u1	Length of the basic control perimeter, mm	4649.56	4900.88
vEd,c	Stress along column face, MPa	3.38941	2.42101
vRd,c	Design value of punching shear resistance without reinforcement, MPa	0.550655	0.591891
vm	Min allowed resistance of concrete, MPa	0.486553	0.542218
vEd	Stress in basic contour, MPa	0.916057	1.30362
uo	Ultimate perimeter of contour in reinforcement zone, mm	7734.9	10794
Rp	Curvature radius of ultimate perimeter, mm	826.048	1447.92
fywd,ef	Effective design strength of punching shear reinforcement, MPa	317.5	295
Asw	Design area of reinforcement, mm2	982.271	1904.32
Asw/u1	Area of reinforcement per 1r.m. of basic perimeter, mm2	0.211261	0.388566

Toster

TOSTER (Chapter)

This chapter contains module that enables you to carry out static analysis of 2D and 3D bar systems in bending torsion.

TOSTER. Thin-walled bar systems

Figure 15.1

TOSTER - General notes

Thin-walled bar systems (2D and 3D) have seven degrees of freedom per node: three translations (X, Y, Z), three rotations (UX, UY, UZ) and warping (Ud).

Model of the frame is located in the XOZ-plane. X-axis is horizontal while Z-axis is vertical. Y-axis is directed out of the plane and generates the right Cartesian coordinate system with the X and Z-axes. Let us refer this coordinate system as global or principal one.

Nodal loads – forces and moments are directed relative to the global coordinate system. Force is considered to be positive if it is directed against appropriate axis. Moment is considered to be positive if it is directed clockwise when you look from the end of appropriate axis.

Initial translation is considered to be positive if it acts along direction of appropriate axes.

Initial rotation is considered to be positive if it is directed anti-clockwise when you look from the end of appropriate axis.

Restraints imposed at support nodes are directed relative to the global coordinate system. Full restraint of a node is defined with seven restraints – X, Y, Z, UX, UY, UZ, Ud.

Local coordinate system of every bar is also right Cartesian coordinate system. X1-axis is longitudinal axis of the bar; it passes from the beginning of the bar up to its end through the *gravity centre* of the section. As a rule, node of the model with the smaller number is considered as beginning of the bar while node of the model with the greater number is considered as the end of the bar.

Y1-axis and Z1-axis – principal central axes that also pass through the gravity centre.

Local axes are generated according to the rules below.

For vertical bars:
– if the X1-axis is directed upward, then the Y1-axis is horizontal and directed out of the plane at us and the Z1-axis is directed left-to-right;
– if the X1-axis is directed downward, then the Y1-axis is horizontal and directed out of the plane at us and Z1-axis is directed right-to-left.

For arbitrary oriented bars:
– the Z1-axis is always directed to the upper half-space;
– if the X1-axis is directed left-to-right, then the Y1-axis is horizontal and directed out of the plane from us;
– if the X1-axis is directed right-to-left, then the Y1-axis is horizontal and directed out of the plane at us.

Local load is oriented relative to local coordinate axes. Sign convention for the local load is similar to the sign convention for the nodal load.

If ***gravity centre and rigidity centre of the section do not coincide***, then ***local load*** (that passes through gravity centre of the section rather than through rigidity centre) ***will cause local torsion moment***. It causes warping of the section and bimoment appears in the section.

Since longitudinal axes of bars (X1) considered at nodes of the model pass through gravity centres of their sections, then nodal force will also pass through their common gravity centre. That's why, nodal force in the plane of frame will not cause warping and bimoment.

To define more precisely parameters of bending torsion, it is advisable to provide intermediate division for bars between nodes of the frame. For example, if the model represents one bar fixed at both ends, then it is advisable to divide this bar into two or more bars.

Hinges are not supported in this version of the program.

Angle of pure rotation (rotation angle of principal central axes relative to location accepted by default) is not supported in this version of the program. That's why, assigned sections of bars should be symmetric relative to the local Y1-axis.

Input data

Let's consider the order in which the input data is generated on the example of the frame presented in the Figure below.

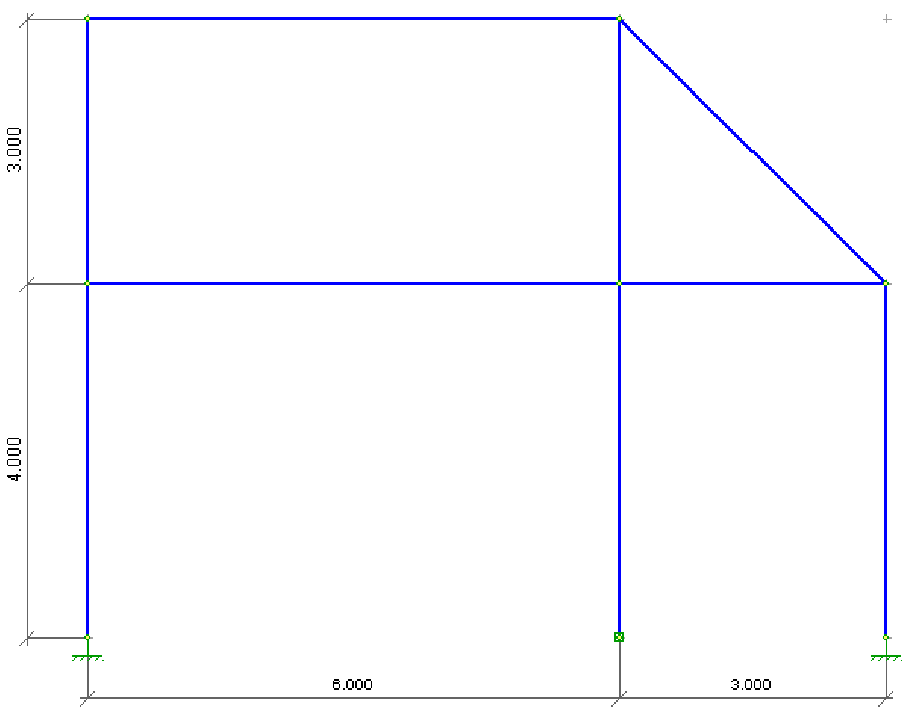


Figure 15.2

Step 1. Generate model geometry

To start the work, on the MODEL menu, click **Grid** (button  on the toolbar). In the **Grid generation** dialog box, define steps along axis 1 (horizontal axis X) and steps along axis 2 (vertical axis Z) with account of intermediate division of bars between nodes – the first span is divided into 6 elements by 1m and the second span is divided into 4 elements by 0.75m; the first storey is divided into 4 elements by 1m and the second storey is divided into 4 elements by 0.75m. Click **Apply**. Nodes of the grid will be displayed on the screen. They are denoted with grey cross symbols. To present defined dimensions of the grid on the screen, click **Display dimensions** button  .

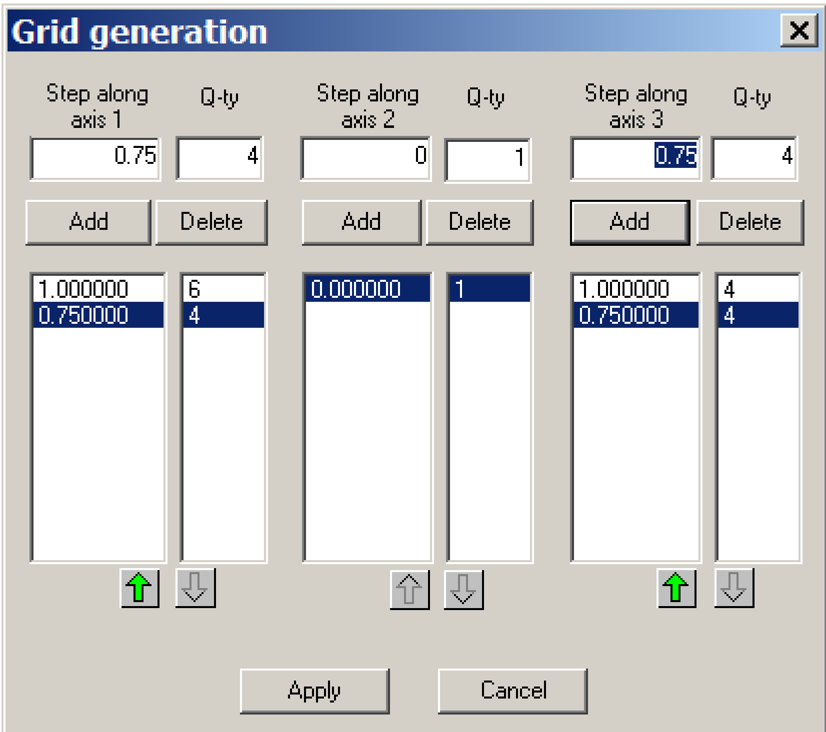


Figure 15.3

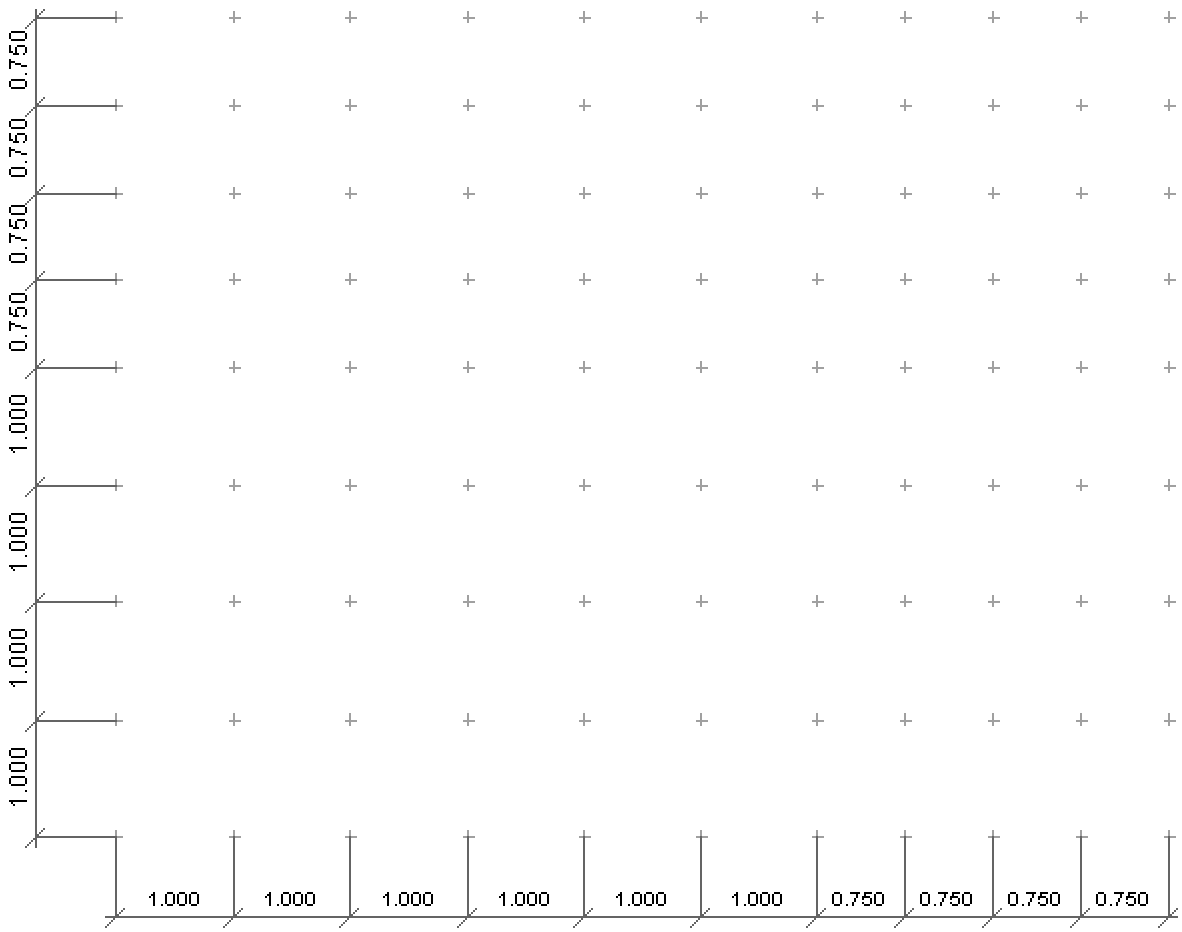


Figure 15.4

To define bars, on the MODEL menu, click **Bars** (button  on the toolbar) and connect nodes of the grid.

When you define bars, the mouse buttons work in the following way.

Left mouse button: the first click – to select the first node, the second click – to select the second node. Right mouse button: to unselect the first node.

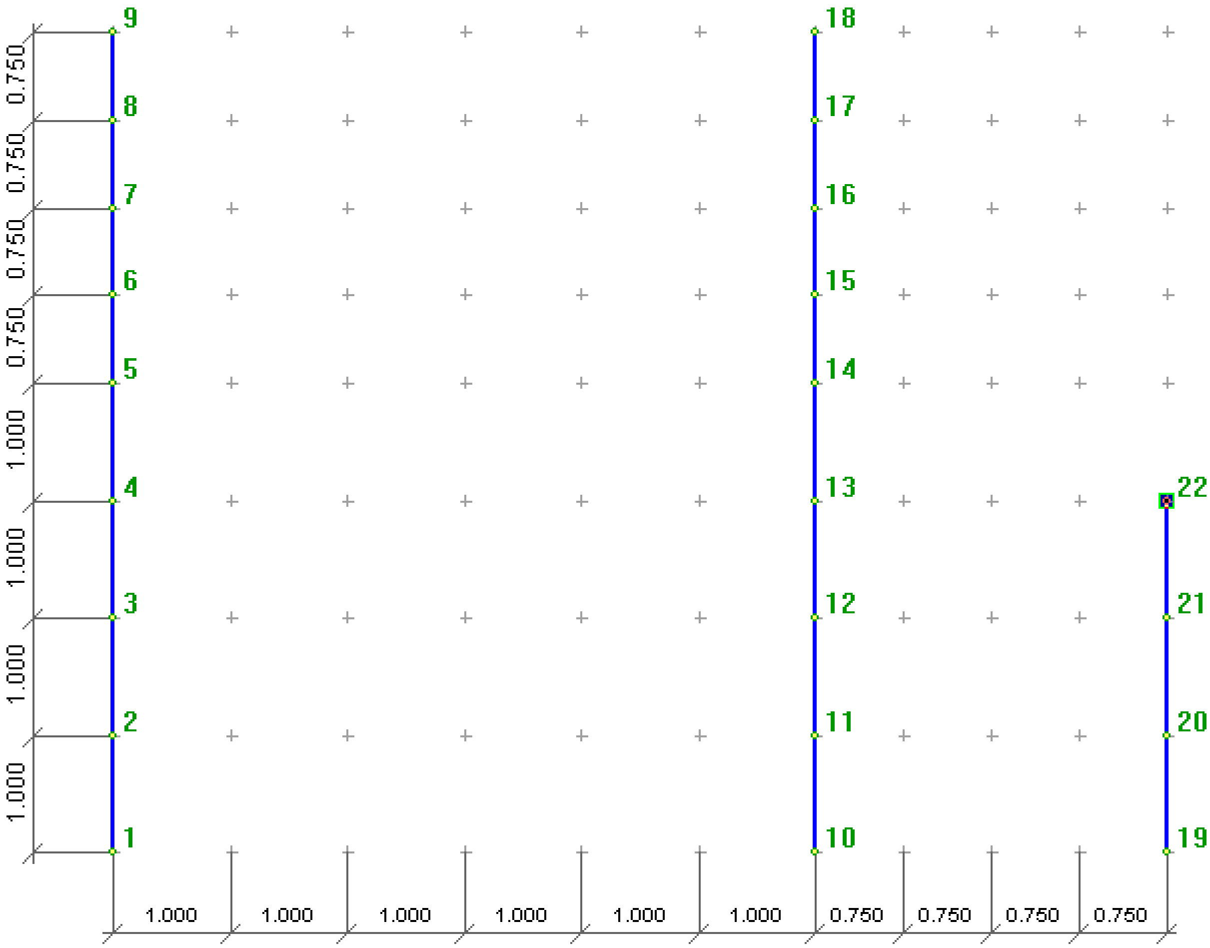

9
8
7
6
5
4
3
2
1
18
17
16
15
14
13
12
11
10
22
21
20
19
0.750
0.750
0.750
0.750
1.000
1.000
1.000
1.000
1.000 1.000 1.000 1.000 1.000 1.000 0.750 0.750 0.750 0.750

Figure 15.5 a)

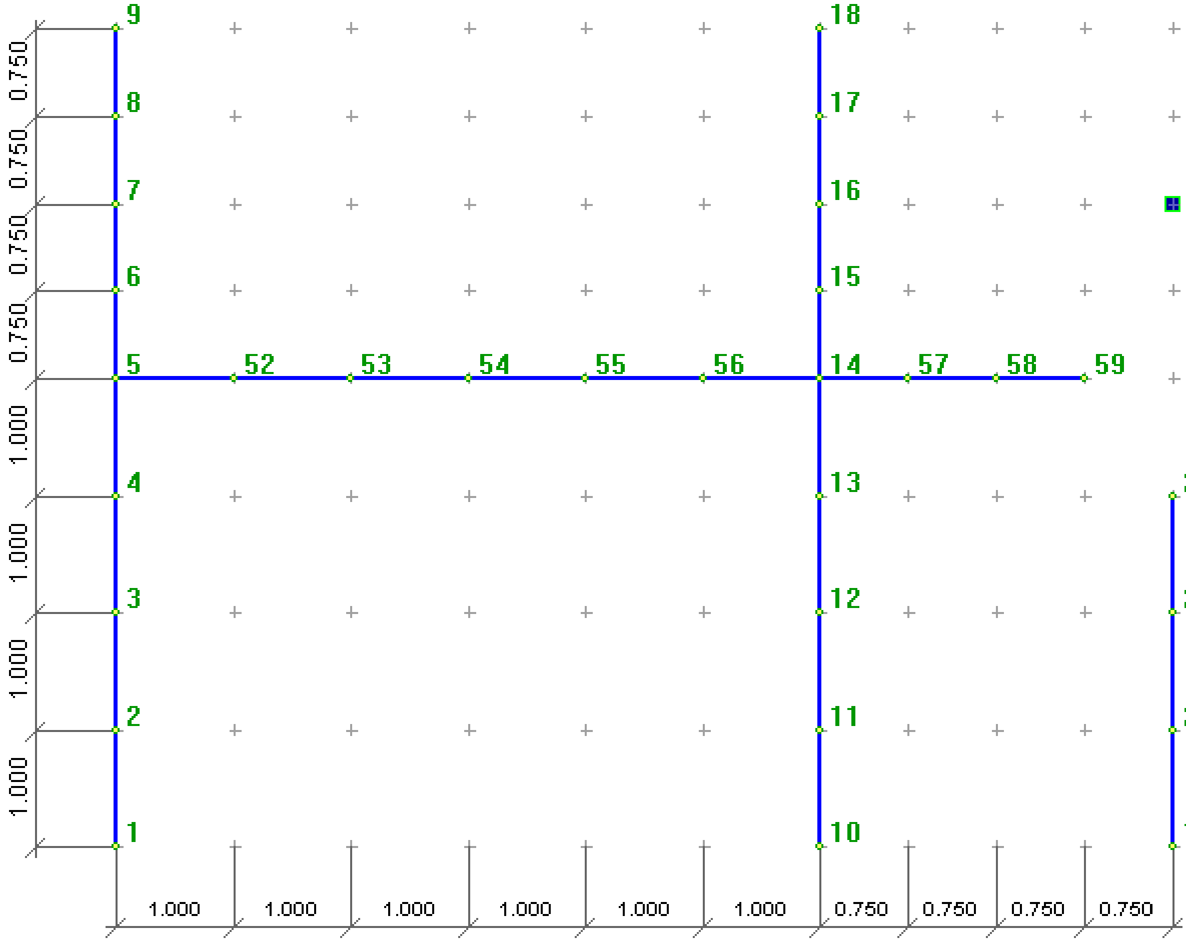


Figure 15.5 b)

Before you define bars it is recommended to present node numbers on the model (click **Display nodes** button on the toolbar). Node numbers will be displayed in green colour when you define bars.

Since the less number is assigned to the first node, it is recommended to define bars from bottom to top and from left to right in order to unify direction of the X1-axis.

When model geometry is generated, click the **Bars** button once again in order to make this command (mode) not active.

Click **Display nodes** button once again in order to make this command (mode) not active and then click **Display bars** button in order to display numbers for bars. Numbers of bars will be displayed in red.

To hide presentation of grid, click the **Grid** button once more to make this command (mode) not active.

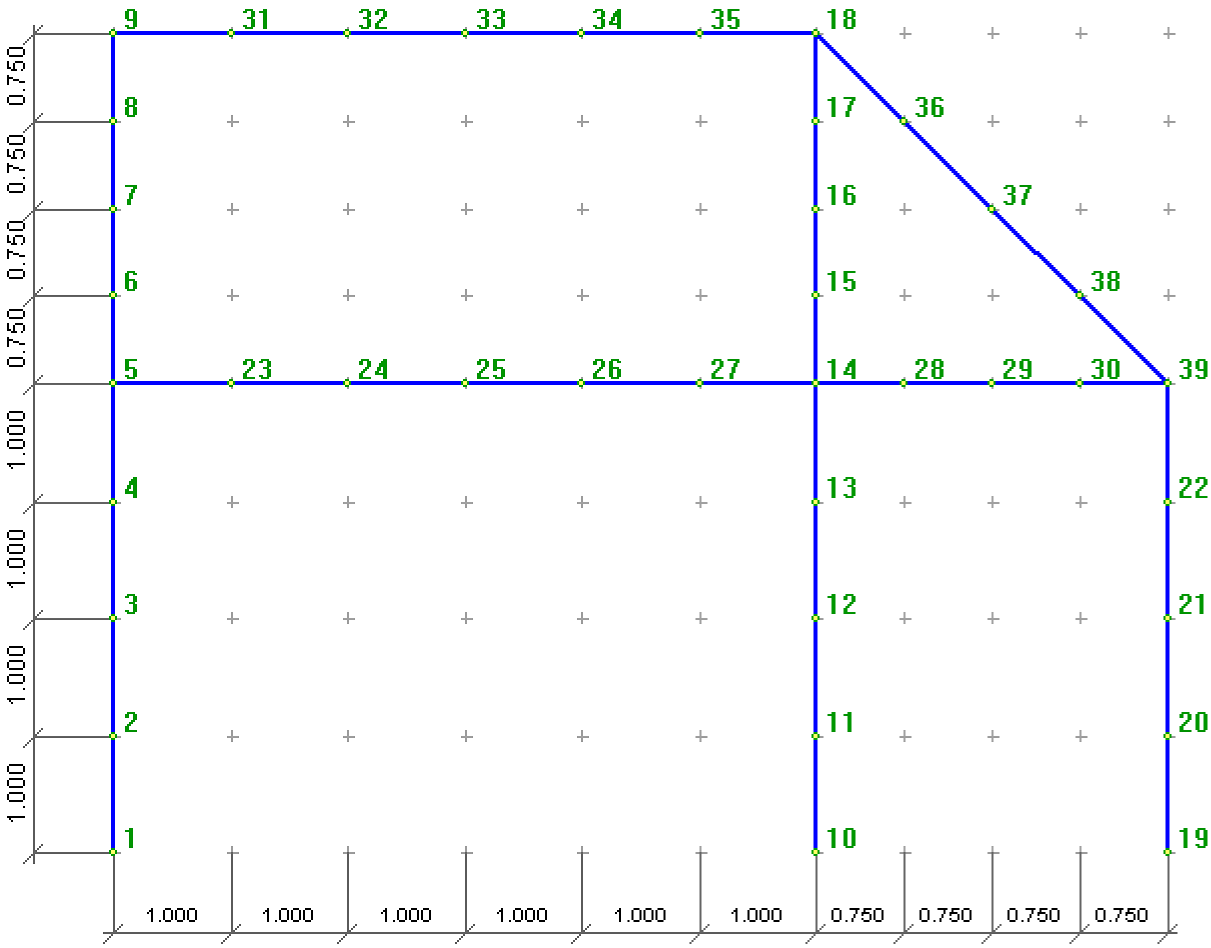

9
31
32
33
34
35
18
8
17
36
7
16
37
6
15
38
5
23
24
25
26
27
14
28
29
30
39
4
13
22
3
12
21
2
11
20
1
10
19
0.750
0.750
0.750
0.750
1.000
1.000
1.000
1.000
1.000
1.000
1.000
1.000
1.000
1.000
0.750
0.750
0.750
0.750

Figure 15.6 a)

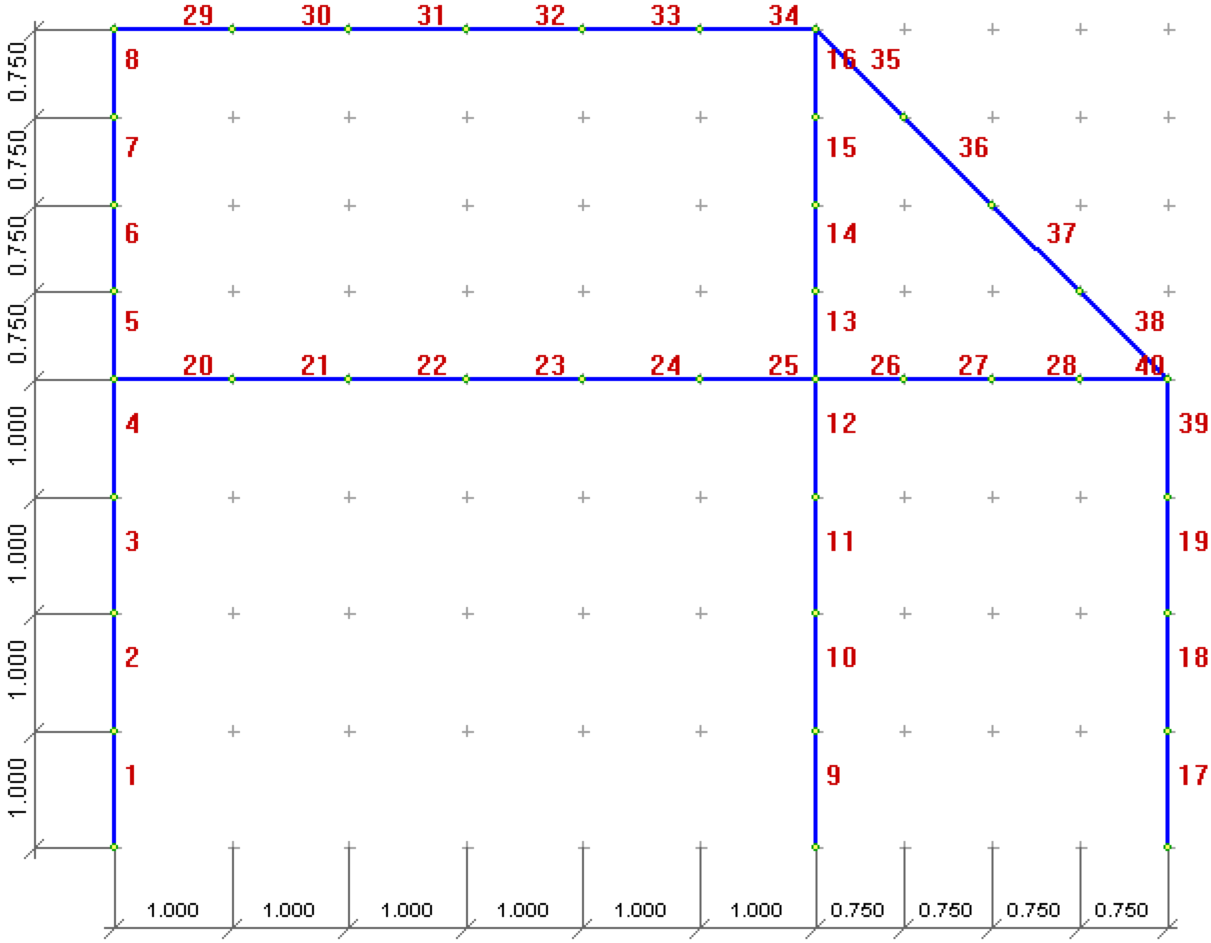


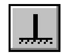
Figure 15.6 b)

Step 2. Define boundary conditions (restraints)

To define the type of restraint, on the MODEL menu, point to **Add restraint** and click appropriate command from the list or click necessary button on the toolbar.

Notation for restraints on the toolbar (except **Restrain all** command) correspond to restraints in the plane frame. The Restrain all option means restraints at all seven degrees of freedom. To define nonstandard restraints, on the MODEL menu, point to **Add restraint** and click **Arbitrary**. In the **Assign restraints** dialog box, select appropriate check boxes and click **Apply**.

Define the nodes where the restraint should be imposed on.

In this example, select **Restrain all** option (button  on the toolbar) and define two extreme bottom nodes. Then on the EDIT menu, click **Select nodes** (button  on the toolbar) and select the middle node. On the MODEL menu, point to **Add restraint** and click **Arbitrary**. In the **Assign restraints** dialog box, select X, Y, Z, Ud check boxes and click **Apply**.

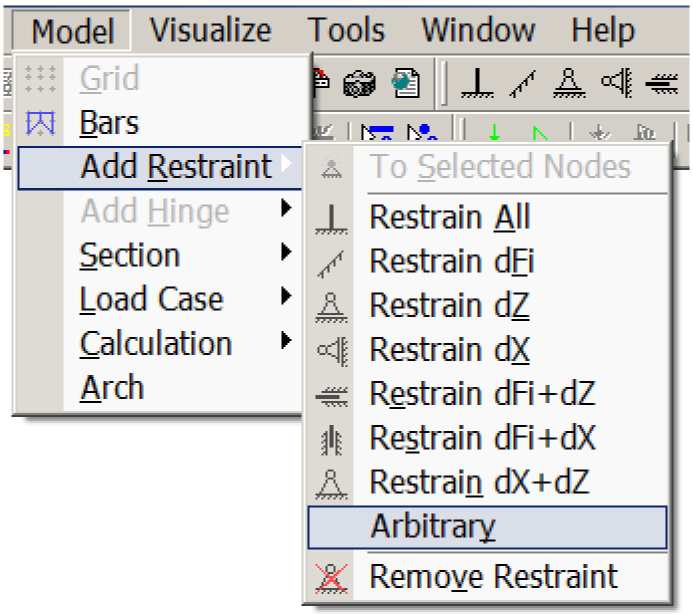


Figure 15.7 a)

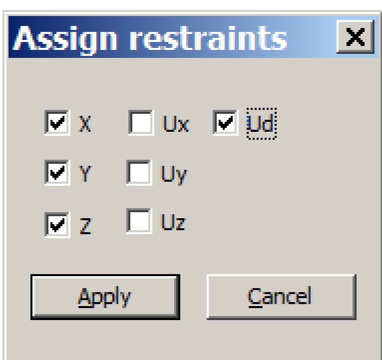


Figure 15.7 b)

Step 3. Define sections for thin-walled bars

To define section of bars, on the MODEL menu, point to **Section** and click **Select section**. (button  on the toolbar). In the **Cross-section** dialog box, click **Add**.

In this case, the window of Parametric thin-walled sections module will be displayed on the screen. The work with this program is described in this manual.) Select the **C-shape**, define its properties and click **OK**.

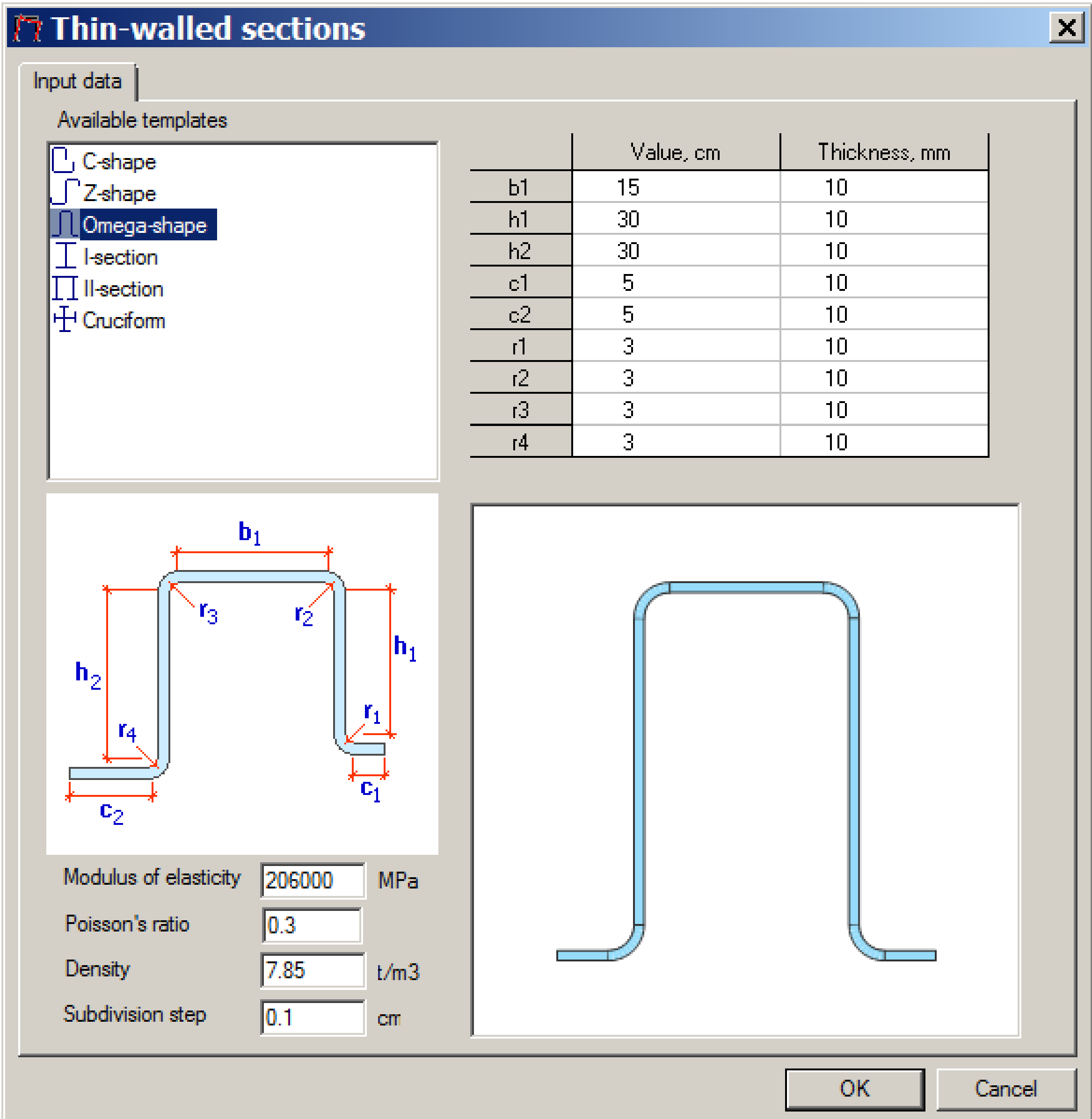


Figure 15.8

After calculation, the **Cross-section** dialog box appears on the screen. In the list of sections, select the **C-shape** and click **Set as current**. Then click **Apply**.

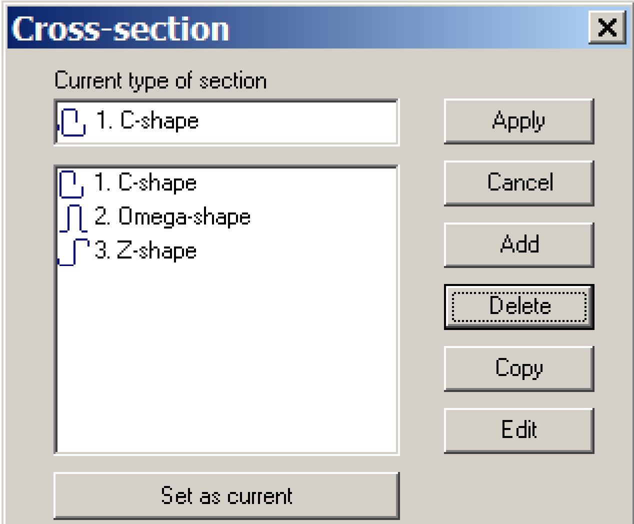


Figure 15.9

To assign current section to the bars of the model, you could follow one of the sequence presented below.
1) the first sequence of commands:

– On the EDIT menu, click **Select bars** (button on the toolbar).
– Select with the pointer appropriate bars.

– On the MODEL menu, point to **Section** and click **Assign section** (button on the toolbar).

– On the MODEL menu, point to **Section** and click **Apply to selected bars** (button on the toolbar).

2) the second sequence of commands:

– On the MODEL menu, point to **Section** and click **Assign section** (button on the toolbar).
– Select appropriate bars in order to assign the section to them.
– Click **Assign section** once again to make this command not active.

The first method is useful when you define section for many bars.
The second sequence - when you define the section for certain selected bars.

To define numerical stiffness properties, in the **Cross-section** dialog box, hold down the SHIFT key and click **Add**. In the **Numerical** dialog box, define appropriate values.

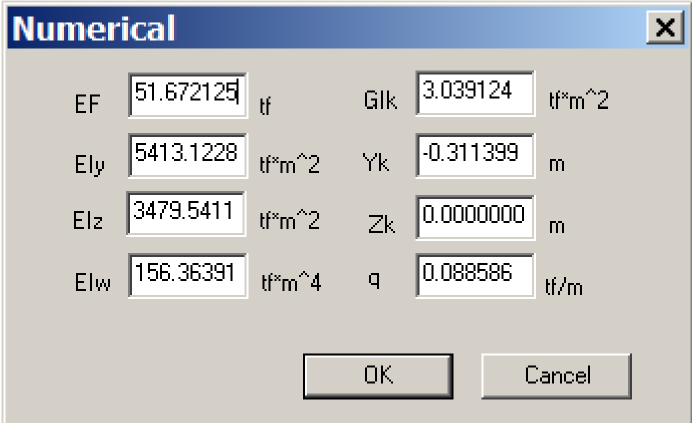


Figure 15.10

EF – axial stiffness;
EIy – flexural stiffness in plane;
EIz – flexural stiffness out of plane;
EIw – warping stiffness;
GIk – torsional stiffness;
Yk, Zk – coordinates of rigidity centre in the Y1, Z1 axes;
q – unit weight.

To display (**Numerical** dialog box) and edit stiffness values (computed in the Parametric thin-walled sections module), in the **Cross-section** dialog box, hold down the SHIFT key and click **Add**.

Step 4. Define loads
Let's define two load cases:
1) dead weight;
2) loads at nodes and bars of design model.

To define dead weight, on the MODEL menu, point to **Load case** and click **Add dead weight**. If material density (or unit weight) is not defined, then you will see appropriate warning and the dead weight will not be added.

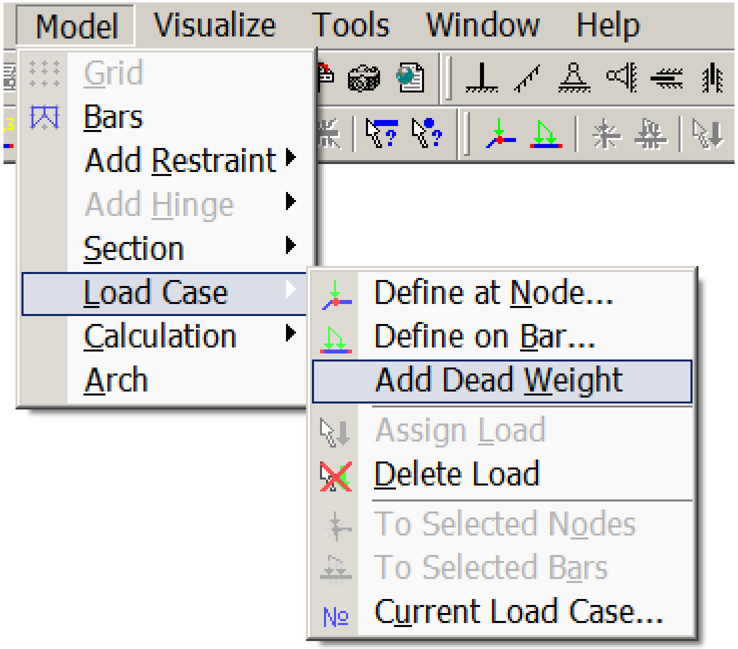


Figure 15.11

Load for the first load case in presented in the figure below.

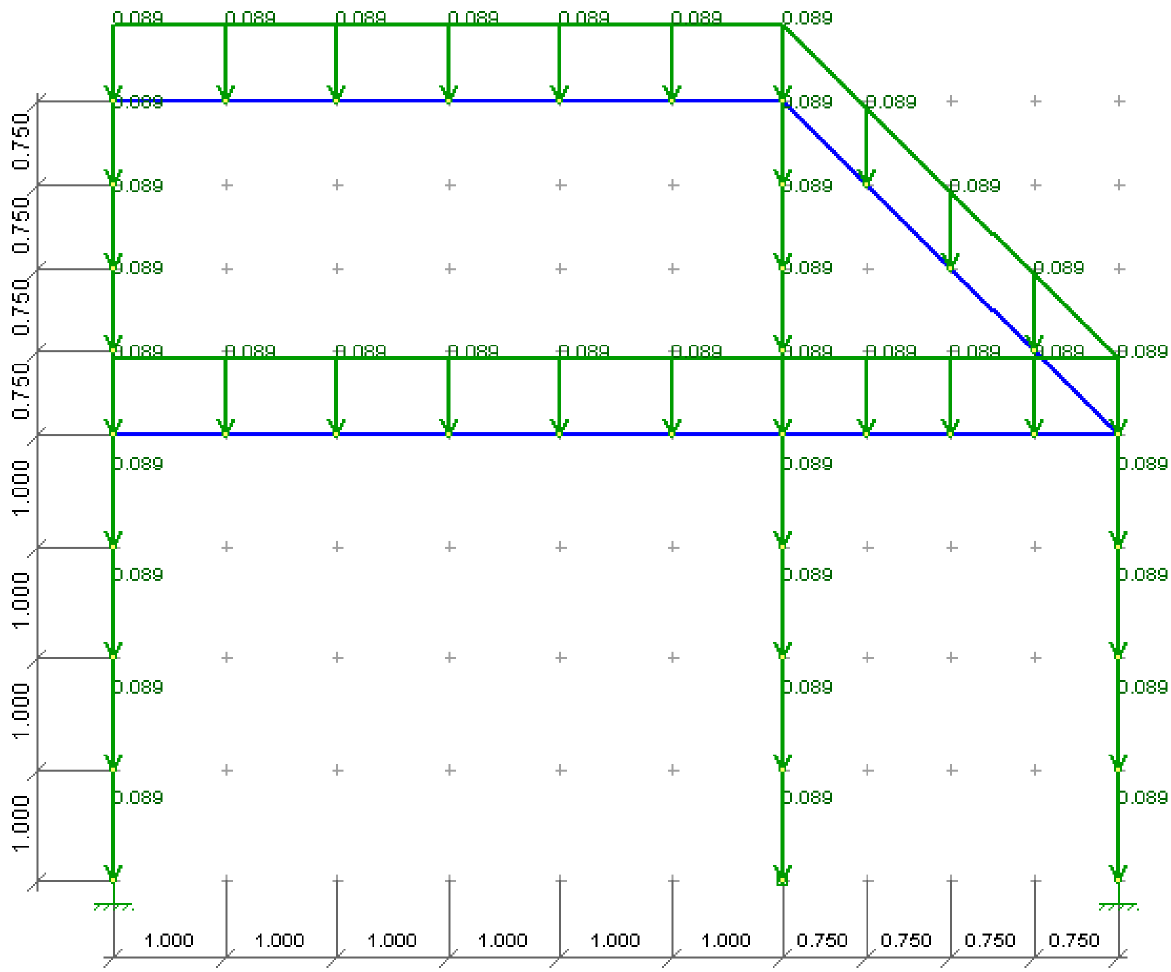


Figure 15.12

To define the second (the third, the fourth, ...) load case, on the MODEL menu, point to **Load case** and click **Current load case** (button  on the toolbar).
Load for the second load case is presented in figure below.

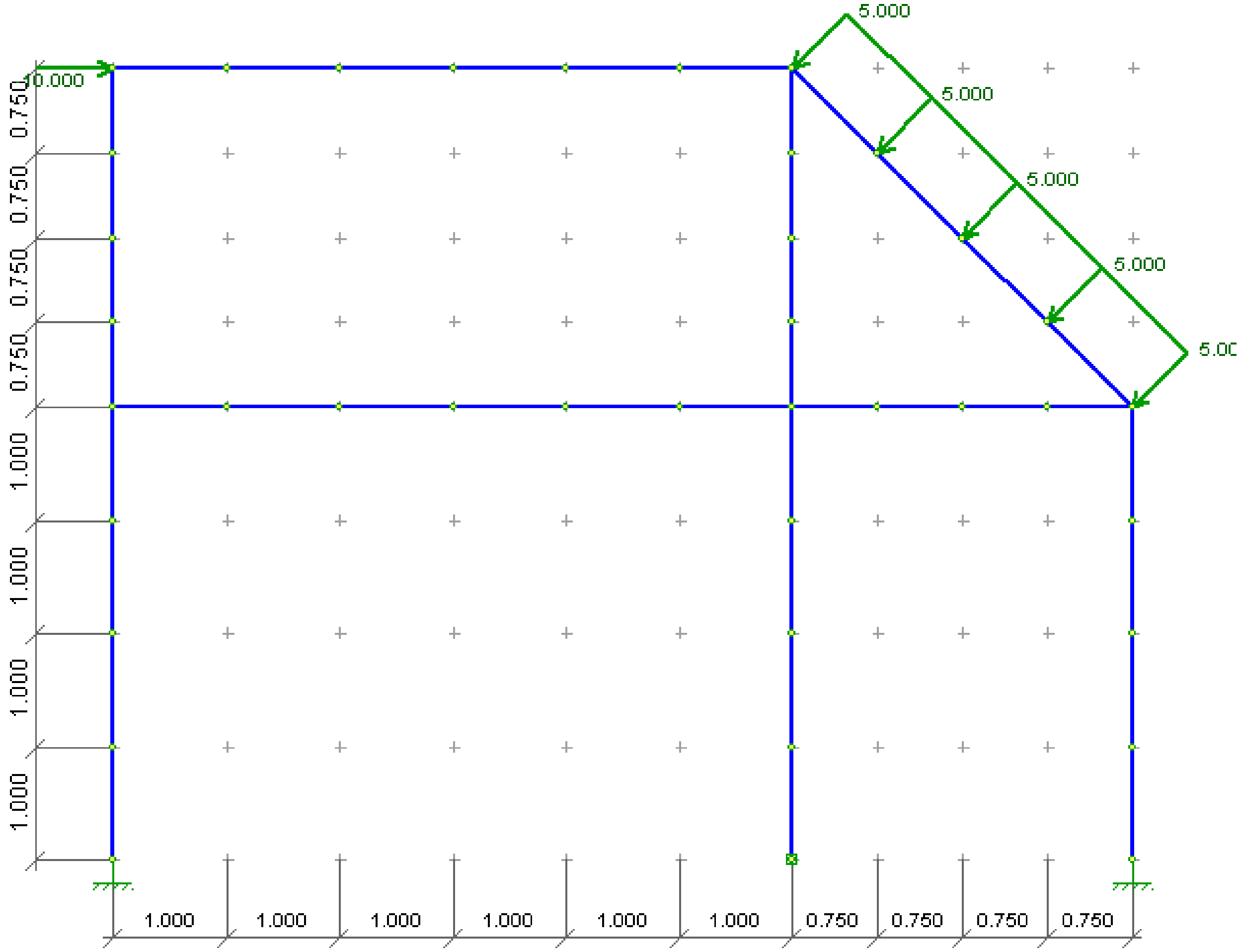


Figure 15.13

In the dialog box, define the number of the current load case.
To define nodal loads and initial displacements, on the MODEL menu, point to **Load case** and click **Define at node** (button on the toolbar).

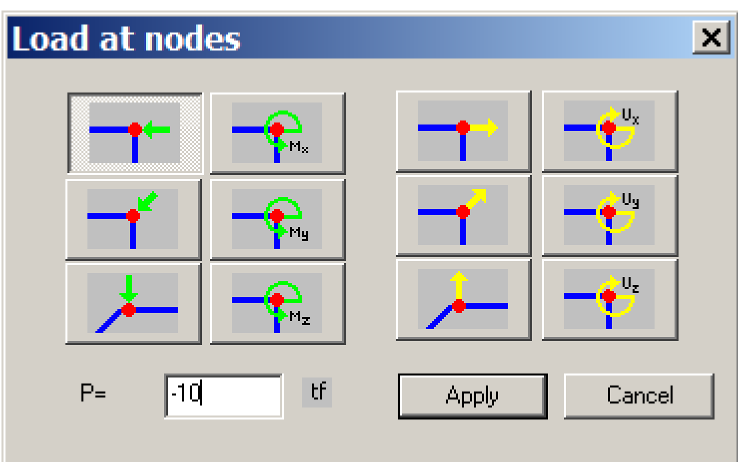


Figure 15.14

In the **Load at nodes** dialog box, define horizontal load P = – 10 tf. Click **Apply**.

Then on the MODEL menu, point to **Load case** and click **Assign load** (button on the toolbar) and define the node where load should be applied to (in this example it is the upper left node).

In the **Load at nodes** dialog box, buttons with green arrows corresponds to loads relative to approximate axis in global coordinate system while buttons with yellow arrows - to cinematic load (initial displacements of nodes). One important point to remember is that initial displacements of nodes may be assigned only to nodes in which there are no restraints.

To define loads on bars (or span loads), on the MODEL menu, point to **Load case** and click **Define on bar** (button on the toolbar).

In the **Load on bars** dialog box, define uniformly distributed load on inclined bars along the whole length in local coordinate system along the Z1-axis. The load value should be 5 tf/m. Click **Apply**.

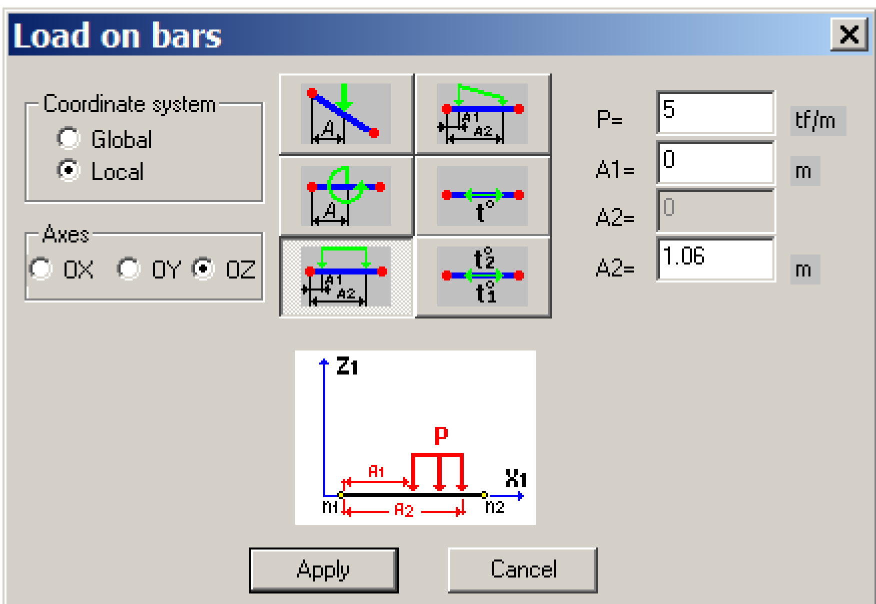


Figure 15.15

When all loads are defined, click **Assign load** button once again in order to make this command not active.

In the **Load on bars** dialog box it is possible to define the following types of loads relative to global or local axes X(X1), Y(Y1), Z(Z1) at certain distance from beginning of the bar:

– concentrated load;
– concentrated moment;
– uniformly distributed load;
– trapezoidally distributed load;
– uniform heating along the local X1-axis;

– nonuniform heating of upper and lower (relative the Z1-axis) fibres of the section.
To ensure the right distance for load, on the TOOLS menu, use the **Length** command (button on the toolbar) in order to measure the length of inclined bar.

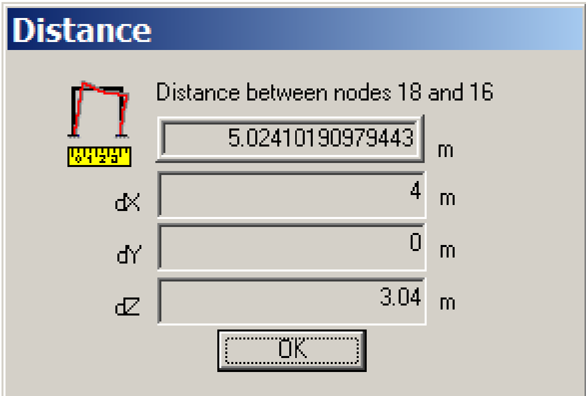


Figure 15.16

To start calculation, on the MODEL menu, point to **Calculation** and click **Analyse problem** (button on the toolbar).
When you obtain message that calculation is complete, it is possible to evaluate analysis results.

Options to edit the model

Principal commands (modes) to edit the model are presented on the EDIT menu.

Undo
Select nodes
Select bars
Delete nodes
Delete bars
Remove restraint
Delete hinge
Delete load (MODEL menu / **Load case**)

Options to visualize the model

All commands to visualize the model are presented on the VISUALIZE menu. With the help of these commends you could show or hide the following data:

Numbers of nodes
Numbers of bars

Dimensions of the specified grid
Restraints
Hinges
Loads
Load values

Tools
Length between two nodes
Angle by three nodes .
Information about bar
Information about node

Zoom - to enlarge the model fragment
Fit in window - to display all objects on the screen

The EDIT menu contains the **Pack model** command. Use this command to delete nodes and elements that were eliminated from design model by the user.
Important. After the pack procedure all previous **Undo** and **Redo** actions will be unavailable. The **Undo/Redo** history will start once again from this procedure.

Circular arc

On the MODEL menu, click **Arch**. In the dialog box (see Fig. 15.18), define coordinate of the centre of circle (Xo, Yo), radius of arch (R), number of bars (N), initial and final angles (A1, A2) in degrees (measured anti-clockwise from the X-axis of the global coordinate system when you look at the screen). Click **Apply**.

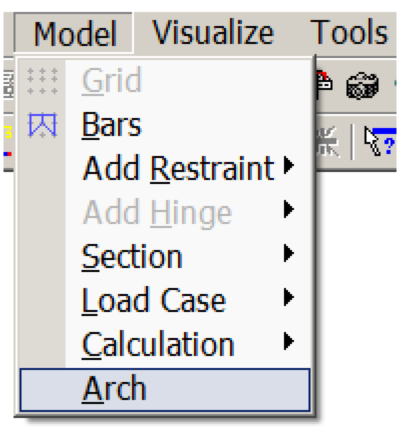


Figure 15.17

When you define parameters of arc (as presented in Figure 15.18a), the arc will be presented on the screen. Nodes and elements of the arc will be numbered from right to left. To obtain automatic numbering of nodes and elements from left to right, specify A1 = –180° and A2 = –360°.

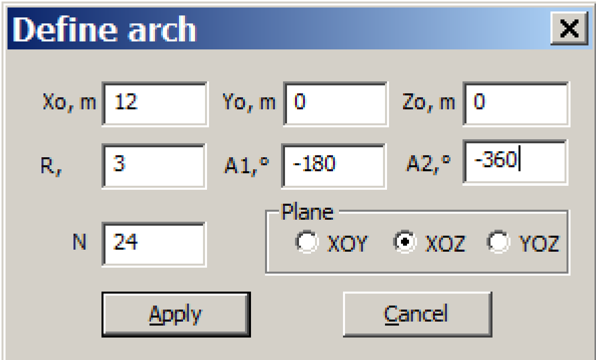


Figure 15.18 a)

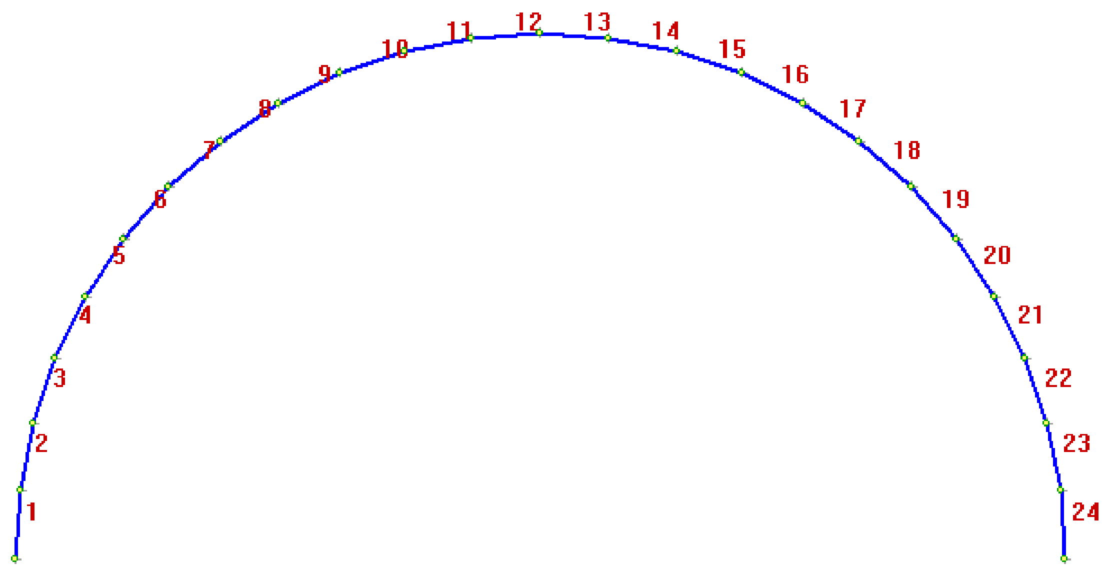


Figure 15.18 b)

Stiffness properties, restraints and loads are defined as described earlier.
In the picture below you will see combination model with dead weight applied to it.

 www.liraland.com

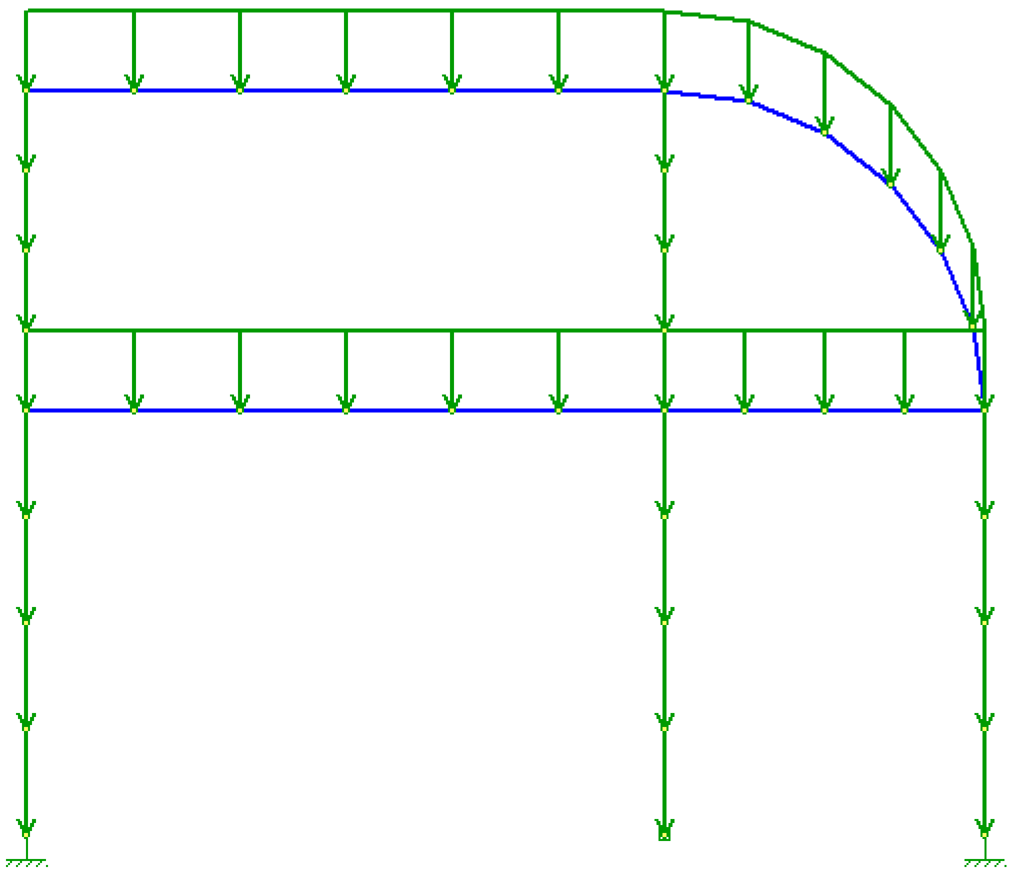


Figure 15.19

Output data

Define the number of the load case that you would like to evaluate (button on the toolbar).

To display the output data, on the MODEL menu, point to **Calculation** and click one of the following commands to display appropriate data on the screen:

– Deformed shape;
– Axial force diagram N;
– Shear force diagram Qz, Qy ;
– Moment diagram Mx, My, Mz ;
– Warping moment diagram B.

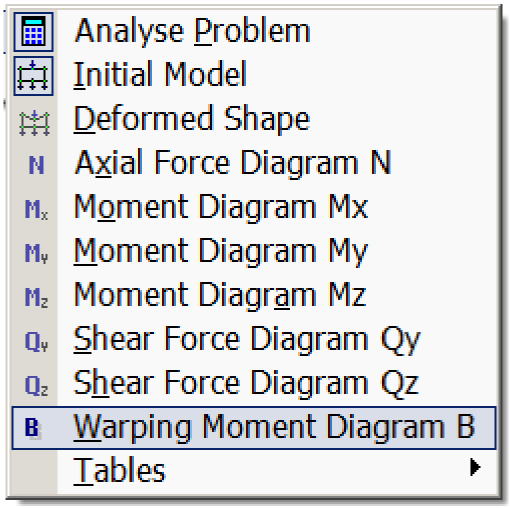


Figure 15.20

To display numerical values of nodal displacements and forces in bars, on the MODEL menu, point to **Calculation / Tables** and click **Nodal displacements**  or **Internal forces at bar ends**  .

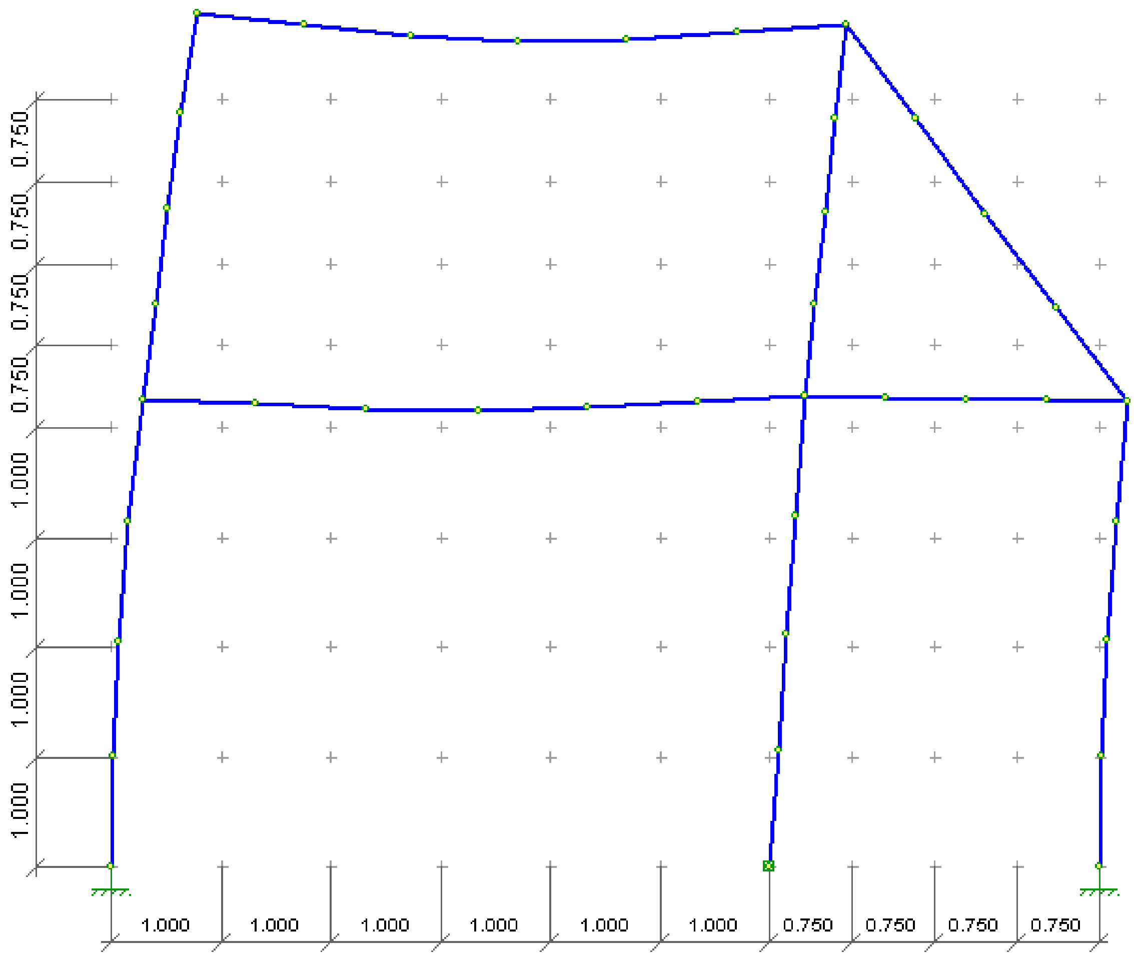


Figure 15.21 a) Deformed shape from load case 1

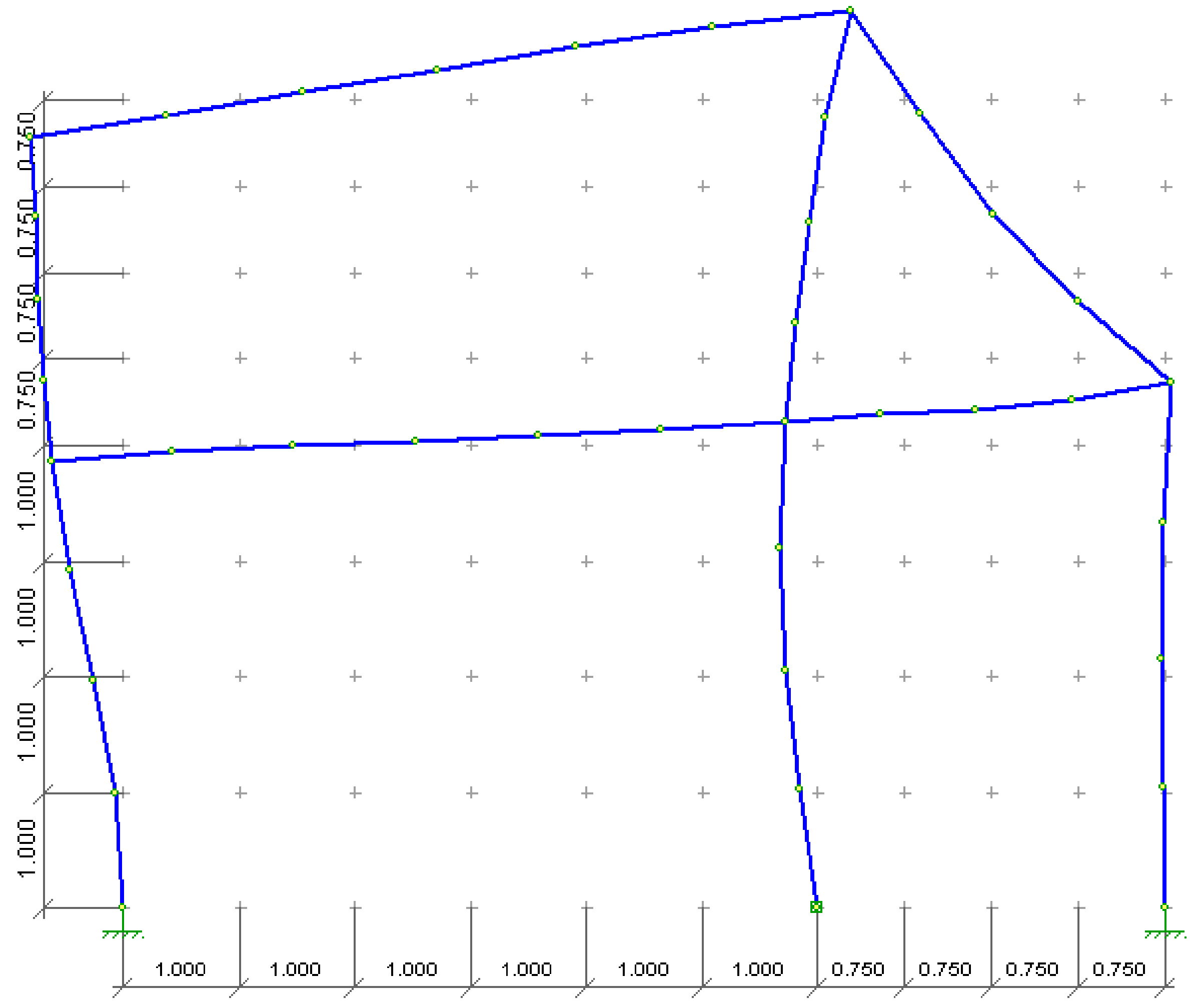


Figure 15.21 b) Deformed shape from load case 2

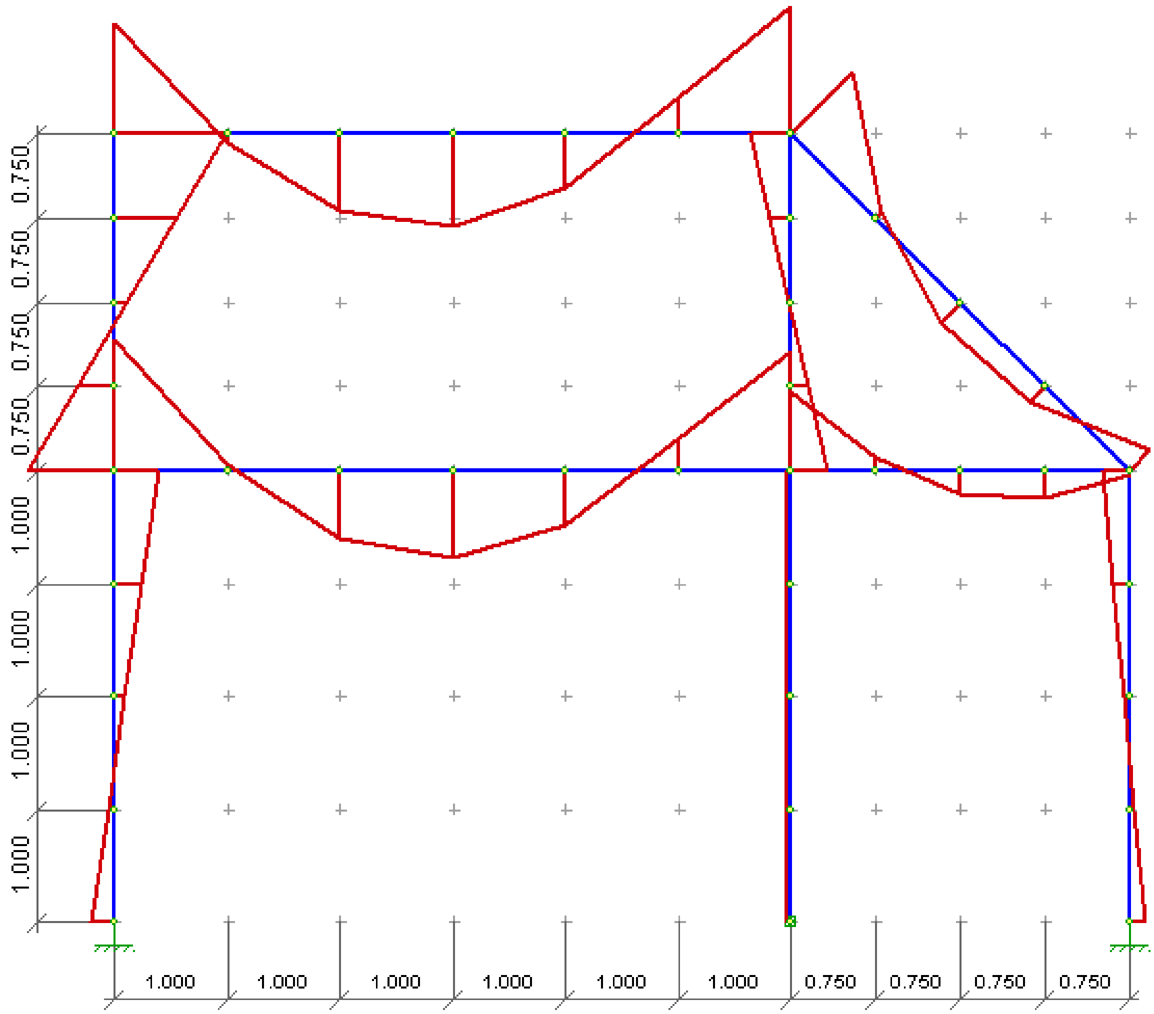


Figure 15.22 a) Diagram of warping moments B from load case 1

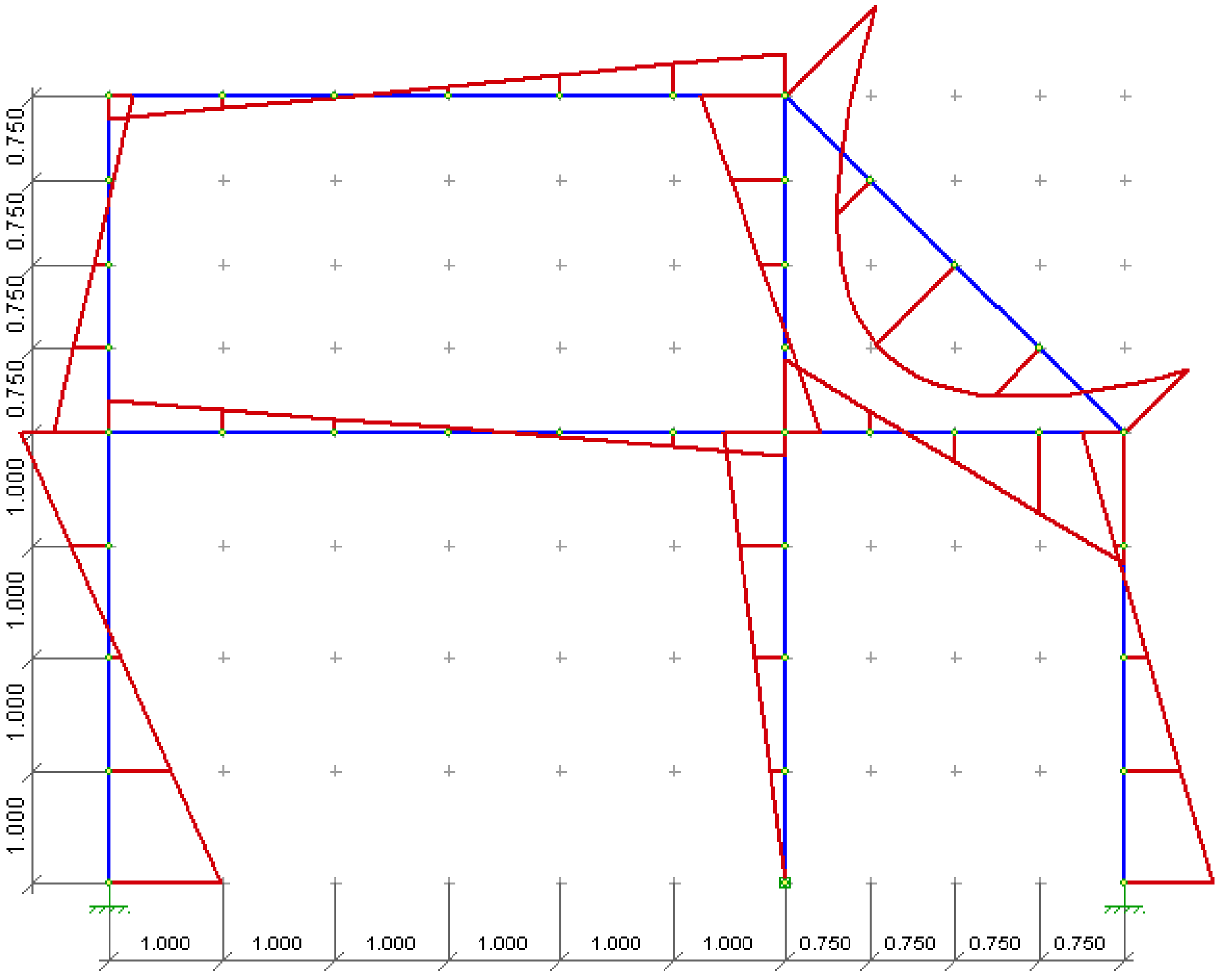


Figure 15.22 b) Moment diagram My from load case 2

To present numerical values of nodal displacements or forces in bars, on the MODEL menu, point to **Calculation**. Then point to **Tables** and click either **Nodal displacements** (button on the toolbar) or **Internal forces at bar ends** (button on the toolbar).

Displacement/Load case 1

File

N...	X (mm)	Y(mm)	Z (mm)	Ux(rad...	Uy (ra...	Uz(rad...	Ud(ra
1	0	0	0	0	0	0	0
2	2.26212	-6.29733	-0.125...	0.7173...	1.0736	0.7587...	-0.027
3	0	0	0	0	0	0	0
4	2.19589	-1.22416	-0.129...	-0.045...	0.0575...	0.6082...	-0.038
5	0	0	0	0	0	0	0
6	1.96547	-0.175...	-0.165...	-0.258...	0.5635...	0.37035	0.007
7	0	0	0	0	0	0	0
8	1.00751	-6.20984	-0.055...	1.71536	0.79128	0.69958	-0.056
9	0	0	0	0	0	0	0
10	0.9472...	-1.21499	-0.111...	0.56565	-0.107...	0.7544...	-0.048
11	0	0	0	0	0	0	0

Figure 15.23 a)

Forces/Load case 1

File

Ele...	Sec...	N (tf)	Qy(tf)	Qz (tf)	Mx(tf*m)	My (tf...	Mz(tf*m)	B(tf*m2)
1	1	-9.67...	-5.84...	-0.95...	-0.50...	2.622...	-9.36...	-0.12...
1	2	-9.67...	-5.84...	-0.95...	-0.50...	-0.25...	8.160...	0.144...
2	1	-9.99...	-1.44...	-3.10...	-0.40...	4.717...	-2.1305	-0.09...
2	2	-9.99...	-1.44...	-3.10...	-0.40...	-4.59...	2.206...	0.124...
3	1	-12.7...	-0.62...	-1.64...	-0.24...	3.092...	-0.72...	-0.06...
3	2	-12.7...	-0.62...	-1.64...	-0.24...	-1.84...	1.161...	0.056...
4	1	-4.31...	-4.06...	0.263...	-0.47...	0.477...	-7.54...	-0.10...
4	2	-4.31...	-4.06...	0.263...	-0.47...	1.2687	4.664...	0.153...
5	1	-8.55...	-0.41...	-1.62...	-0.50...	2.326...	-1.08...	-0.11...
5	2	-8.55...	-0.41...	-1.62...	-0.50...	-2.56...	0.140...	0.157...
6	1	-10.6...	0.396...	-0.92...	-0.08...	1.645...	0.302...	-0.01...
6	2	-10.6...	0.396...	-0.92...	-0.08...	-1.13...	-0.88...	0.031...
7	1	-3.82...	-1.1041	2.700...	0.368...	0.127...	-2.11...	0.109...
7	2	-3.82...	-1.1041	-3.29...	0.368...	-4.07...	2.302...	-0.10...
8	1	-3.47...	-0.98...	3.286...	0.553...	-1.14...	-2.00...	0.150...
8	2	-3.47...	-0.98...	-2.71...	0.553...	-2.99...	1.935...	-0.15...
9	1	5.054...	0.784...	6.9758	0.126...	-7.79...	1.6063	0.034...
9	2	5.054...	0.784...	-1.0242	0.126...	4.1106	-1.5318	-0.00...
10	1	0.529	0.697	4.634	0.067	-3.42	1.3021	0.006

Figure 15.23 b)

Use the **Information about bar** button ( ) to preview force values selectively, generate diagrams of longitudinal and transverse forces, moments and displacements.
Use the **Information about node** button ( ) to preview nodal displacements selectively.

The figures below illustrate design model and analysis results.

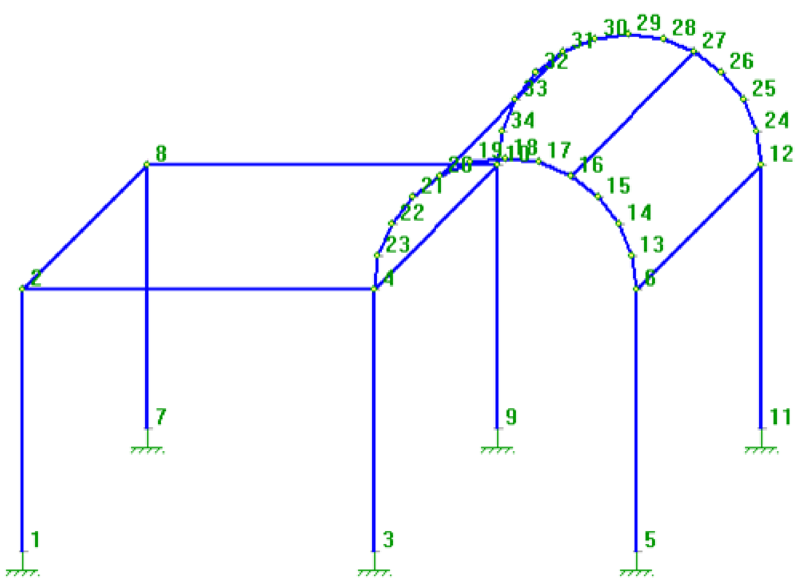


Figure 15.24. Nodes

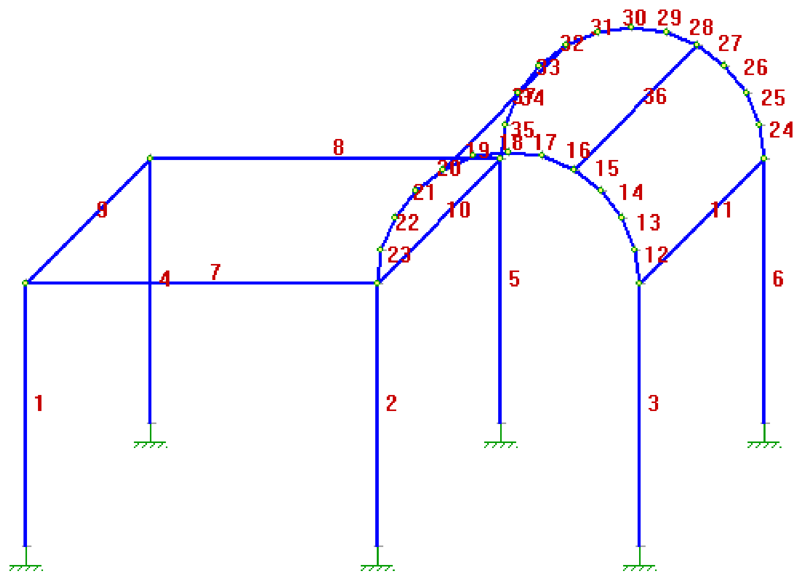


Figure 15.25. Elements

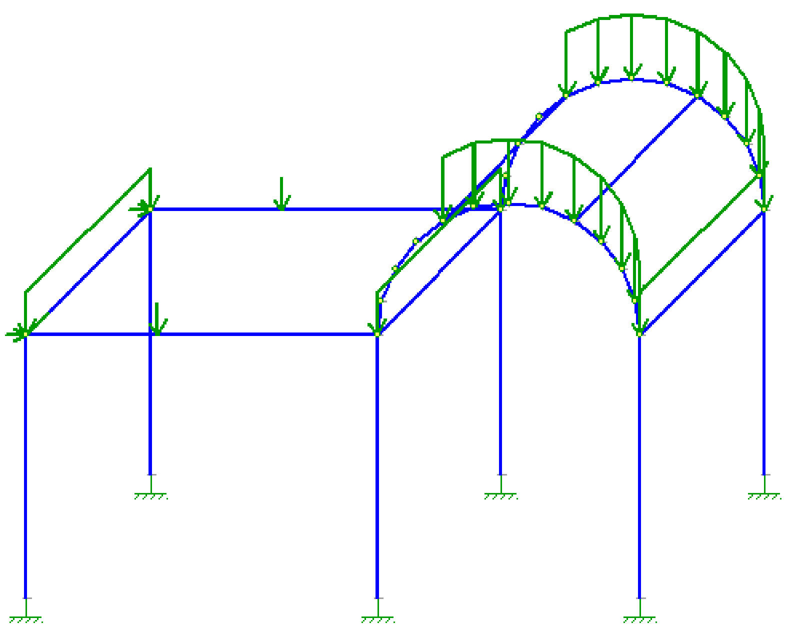


Figure 15.26. Loads

 www.liraland.com

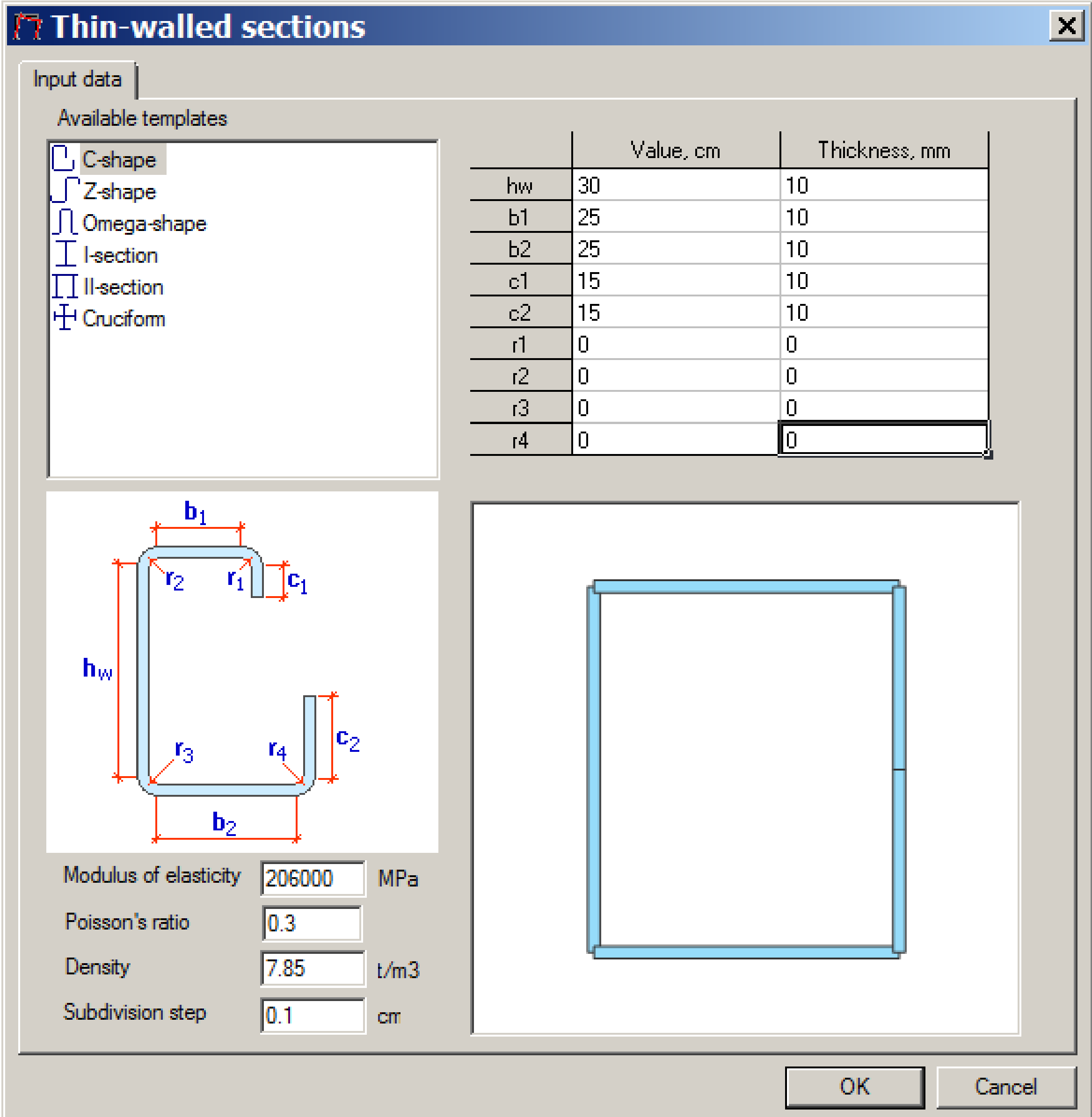


Figure 15.27. Section

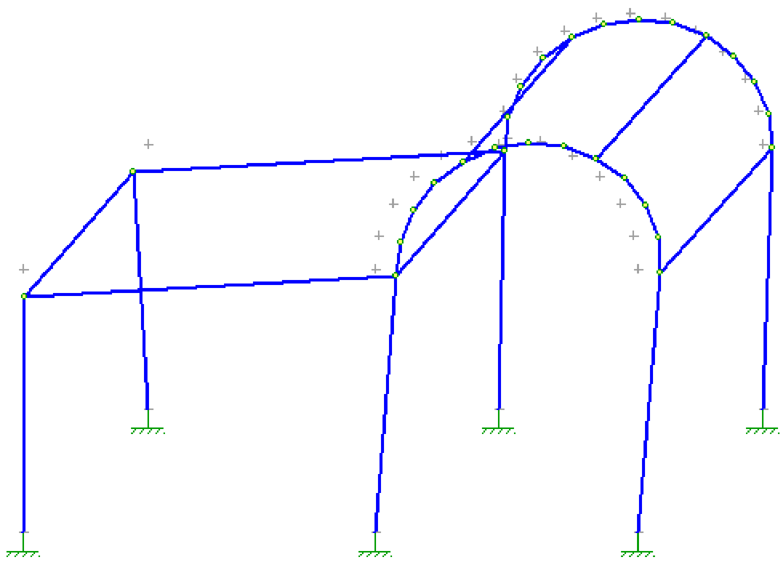


Figure 15.28. Deformed shape

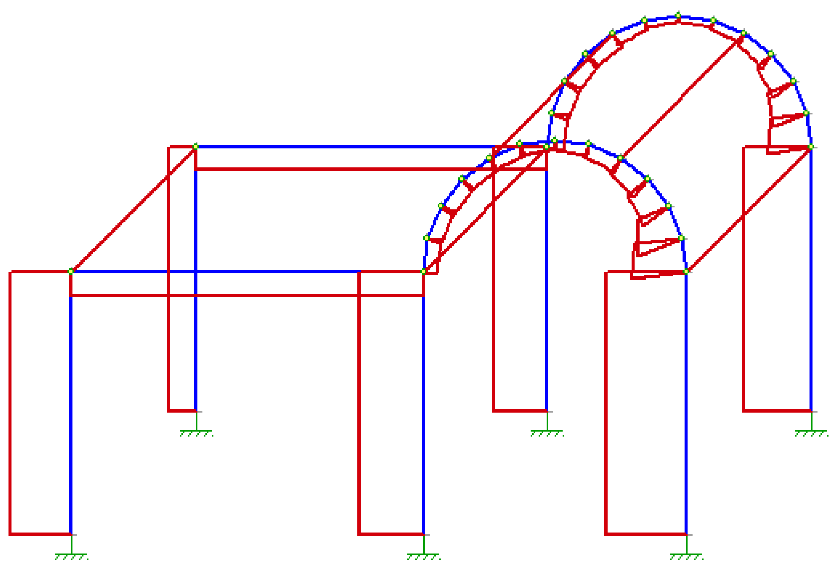


Figure 15.29. Diagram N

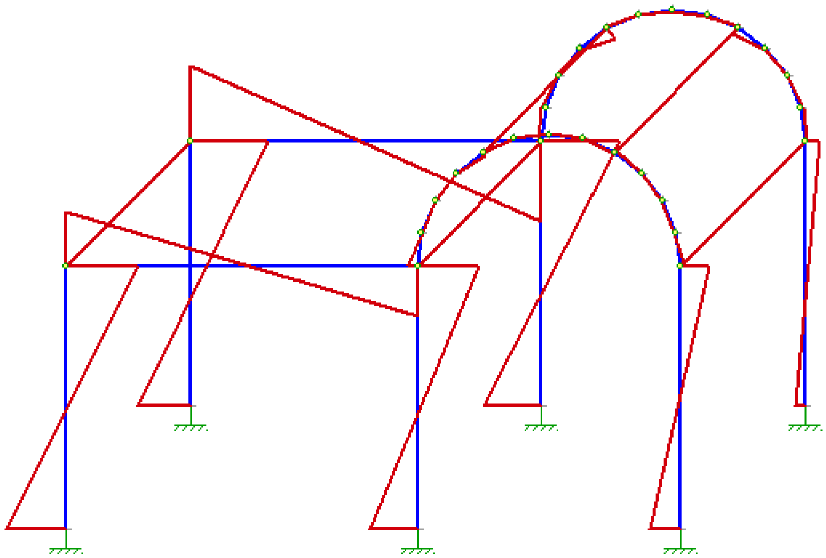


Figure 15.30. Diagram B

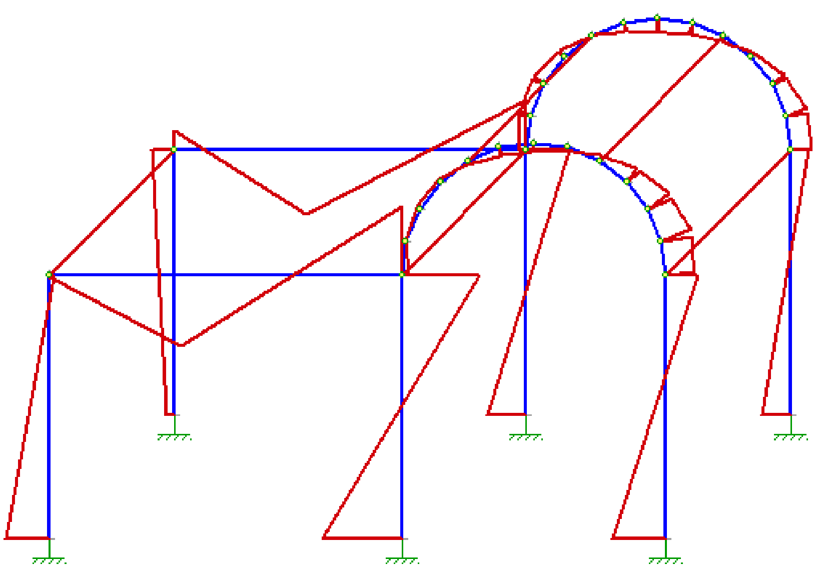


Figure 15.31. Diagram My

Sign convention

Sign convention for displacements

Linear translations (X, Y, Z) are considered positive if they are directed along appropriate axes.
Rotations (UX, UY, UZ) are considered positive if it rotates anti-clockwise when you look from the end of appropriate axis. Warping (Ud) is considered positive if positive rotation UX increases from beginning of the bar up to its end.

Sign convention for forces

Axial forces (N) is considered positive if the bar is in tension.
Sign convention for other forces relates to the bar section that belongs to its end:
– positive bending moment (Mx) rotates anti-clockwise when you look from the end of the X1-axis;
– positive bending moment (My) rotates anti-clockwise when you look from the end of the Y1-axis;
– positive bending moment (Mz) rotates anti-clockwise when you look from the end of the Z1-axis;
– positive shear force (Qy) acts along the Y1-axis;
– positive shear force (Qz) acts along the Z1-axis;
– positive warping moment (B) is presented with pair of moments MB(top) and MB(bot) that rotate about the Z1-axis; MB(top) is applied to the top (positive direction of the Z1-axis) edge of the section while MB(bot) is applied to the bottom (negative direction of the Z1-axis) edge; in this case MB(top) is directed anti-clockwise while MB(bot) - clockwise when you look from the end of the Z1-axis.

Prestressing

Prestressing (Chapter)

The chapter contains modules that enable you to analyse and check bearing capacity of sections of reinforced concrete bars with prestressed reinforcement.

Prestressing
Analysis of RC support sections (posts)

Figure 16.1

Prestressing

This module enables you to analyse and check bearing capacity of reinforced concrete sections of bars with prestressed reinforcement. In the *check* mode of the program it is possible to compute reserve factor for the section. In the *selection* mode of the program it is possible to determine area of lower and upper reinforcement and appropriate prestressing.

The following building codes are supported:

– SNIP 2.03.01-84* Concrete and reinforced concrete structures;
– SP 52-102-2004 Prestressed reinforced concrete structures;
– Eurocode 2 Design of concrete structures;
– SP 52-104-2006 Steel fiber reinforced concrete structures.

Available classes of concrete: B12,5 B15 B20 B25 B30 B35 B40 B45 B50 B55 B60.
For Eurocode: C12 C16 C20 C25 C30 C35 C40 C45 C50 C55 C60 C70 C80 C90.

Process of hardening - natural hardening or thermal treatment (steaming, autoclave hardening).

Service conditions - natural humidity, water saturation, alternately water saturation and drying.

Strength of concrete in transfer Rbp is assigned as equal not less than 11 MPa. For rebars of class AVI, reinforcing cables of classes K-7 and K-19 and wire reinforcement without buttenheads - not less than 15,5 MPa. Moreover, strength of concrete in transfer should be not less than 50% of the strength for defined concrete class.

The following classes of reinforcement are available:
– A-I, A-II, A-III, A-IIIb (with monitoring elongation and stress) A-IIIb (with monitoring only elongation), A-IV, A-V, A-VI, Bp-I, B-II, Bp-II, K-7, K-19;
– A240, A300, A400, A500, A600, A800, A1000, B500, Bp1200, Bp1300, Bp1400, Bp1500, K1400, K1500.

Max diameter of reinforcement - 40 mm (for Eurocode - 50).

Distance to reinforcement - distance from gravity centre of reinforcement group up to the nearest side of section (lower reinforcement to the lower side, upper reinforcement - to the upper side).

Methods of prestress - mechanical, electrothermal, electrothermomechanical.

Only pretension of reinforcement onto stressing abutments is considered in the current version of the program.

Temperature drop is accepted as 65 degrees, if there is no accurate data.

Parameter **dl** - buckling of an anchor, is taken as equal to *2 mm*.

To take account of friction between the tendons and their sheathing, define parameters μ, κ, θ and χ (in SNIP μ corresponds to δ and κ corresponds to ω.

For calculation according to Eurocode, for prestressed reinforcement it is necessary to define shape, area of one rebar, characteristic tensile strength of prestressing steel *fpk*, characteristic 0.1% proof-stress of prestressing steel *fp0,1k* and class, indicating the relaxation behaviour - ordinary or low relaxation.

The following classes of fiber are available:

– steel fiber, milled from slabs, according TU 0882-193-46854090.
– steel fiber, cut from sheet steel, according to TU 0991-123-53832025.
– steel fiber, chopped from wire, according to TU 1211-205-46854090.

Input data

The dialog box is presented in the figure below.

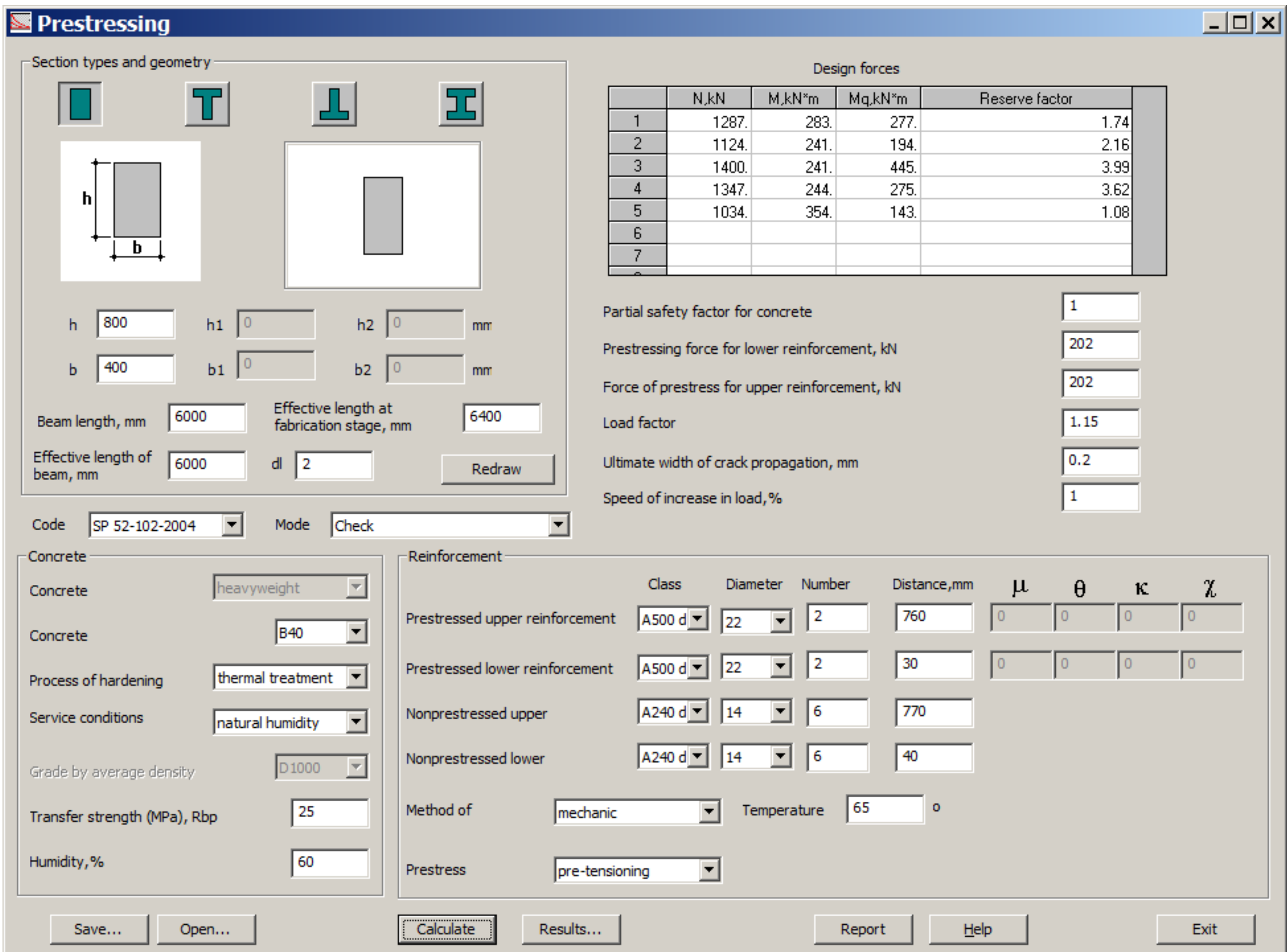


Figure 16.2

In the dialog box define dimensions for section and structure, parameters of concrete and reinforcement, design forces, prestress for the reinforcement.

Define type of section – rectangle, T-section with flange at the top, T-section with flange at the bottom, I-section. Select appropriate building code. Then define dimensions for the section, length of beam, class of concrete, classes of prestressed and nonprestressed reinforcement, number of rebars, their diameters and distances to them.

To check T-section or I-section with trapezoidal flanges, define average heights of flange overhangs.

To display schematic presentation pf the section according to specified dimensions, click **Redraw**.

Define design forces in the appropriate table. To convert to normative values, the load factor γ_f is applied in the program.
N - axial force applied at the gravity centre, kN;
M - moment from external forces, kN*m;
Mq - moment from dead weight of the structure in the section, kN*m.

Sign convention: positive moment corresponds to tension in lower fiber (according to schematic presentation of section), positive axial force corresponds to compression.

Partial safety factor for concrete γ_{b2} is defined depending on load type (by default 1).

Prestressing force if defined within the limits that depend on the class and area of reinforcement. In the mode of selecting prestressed reinforcement, recommended value of prestressing force for the certain reinforcement is determined by the program.

Speed of increase in load is required to determine reserve factor for the section. The speed shows which part of load should be added per one step. It is recommended to define the speed value within the limits 1-5%. If speed of increase in load is rather high, calculation accuracy of reserve factor for the section is reduced.

When the input data is defined, click **Calculate**.

Output data

Reserve factor for the section is displayed in the appropriate column in the **Design forces** table.

To display **Output data** for the calculation, click **Results**.

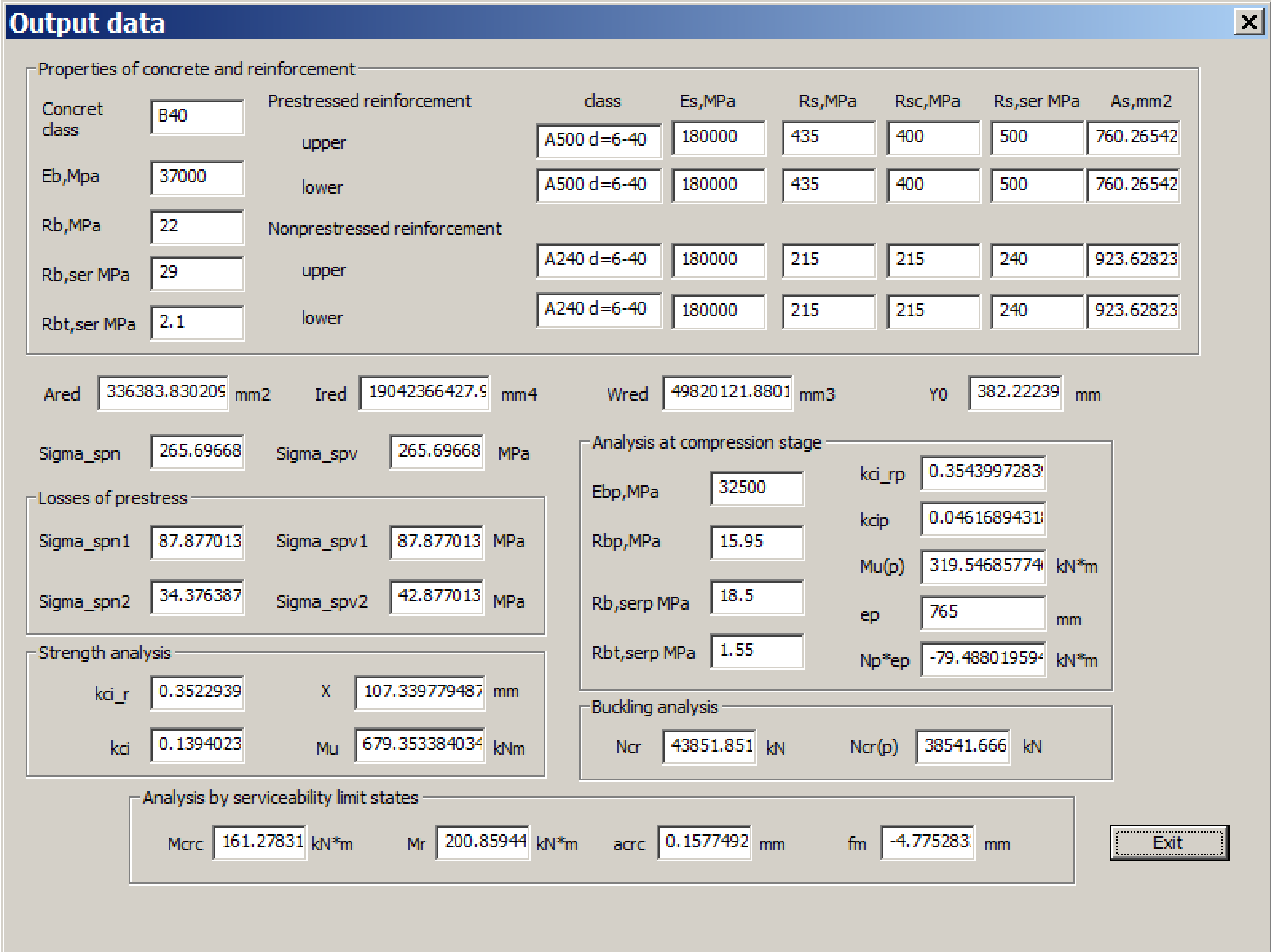


Figure 16.3

To save the model in **.pna* format, click **Save**.
To open the input data of the saved model, click **Open**. The calculation should be made once again.
To present a report document in HTML format, click **Report**.

Notation in output data:
Ared - area of equivalent section;
Ired - moment of inertia of equivalent section;
Wred - section modulus of equivalent section;
y0 - distance from lower fiber up to gravity centre of the section;
σspn - stress in lower prestressing reinforcement without account of losses;
σspv - stress in upper prestressing reinforcement without account of losses;
σspn1 - stress in lower prestressing reinforcement with account of immediate losses;
σspv1 - stress in upper prestressing reinforcement with account of immediate losses;
σspn2 - stress in lower prestressing reinforcement with account of all losses;
σspv2 - stress in upper prestressing reinforcement with account of all losses;
ξr - ultimate relative height of compression zone of concrete;
ξ - relative height of compression zone of concrete;
x - height of compression zone of concrete;
Mu - ultimate moment for the section;

Ncr - axial force in buckling;
Mcrc - moment in crack formation;
Mr - moment of external forces relative to the axis that is parallel to zero line and passes through the core;
acrc - max width of crack propagation;
ξrp - ultimate relative height of compression zone at fabrication stage;
ξp - relative height of compression zone at fabrication stage;
Mu(p) - bending moment that the section could take from prestressing;
Np*ep - max moment generated in the section during prestress;
ep - eccentricity of prestressing force;
Ncr(p) - critical prestressing force for reinforcement in buckling.

Analysis of RC support sections (posts)

The module enables you to analyse strength in reinforced concrete (RC) supports of ring sections with prestressed reinforcement.

The following building codes are supported:
— DBN V.2.6-98:2009 'Concrete and reinforced concrete structures. Basic provisions'.
— DSTU B V.2.6-156:2010 'Concrete and reinforced concrete structures from heavyweight three-component concrete'.

Materials
Concrete – heavyweight concrete of classes: C12/15, C16/20, C20/25, C25/30, C30/35, C32/40, C35/45, C40/50, C45/55, C50/60.
Prestressed reinforcement: A600, A600C, A600K, A800, A800K, A800CK, A1000, Bp1200, Bp1300, Bp1400, Bp1500, K1400 (K-7), K1500 (K-7), K1500 (K-19).
Nonprestressed reinforcement: A240, A400, A500, B500.
According to DSTU B.V.2.6-156:2010 sect.3.2.2.2 and sect.3.2.2.4, prestressed rebars may be applied as nonprestressed reinforcement.

Input data

Concrete: Class of concrete, strength of concrete at transfer **Rbp**, hardening process (natural hardening, thermal treatment) and stress-strain diagram for concrete.

Reinforcement: it is possible to define 2 reinforcement layers with different location for nonprestressed reinforcement and one reinforcement layer for prestressed reinforcement. For every layer, you define class of reinforcement, diameter of rebars, number of rebars, distance from rebar to external border of a pipe **a** and angle for distance relative to the Y-axis. For prestressed reinforcement it is necessary to define prestressing force.

Load: **N** – axial force (compression with sign '-'), **My** – bending moment relative to the Y-axis ('+' if the lower edge of section is in tension, '-' if the upper edge of section is in tension), **Mz** – bending moment relative to the Z-axis ('+' if the right edge of section is in tension, '-' if the left edge of section is in tension). It is possible to save the input data to the file or open previously saved problem. To do this, use appropriate buttons.

To start calculation, click **Calculate**.

Output data

When calculation is complete, you will obtain the following results:
– height of compression zone of concrete **x**,
– areas of nonprestressed **As** and prestressed **Ap** reinforcement,
– stress in reinforcement with account of all losses **Sigp**,
– short term and long term losses,
– width of crack propagation **a_crc**,
– distance between cracks **l_crc**.

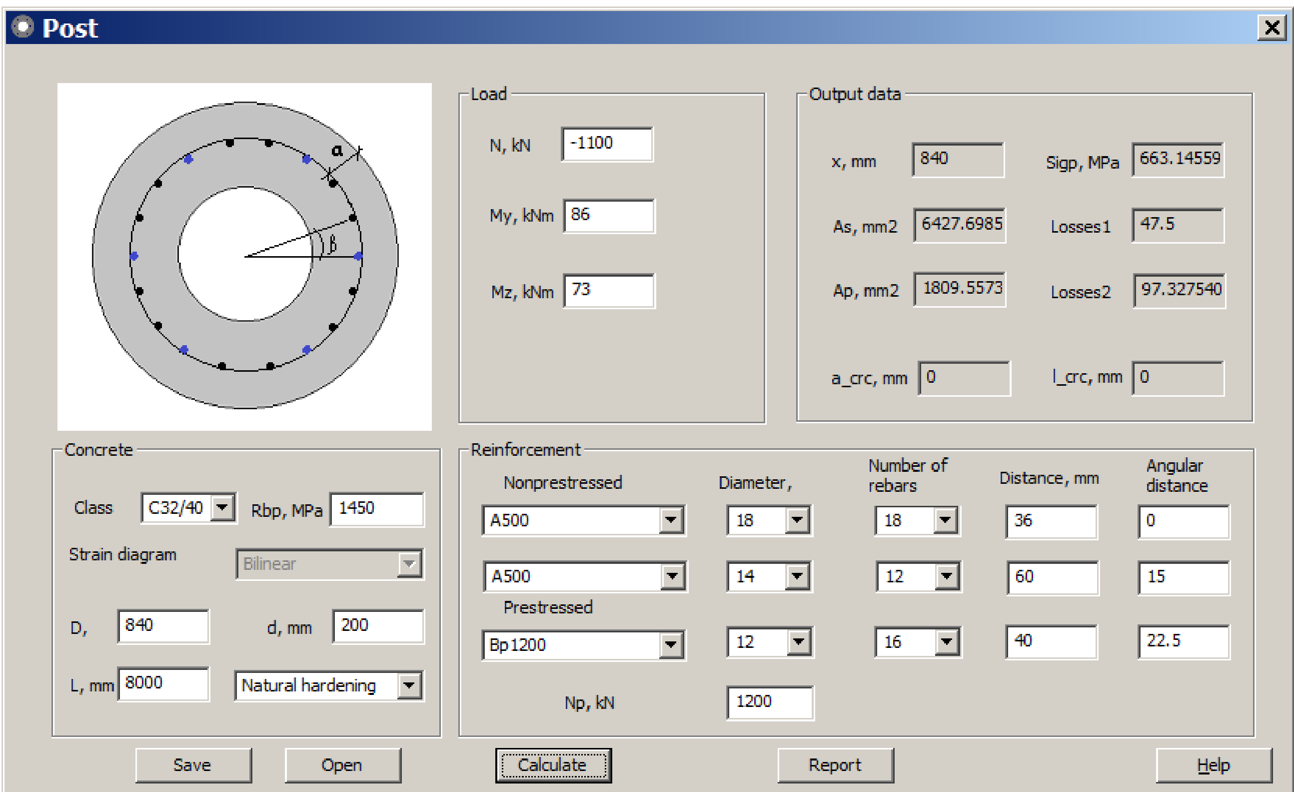


Figure 16.4

To present a report document in HTML format, click **Report**.

The report contains physical and mechanical properties of materials as well as the output data:
Prestressing force, kN – **Np**
Short term losses, MPa
Long term losses, MPa
Stress in prestressed reinforcement with account of all losses, MPa – **Sigp**
Percentage of reinforcement for section, **%**

Relative height of compression zone of concrete – **ksi**
Height of compression zone of concrete, mm – **x**
Width of crack propagation, mm – **acrc**
Max depth of cracks, mm – **hcrc**
Distance between cracks, m – **lcrc**
Stress in extreme tensile reinforcement, MPa – **Sigma_kr**
Relative strain in extreme compressed fibre of concrete – **EpsBMax**

Soil

Soil (Chapter)

The module enables you to calculate settlements of slab and/or pile foundations and subgrade moduli C1, C2 of soil under arbitrary contour of foundation slab or pile foundation grillage.

Moduli C1 and C2 according to soil model

Soil

Foundation slab or pile foundation grillage are presented as uniformly distributed load that correspond to the slab shape and is applied to the specified contour of soil model. Load may be applied to the whole contour of the model and to its separate zones (zones may have arbitrary shape in plan).

Every applied load may be defined as the load from existing building. In this case, the program computes influence that the settlement of buildings under construction has on the settlement of existing buildings. It is also possible to compute skews of existing buildings.

Computation is made according to automatically generated soil model. To generate soil model, define data about location of boreholes and properties of soil layers (GE) in them.

Contour of soil model in plan is determined by location of boreholes, overall dimensions of load contours as well as specified grids of axes for existing buildings and buildings under construction. Grids of axes are the supplementary element for model generation and, as a rule, this element determines overall dimensions of the construction site.

If when you generate the soil model, point (where subgrade moduli and other parameters should be calculated) is located between the specified boreholes, then soil model is generated by interpolation. In this case, interpolation zone is determined with the polygon of the specified boreholes. If the point is located outside the polygon of specified boreholes, then soil model is generated by extrapolation. Extrapolation zone is determined with overall dimensions of grids and loads outside the polygon of specified boreholes. If grids or loads are partially or completely defined outside the polygon of specified boreholes, then when extrapolation zone is determined on plan, additional boreholes will appear automatically.

The following building codes are supported:
SNIP 2.02.01–83*,
SP 50-101-2004,
DBN V.2.1-10:2009,
SP 22.13330.2011,
SP 24.13330.2011.

Analysis of settlements is carried out by the model of linear elastic half-space.

Input data

Soil properties

On the MODEL menu, click **Soil properties** (button  on the toolbar).

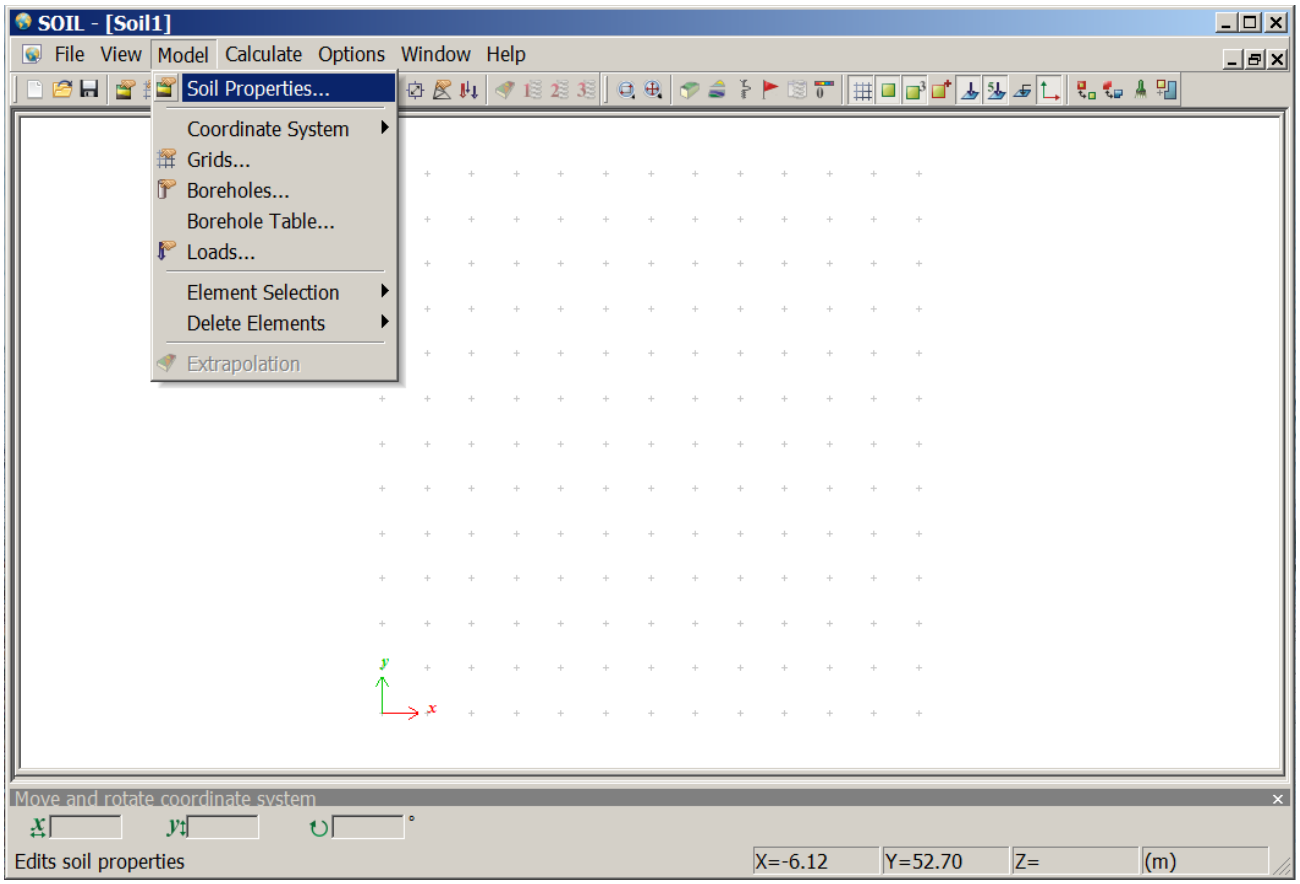
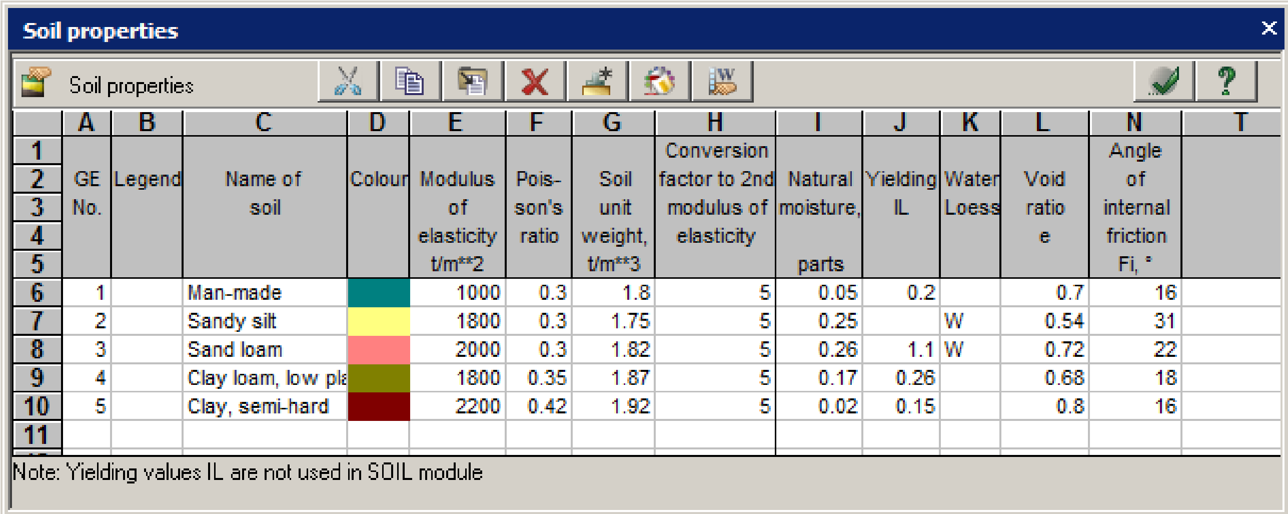


Figure 17.1

The **Soil properties** dialog box appears on the screen.



	A	B	C	D	E	F	G	H	I	J	K	L	N	T
1–5	GE No.	Legend	Name of soil	Colour	Modulus of elasticity t/m**2	Pois-son's ratio	Soil unit weight, t/m**3	Conversion factor to 2nd modulus of elasticity	Natural moisture, parts	Yielding IL	Water Loess	Void ratio e	Angle of internal friction Fi, °	
6	1		Man-made		1000	0.3	1.8	5	0.05	0.2		0.7	16	
7	2		Sandy silt		1800	0.3	1.75	5	0.25		W	0.54	31	
8	3		Sand loam		2000	0.3	1.82	5	0.26	1.1	W	0.72	22	
9	4		Clay loam, low pla		1800	0.35	1.87	5	0.17	0.26		0.68	18	
10	5		Clay, semi-hard		2200	0.42	1.92	5	0.02	0.15		0.8	16	
11														

Note: Yielding values IL are not used in SOIL module

Figure 17.2

To edit the table with soil properties, use appropriate buttons above the table – cut, copy, paste, delete, select default value for constructed soil, select colour of soil, define presence of groundwater.

Grids

To generate polygonal contour, define the grid in appropriate dialog box – on the MODEL menu, click **Grids** (button on the toolbar). When you start the program, the grid No.1 is already displayed in the working area of the screen. If required, it may be deleted.

In the **Grids** dialog box, do the following:

– in the **Grid No.** box, define the grid number (by default, grid No. 1 is already defined);
– define number of steps and their values along the X and Y-axes;
– if required, define grids relative to the vertical axis;
– click **Apply** .

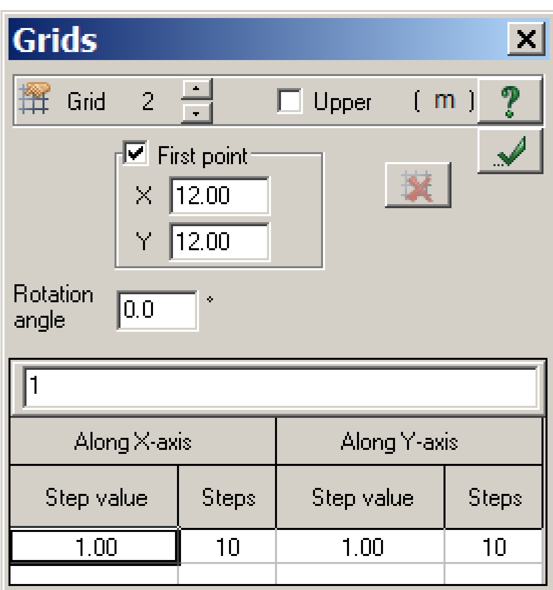


Figure 17.3

Boreholes

To define boreholes, on the MODEL menu, click **Boreholes** (button  on the toolbar). Define number for the borehole in the appropriate box and coordinates of the borehole as well as mouth elevation. In the **GE** list, select certain number of soil from the table of soil properties and in the first row of the table define (if appropriate check box is selected) either depth of layer or bottom elevation, etc. Boreholes appear on the screen as you define them.

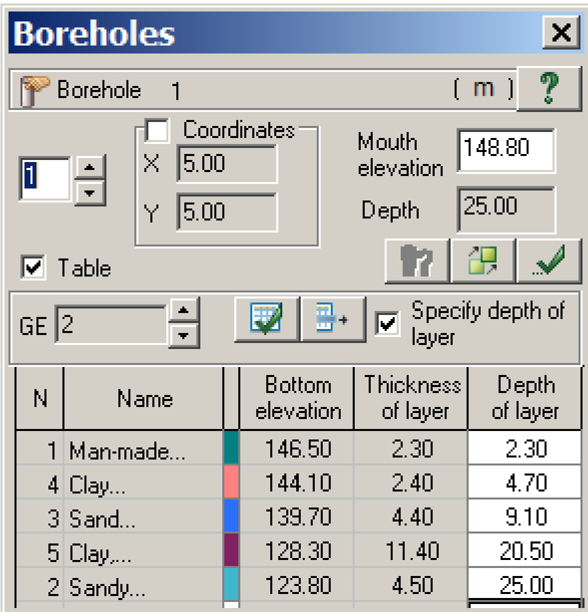


Figure 17.4

To display numbers of boreholes, on the VIEW menu, point to **Show objects** and click **Borehole numbers** (button  on the toolbar).

Loads

To define loads, on the MODEL menu, click **Loads**. Appropriate dialog box appears on the screen.
Two types of load may be defined: rectangular load or arbitrary load.

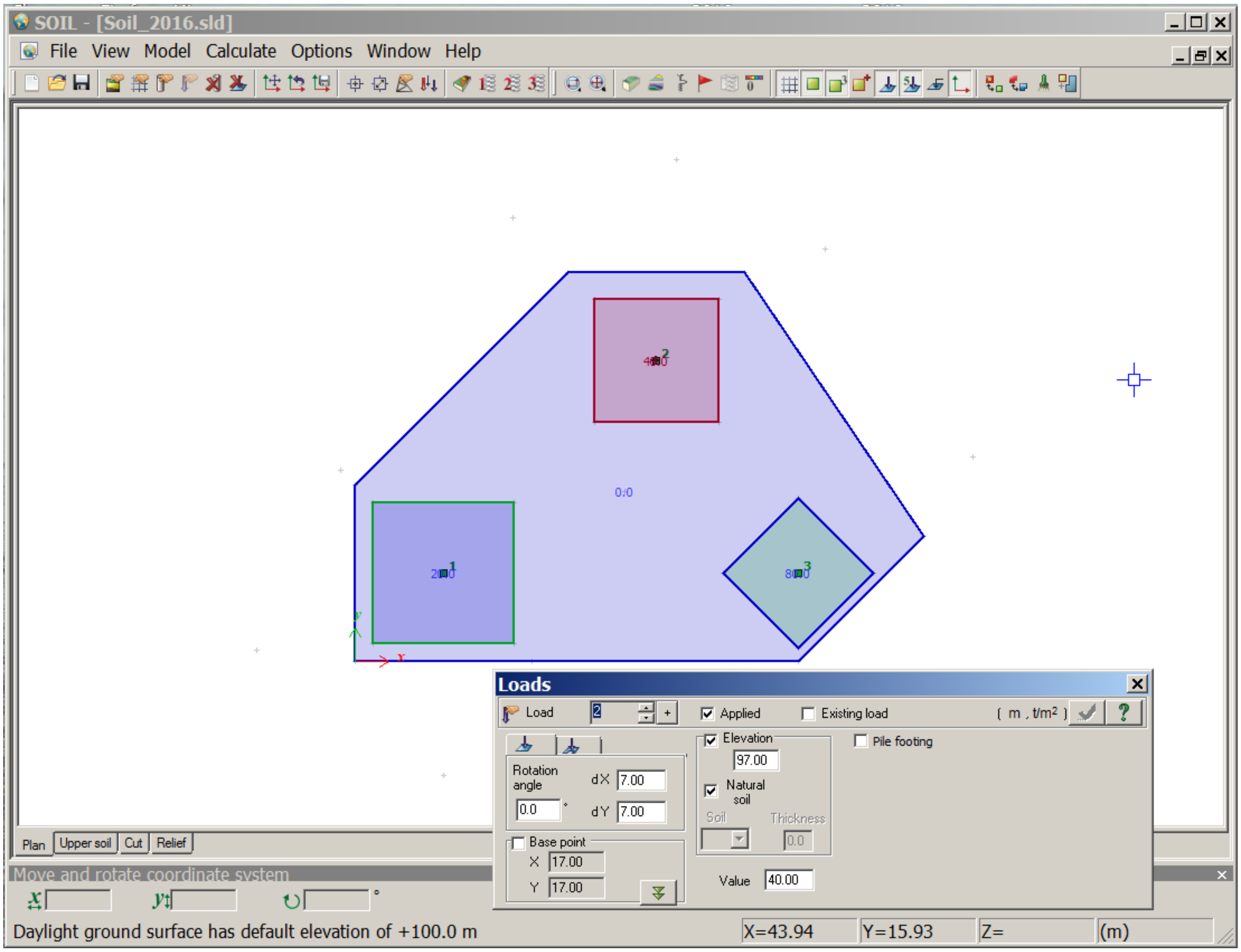
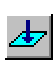


Figure 17.5

To define arbitrary contour of load, in the **Loads** dialog box, click the **Specify load on plan** button  and define (in sequence) grid nodes that generate required polygon. To close the contour, click its first node. It is possible to specify load by coordinates for vertices in the load contour.

To display the sketch in the dialog box, click the **Preview load and edit base point** button . In the extended area of the dialog box, click (if required) the base point of rectangular load as it is shown in the figure below.

To define rectangular load, click the  tab in the dialog box.

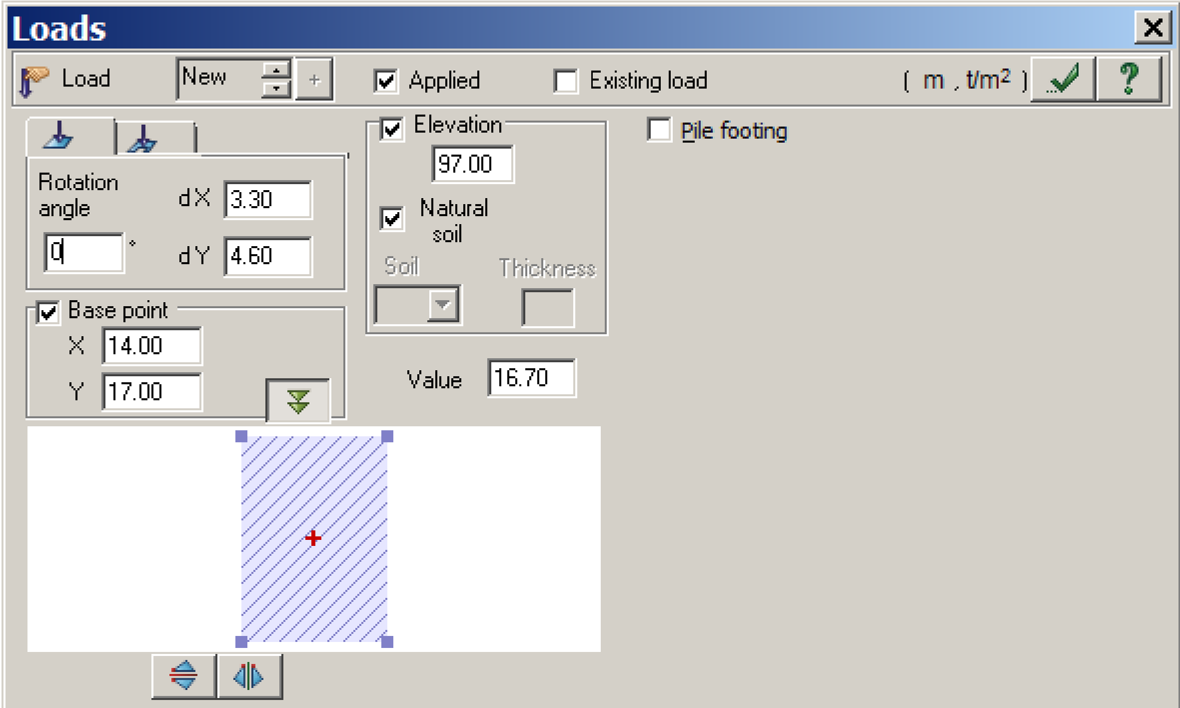


Figure 17.6

If the load is applied to the pile footing, then in the **Loads** dialog box, select appropriate check box and define required parameters.

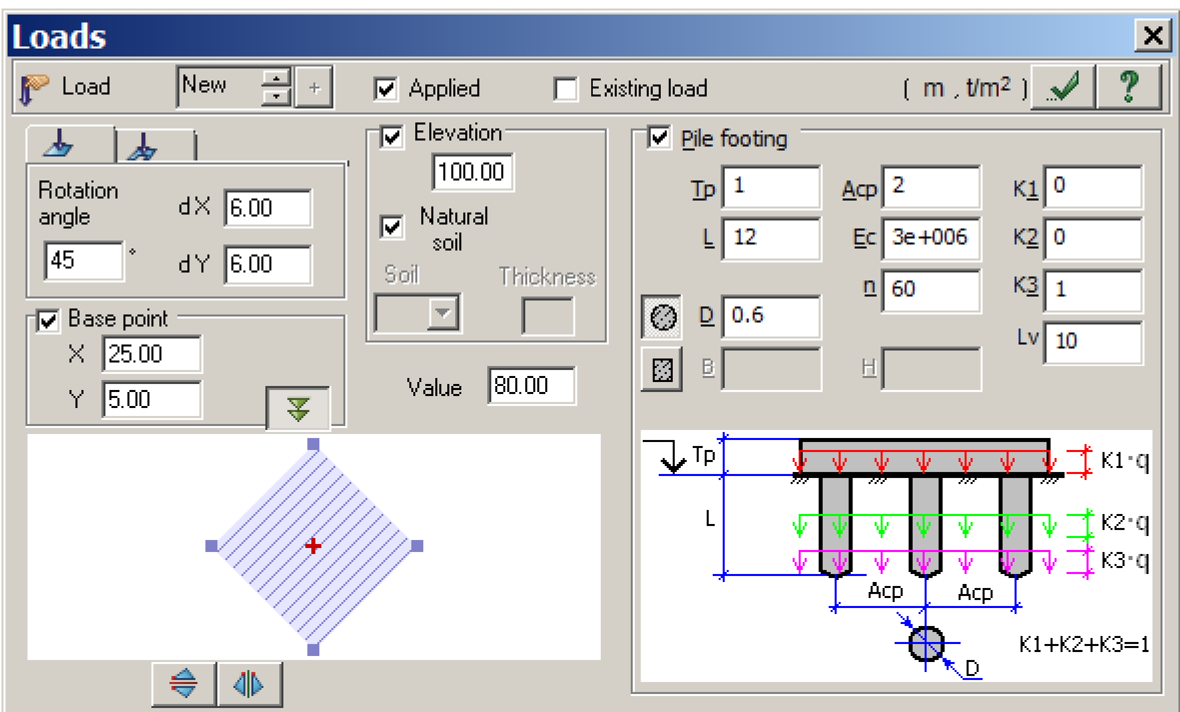


Figure 17.7

For analysis of pile footing, the following parameters should be defined:
Tp – thickness of foundation slab.
L – pile length.

D – diameter of piles with circular cross-section.
B, H – dimensions for square or rectangular cross-section of pile.
Acp – mean step of piles.
Ec – modulus of elasticity for pile material.
n – number of piles.
K1 – fraction of load transmitted to soil at base of foundation slab.
K2 – fraction of load transmitted to soil by friction along side surfaces of piles.
K3 – fraction of load transmitted to soil below the pile toe (at level L).
Lv – number of levels through which the load from friction along side surface of piles is transmitted. If K2=0, this value is not considered in analysis.

Analysis of pile footing as equivalent one in strict compliance with building code is carried out for K1 and K2 = 0 and K3 = 1. Another distribution of load is permitted, e.g. K1=0.05, K2=0.9, K3 = 0.05 (or other), that provides smooth accumulation of settlement from piles. In this case, K2 may be divided into **Lv** levels (e.g. Lv = 10). Then stress diagrams in soil will be of step-type and present elevations of application of appropriate load fractions.

In analysis according to SP 24.13330.2011, weight of soil is not considered in the volume of equivalent footing.

In analysis according to DBN V.2.1-10:2009, weight of soil is considered in the volume of equivalent footing.

Depth of compressible stratum Hc for pile footing is computed from the elevation of load application.

Visualization of soil model

When boreholes, GE, loads and grids are defined, on the MODEL menu, click **Extrapolation**. The soil model will be generated automatically. To visualize 3D model, on the VIEW menu, click **3D view**.

To present arbitrary soil profile, on the VIEW menu, click **Arbitrary soil profile** (button  on the toolbar). The soil profile will be presented in the floating box that is located (by default) in the lower part of the screen. To generate arbitrary soil profile, click the **Specify points on plan** button and then specify (in sequence) on model the beginning and the end of the soil profile line. Current soil profile and coordinates of the specified points will be displayed in the **Arbitrary soil profile** floating box.

You could also specify coordinates of the soil profile line. To do this, select appropriate check box and specify coordinates of two points in the current coordinate system.

When you right-click the sketch of the soil profile, you will see the ToolTip with the name of GE and its parameters – modulus of elasticity *E*, Poisson's ratio and soil density. When you move the pointer over the sketch in the **Arbitrary soil profile** floating box, coordinates of the pointer location in the current coordinate system as well as elevation will be displayed in the right bottom corner of the box.

When you click **Vertical mark** button , you could specify arbitrary point on the soil profile. To do this, move the vertical line of the mark with the pointer and click it. Location of the point will be specified and indicated on the image with vertical dashed line of red colour. Click the **Display parameters of virtual borehole** button to present the **Boreholes** floating box with parameters of virtual borehole, that is, borehole in the specified point of the soil profile. A new number will be assigned to this borehole.

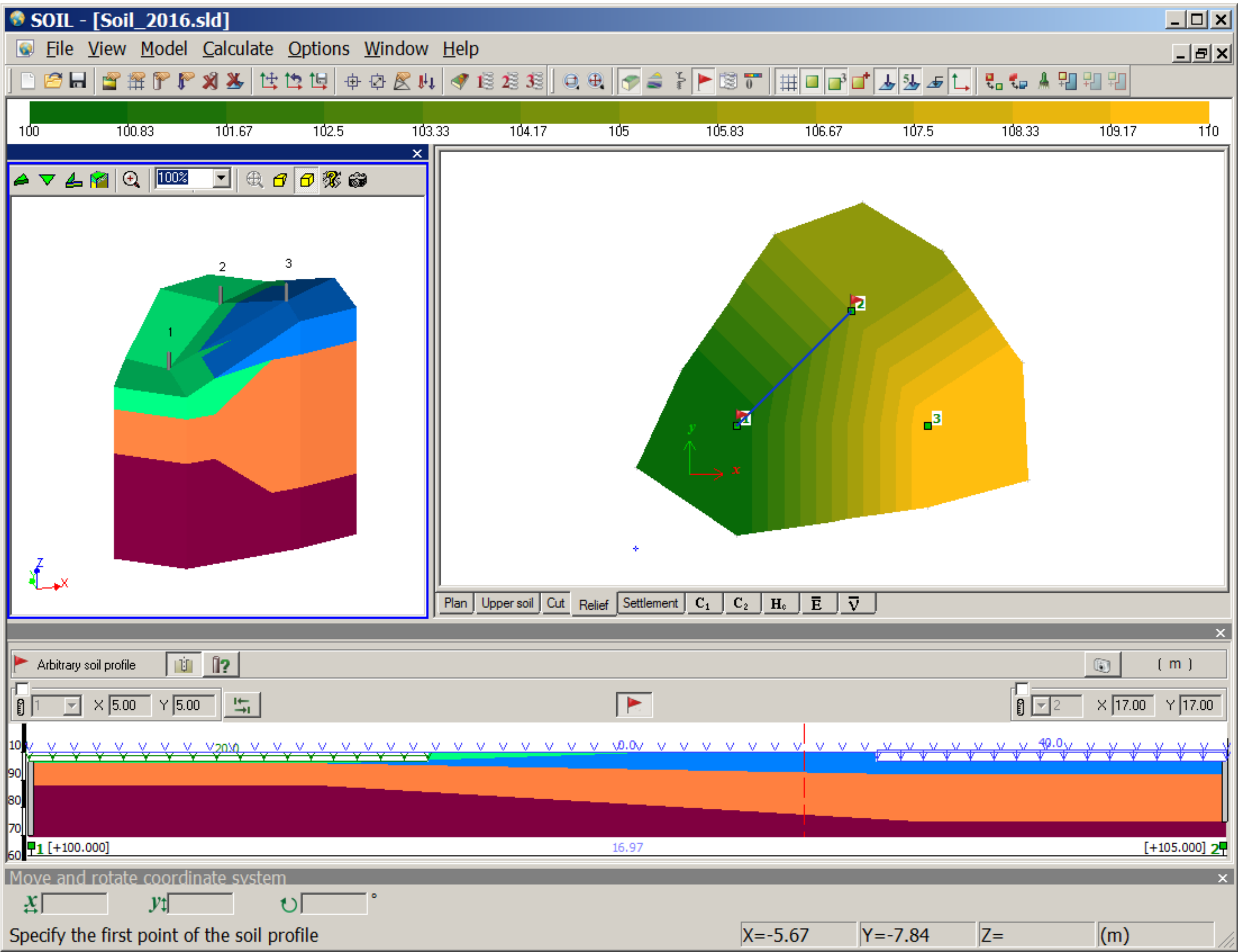


Figure 17.8

Analysis parameters

On the CALCULATE menu, click **Options**. In the **Calculation options** dialog box, select the building code for the calculation procedure.
In the appropriate boxes define the following values:
– coefficient for depth of compressible stratum (default value - 0.5);
– min depth of compressible stratum (default value - 5 m);

– additional stress invariable within compressible stratum along the whole depth (default value 0.0).

If it is necessary to consider weight of soil above elevation of load application, select appropriate check box. Difference between elevation of the soil surface and elevation of load application is the depth of foundation.

Define triangulation step of loads for generation of contour plots (by default, the step is equal to 20.). Calculation speed as well as accuracy in generation of contour plots for analysis results depend on the value of triangulation step.

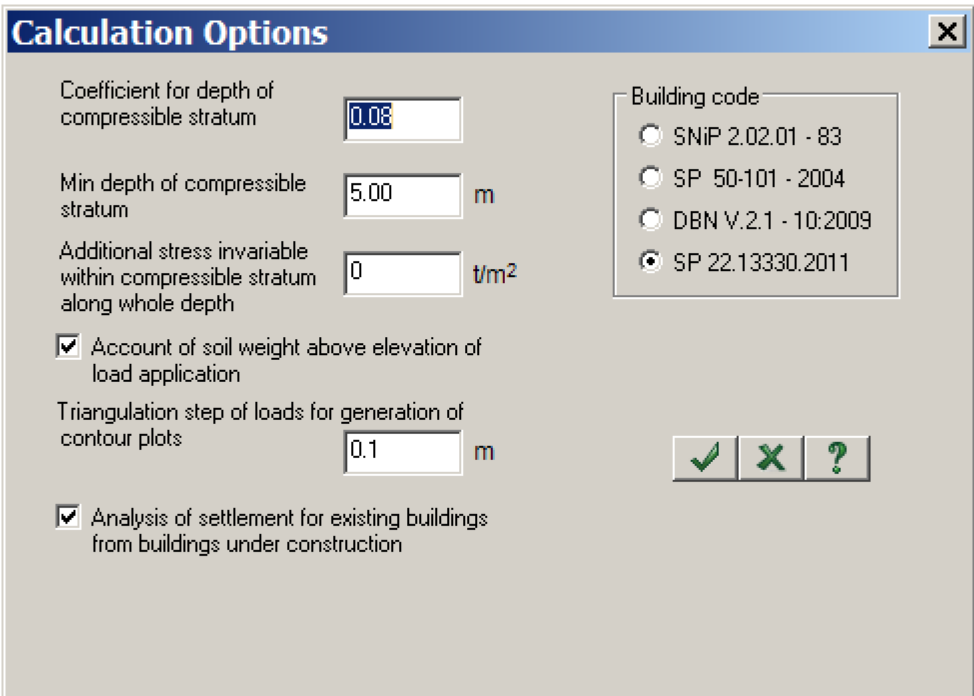


Figure 17.9

To compute subgrade moduli by one of three methods (or by all methods), on the CALCULATE menu, click **By method 1, 2, 3** (buttons 1, 2, 3 on the toolbar).

Output data

Output data contains settlement, subgrade moduli C1 and C2, depth of compressible stratum Hc (according to allowed ratio of additional vertical stress to vertical stress from dead weight of soil), mean modulus of elasticity and Poisson's ratio.
Output data may be presented as contour plots when you click appropriate buttons at the bottom of the screen.

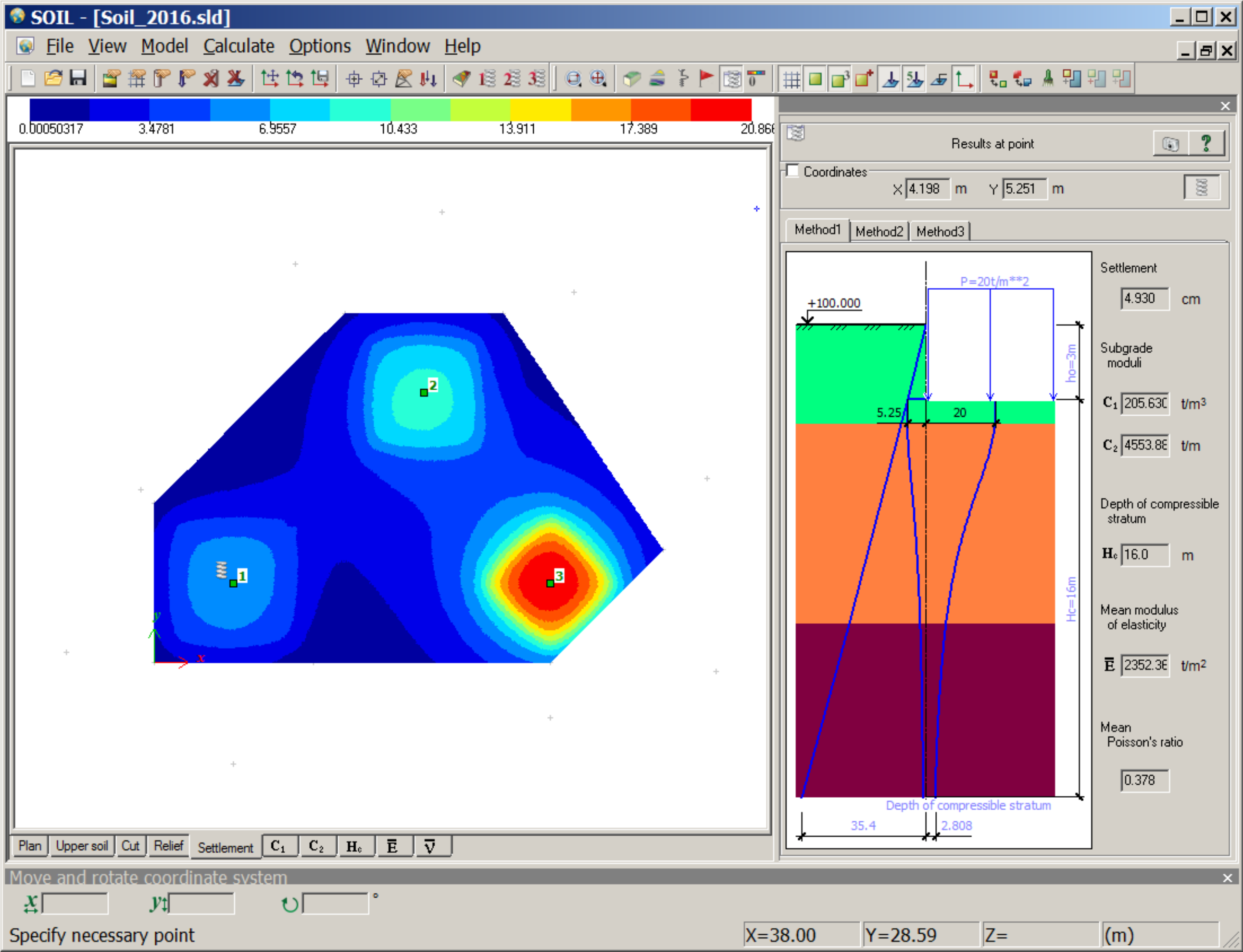


Figure 17.10

In the right part of the **Results** floating box you will see vertical elevation with applied loads and stress diagrams in soil along the vertical at the certain point on plan; values of computed parameters are displayed in appropriate boxes. Certain point on plan may be either defined with coordinates or specified on plan when you click the **Specify on plan** button.

It is also possible to display the plan with specified grids and loads (**Plan** tab) and plan view of the soil surface in colour (**Upper soil** tab). When you click the **Cut** tab, you will see the slider with depth levels. Use this slider to display colour palette of soil at the specified depth.

Click **Relief** tab to display relief of the soil surface (if boreholes with different mouth elevation are defined); in this case loads are not displayed.

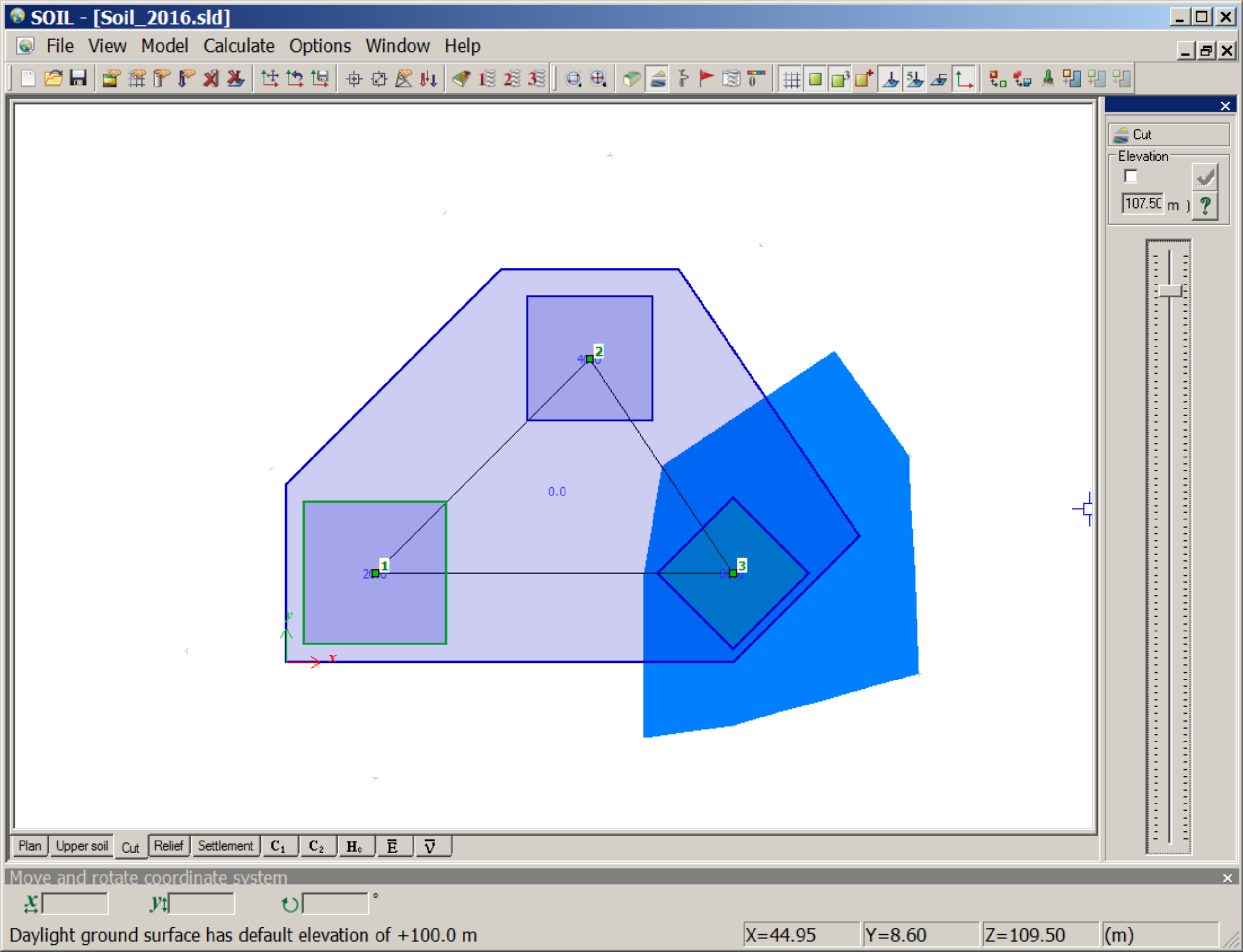


Figure 17.11 a)

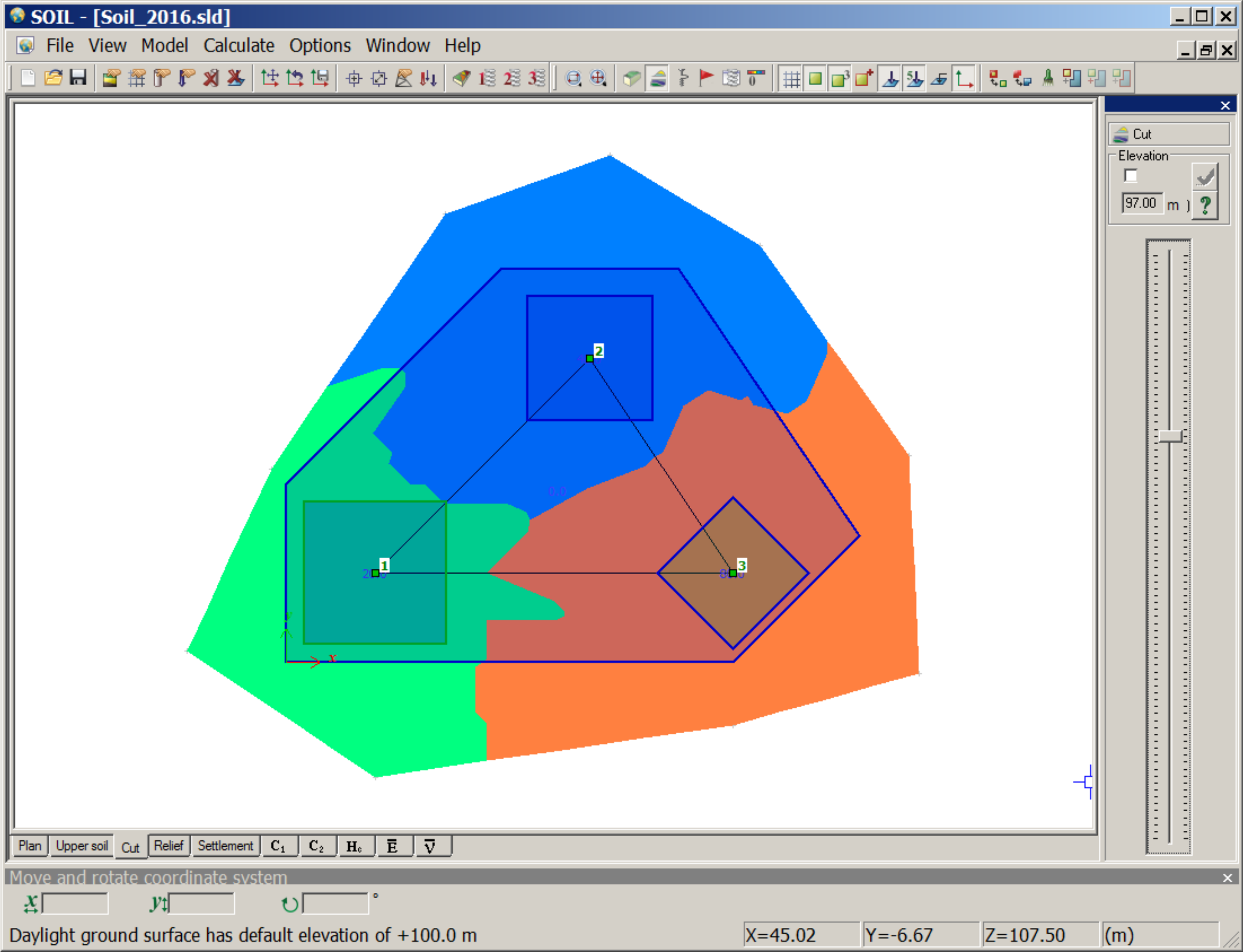


Figure 17.11 b)

Table of settlements

To generate table of settlements, on the CALCULATE menu, click **Table of settlements**. In the table define numbers or names of required points and their X, Y-coordinates. To add the specified data to the table, click **Add**. When complete data is defined, click **Apply**. The points mentioned in the table will be displayed on the screen. Click **Generate**. You will see the dialog box with the file name *NAME_Table_of_settlements.rpt*. When you click **Save**, the table will appear on the screen.

Notation in the table of settlements:
S1 – settlements in points in existing structures;
S2 – settlements in points in structures under construction;
Sr = S1 – S2 – difference of settlements.

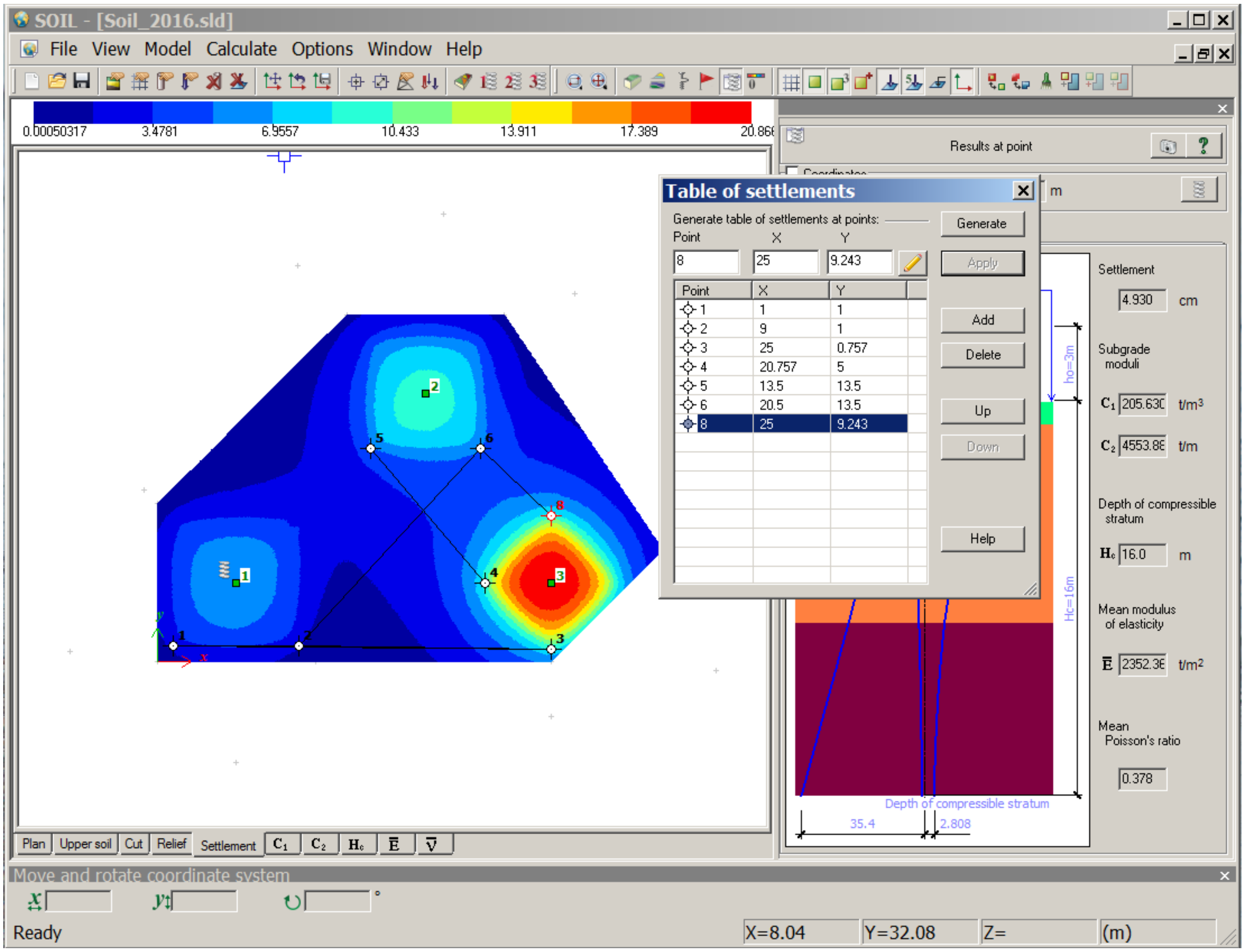


Figure 17.12

ESPRI 2016 SOIL. C:\PROGRAM FILES (X86)\LIRA SAPR\ESPRI 2016\MY_TESTS\SOIL_2016 TABLE OF SETTLEMENTS.RPT. 25 октябрь 2016

Table of settlements

Point	S1, cm	S2, cm	Sr, cm
1	1.66478	1.71682	0.0520419
2	1.325	1.46989	0.144896
3	8.12526e-006	8.47659	8.47659
4	5.17899e-005	9.46533	9.46528
5	0.0094309	4.12172	4.11229
6	0.000776228	4.60474	4.60397
8	1.0867e-005	9.35066	9.35065

Figure 17.13

Table of skews

To generate table of settlements, on the CALCULATE menu, click **Table of skews**. In the table define numbers or names for pairs of required points. To add the specified data to the table, click **Add**. When complete data is defined, click **Apply**. The points mentioned in the table will be connected on the screen with the line. Click **Generate**. You will see the dialog box with the file name *NAME_Table_of_skews.rpt*. When you click **Save**, the table will appear on the screen.

Notation in table of skews:
i, j – numbers of the first and the second points between which different of settlements will be calculated;
Sr,i – settlement at the i-th point;
Sr,j – settlement at the j-th point;
Sr,j – Sr,i – difference of these settlements;
Lj – Li – distance between points;
Skew is equal to (Sr,j – Sr,i) / (Lj – Li)•100%.

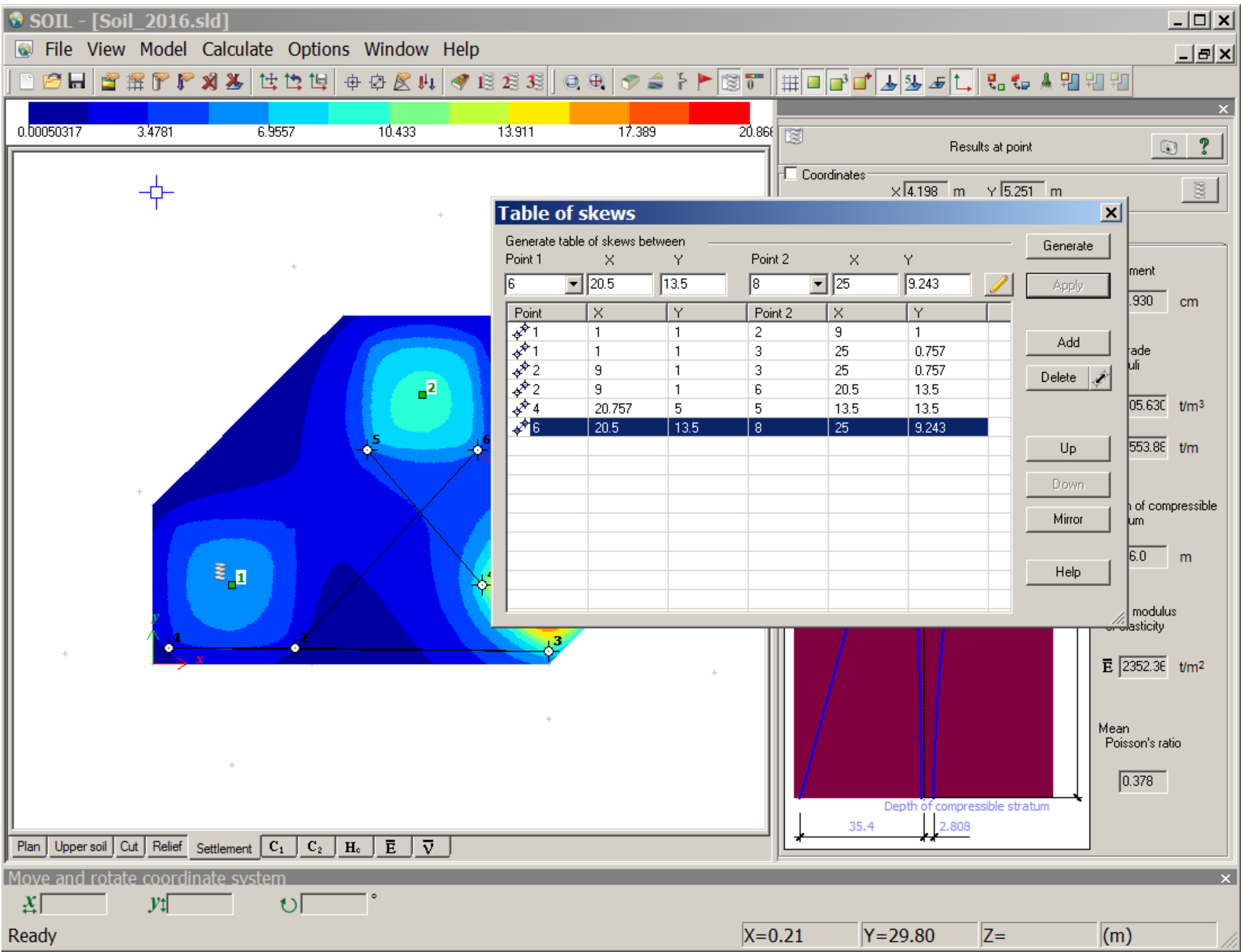


Figure 17.14

ESPRI 2016 SOIL. C:\PROGRAM FILES (X86)\LIRA SAPR\ESPRI 2016\MY_TESTS\SOIL_2016 TABLE OF SKEWS.RPT. 25 октябрь 2016

Table of skews

Point i	Sr,i, cm	Point j	Sr,j, cm	Sr,j - Sr,i, cm	Lij, cm	Skew
1	0.0520419	2	0.144896	0.0928536	800	0.0116067 %
1	0.0520419	3	8.47659	8.42454	2400.12	0.351005 %
2	0.144896	3	8.47659	8.33169	1600.18	0.520671 %
2	0.144896	6	4.60397	4.45907	1698.53	0.262526 %
4	9.46528	5	4.11229	-5.35299	1117.65	-0.47895 %
6	4.60397	8	9.35065	4.74668	619.452	0.766271 %

Figure 17.15

Calculation of subgrade moduli C1, C2 (summary)

Settlement (S) of foundation according to linear elastic half-space is calculated by layer-by-layer sum according to the formula:

$S = 0.8W$ (2)

where $W = \sum_{i=1}^{n} \frac{\sigma_{zp,i} h_i}{E_i}$; (3)

h_i, E_i – thickness and modulus of elasticity of the i-th layer of soil (with account of its division into sublayers) respectively;

n – number of soil layers from the base of foundation up to the depth $Z = H_c$ with account of division into sublayers (i=1~n).

Depth $Z = H_c$ is reached with the help of defined modulus of subgrade reaction.

The following components of settlement are calculated:

$$W_1 = \sum_{1}^{n} \frac{\left(\sigma_{zp,i} - \sigma_{zy,i}\right) h_i}{E_i};$$

$$W_2 = \sum_{1}^{n} \frac{\sigma_{zy,i} h_i}{E_{ei}};$$

$$W_3 = \sum_{1}^{n} \frac{\sigma_{zp,i} h_i}{E_{ei}}$$

where

where

E_i – modulus of elasticity of the i-th layer of soil along the loading path;

E_{ei} – modulus of elasticity of the i-th layer of soil along the unloading path;

by default $E_{ei} = E_i$;

$\sigma_{zp,i}$ – stress at the i-th layer of soil from external load;

$\sigma_{zy,i}$ – stress at the i-th layer of soil from dead weight.

If dead weight of soil at the level of foundation base is greater than mean pressure under the foundation base, then $W = W_3$, otherwise $W = W_1 + W_2$.

Mean (within the limits of fixed depth of compressible stratum Hc) values of modulus of elasticity E_{gr} and Poisson's ratio m_{gr} are used for calculation of subgrade moduli. These values are calculated according to formulas:

$$E_{gr} = \frac{\sum_{i=1}^{n} \sigma_{zp,i} h_i}{H_c}; \quad m_{gr} = \frac{\sum_{i=1}^{n} \nu_i h_i}{H_c}$$

Modulus of subgrade reaction C1 is calculated by three methods.

Method 1. Subgrade modulus C1 is calculated according to formula:

$$C_1 = \frac{E_{gr}}{H_c\left(1 - 2m_{gr}^2\right)}$$

Method 2. Subgrade modulus C1 is calculated according to Winkler method:

$$C_1 = \frac{q}{S},$$

where
q – mean pressure under the base of foundation;
S – settlement of soil.

Method 3. Just as in method 1, formula (6) is used to determine subgrade modulus C1. The difference is that in this case correction factor u to modulus of elasticity of the i-th sublayer is introduced for calculation of mean modulus of elasticity. This factor varies from $u_1 = 1$ at the level of foundation base up to $u_n = 12$ at the level of calculated depth of compressible stratum. It is accepted that factor u varies according to parabola law:

$$u = \frac{11z^2}{H_C^2} + 1 \quad .$$

Moreover, it is also accepted that additional vertical stress is distributed uniformly along the depth. Then

$$E_{gr3} = \frac{H_C}{\sum_{i=1}^{n} \frac{h_i}{u_i E_i}}$$

Method 3 is experimental and suggested in order to more precisely simulate changes in soil strength along the depth. In method 1, it is impossible to take into account that modulus of elasticity increases along the depth. It causes too high values of settlements and therefore too low values of subgrade modulus C1.

For methods 1 and 3, subgrade modulus C2 is calculated according to the formula:

$$C_2 = \frac{C_1 H_C^2 \left(1 - 2m_{gr}^2\right)}{6\left(1 + m_{gr}\right)}$$

For method 2, subgrade modulus C2 is not calculated.

Bibliography

1. Designer's manual. Design-theoretical. In 2 books. M. Stroyizdat, 1972.
2. Warren C. Young. Roark's Formulas for Stress and Strain. Sixth Edition. McGraw-Hill Book Company, 1989. P. 348-355.
3. Manual on soil mechanics and dynamics of soil. V.Shvets, L.Ginzburg, etc. – K.: Budivelnik, 1987. – 132 p.
4. SNIP 2.02.01-83. Foundation soils of buildings and structures. – M. Stroyizdat, 1985. – 40 p.
5. A.Shapoval. Algorithm optimization for analysis of slope stability. Master's research paper. PSACEA – 2004.
6. V.Alexandrov. Methods for determining dangerous combination of loads and their application to software for computer-aided design of building structures: Abstract of a thesis. …PhD in Technical Sciences: 05.01.23. – L., 1973. – 30p.
7. E.Strelets-Streletsky. Methods for determining dangerous combination of stresses during evaluation of strength in elements of structure: Abstract of a thesis. …PhD in Technical Sciences: 01.02.03. – M., 1987. – 22p.
8. Manual on design of residential buildings / TsNIIEPzhilischa Goskomarhitectura. Issue 3. Structures of residential buildings (to SNIP 2.08.01-85).– M. Stroyizdat, 1989. – 304p.
9. Reinforced concrete walls in earthquake-proof buildings: Study and basis of design: Published by USSR-Greece / Edited by G.Ashkinadze, M.Sokolova. – M. Stroyizdat, 1988. – 504p.
10. SNIP 2.03.01-84. Concrete and reinforced concrete structures. – M.1985. – 79p.
11. Building code CEB-FIP (Euro-International Concrete Committee – International Federation for Prestressing) : For building codes for reinforced concrete (volume 2). – M.: NIIZHB Gosstroy USSR, 1984. – 284p.
12. A.Zolotkov. Earthquake resistance of monolithic buildings. – Kishinev, Moldova, 2000. – 283p.
13. Y.Izmajlov. Earthquake-resistant monolithic buildings. Kishinev, Moldova, 1989. – 253p.
15. Y.Izmajlov. Analysis of walls in frameless buildings in destruction along inclined section.- 'Building mechanics and analysis of structures', No.4, 1990 p.63-70.
16. Barda F.,Hanson J.M.,Corley W.G. Sher Strength of Low-Rise Walls with Boundary elements. Reinforsed Concrete Structures in Seismic Zones. Publication SP-53 American Concrete Institute, Detroit, 1977.
17. Hernandez O.B., Zermeno M.E. Strength and Behaviour of Structural Walls with Shear Failure.Proc.7 WCCE, Istambul, 1980.
18. Tassios T.P., Lefas J., Lulurgas S. Response Degradation and Hysteretic Damping of Reinforced Concrete Linear Elements and Shear Walls Under Large Cyclic Post Vielding Deformations, NTU Report, Atens, 1983.
19. V.Maximenko, Y.Voskresenskaya, N.Marjenkov. Engineering methods for evaluation of ultimate state in stiffness diaphragm in earthquake loads. 'Budivelni konstruktsii'. – Kiev: NDIBK, Issue 69, 2008, p.637–645.
20. SNIP 2.01.07-85*. Loads and actions.
21. SNIP 2.05.03-84*. Bridges and pipes.
22. SNIP II-23-81*. Steel structures.

23. SNIP II-7-81*. Construction in seismic areas.
24. SNIP 2.03.01-84*. Concrete and reinforced concrete structures.
25. SNIP 52-01-2003. Concrete and reinforced concrete structures.
26. SP 52-101-2003. Concrete and reinforced concrete structures.
27. TSN 102-00*. Reinforced concrete structures with rebars of classes A500C and A400C.
28. SNIP II-22-81*. Rock / reinforced masonry structures.
29. SNIP 2.03.01-84*. Concrete and reinforced concrete structures.
30. SNIP II-25-80. Timber structures.
31. SNIP 2.02.01-83*. Foundation soils of buildings and structures.
32. SP 50-101-2004. Foundations of buildings and structures.
33. SNIP 2.02.03-85. Pile foundations.
34. SP 50-102-2003. Design and construction of pile foundations.
35. MGSN 2.07-01. Footings, foundations and substructures.
36. SNIP II-3-79*. Construction heat engineering.
37. DSTU 3760-07. Reinforcing bars for reinforced concrete structures.
38. DSTU 3760:2006. Reinforcing bars for reinforced concrete structures.
39. Recommendations on use of reinforcing bars by DSTU 3760-07 in design and manufacturing of RC structures without prestress of rebars. – K.: Gospotrebstandart of Ukraine, 2007.
40. Eurocode 2. Design of reinforced concrete structures.
41. Eurocode 3. Design of steel structures.
42. Manual on design of steel structures (to SNIP II-23-81*).
43. Manual on design of concrete and reinforced concrete structures from heavyweight and lightweight concrete without prestress of reinforcement (to SNIP 2.03.01-84).
44. Manual on design of pile foundations. – M. Stroyizdat, 1980.
45. P.Troitsky, I.Levitansky. Study on actual work of welded frame joint and recommendations on its analysis.//Materials on steel structures, issue 19, 1997.
46. Steel structures in 3 volumes. (Designer's manual)/Edited by V.Kuznetsov (TSNIIPSK named after N.Melnikov). — M.: ACB, 1998.
47. DBN 2.1.2-2:2006. Loads and actions.
48. DBN V.2.1-10:2009. Footings and foundations of structures.
49. DBN V.1.1-12:2006. Construction in seismic areas.
50. DBN V.2.3-14:2006. Transport facilities. Bridges and pipes. Design rules.
51. SP 16.13330.2011. Steel structures.
52. SP 22.13330.2011. Foundation soils of buildings and structures.
53. SP 24.13330.2011. Pile foundations.
54. SP 63.13330.2012. Concrete and reinforced concrete structures. General rules.
55. DBN V.2.6-162:2010. Rock / reinforced masonry structures.
56. SP 64.13330.2011. Timber structures.
57. SP 164. 1325800.2014. Strengthening of reinforced concrete structures with composite materials.
58. Eurocode 5. Design of timber structures.

Index

A

Advanced calculator ESPRI 16
Analysis of inelastic deflections 276
Analysis of RC support sections (posts) 375
Analysis of steel elements 105
Analysis parameters 387
Anchorage of reinforcement by DSTU 3760-07 135
Arbitrary plane frame 49
Areas and volumes 4

B

Bearing capacity of piles by field test results 243
Bibliography 399
Brick pier 176
Brick pier by DBN V.2.6-162 2010 186

C

Cable and string 90
Cable and string. Input data 94
Cable and string. Notation 96
Cable and string. Output data 94
Calculation of subgrade moduli C1, C2 (summary) 395
Circular arc 350
Climatic thermal loads 263
Cold-formed shapes 125
Column strengthened with composed materials 166
Combined piled-raft foundation 244
Composite steel and concrete columns 161
Composite steel and concrete slabs ... 168
Concrete pipe sections 158
Concrete sections with fibre-reinforced polymer (FRP) bars 154
Continuous beam 38

D

Dead weight of multi-layer coating 252
Deflections 275
Design compression strength in masonry 175
Diagram multiplication 6
Diaphragm 297

E

Effective lengths of steel structure elements 117
Ellipsoid 281
Ellipsoid. Bearing capacity of RC elements 281

F

Foundations and beddings 213
Function interpolation 14

G

General notes 332

H

Hazardous energy combinations of forces (EnergyCF) 266

I

Ice loads 261
Influence lines in continuous beam 97
Input data 380
Input data (Toster) 333

L

Linear algebra 8
Load factors 250
Loads and actions 249

M

Masonry and masonry reinforcing 175
Masonry in local compression 180
Mathematics for engineer 3
Mode shapes and frequencies of natural vibrations in cantilever 82
Mode shapes and frequencies of natural vibrations in continuous beam 87
Moduli of subgrade reaction C1, C2 214
Moment of inertia in torsion 33

O

Options to edit and visualize the model 58
Options to edit the model 349
Options to visualize the model 349
Output data 59, 353, 388

P

Parametric joints of steel structures 120
Parametric plane frame 45
Parametric sections 22
Parametric thin-walled sections 23
Pile under combined action of loads ... 228
Polynomial roots 12
Prestressing 369

Principal and equivalent stresses in reinforced concrete structures 146
Principal and equivalent stresses in soil 232
Principal and equivalent stresses in steel structures 113
Properties of concrete 133
Punching shear 309
Punching shear analysis (Belarus) 326
Punching shear analysis (Eurocode) .. 319
Punching shear for arbitrary contour .. 310
Punching shear for rectangular contour 314
R
Rectangular slab 64
Rectangular slab on elastic foundation. 60
Reinforced concrete (RC) structures .. 131
Reinforced concrete shell 140
Reinforced concrete slab 144
Reinforced concrete wall-beam 142
Resonance check for wind turbulence 272
S
Sections 21
Sections of composite timber 209
Sections of glued timber 208
Sections of RC elements 137
Sections of solid timber 206
Separate sections as single unit 26
Settlement of equivalent footing 230
Sheet piling 287
Sheet piling (module) 289
Shell on circular plan 77
Shell on rectangular plan 72
Sign convention 367
Sign conventions for displacements and forces 60
Single pile 224
Slope stability 235
Snow loads 255
Soil 379
Soil (Chapter) 379
Stability factors and buckling modes in cantilever 84
Stability of multi-layer slope 240
Static and dynamic analyses, stability ... 37
Steel structures 101
Steel table 102
Step 1. Generate model geometry 51
Step 2. Define boundary conditions (restraints) and hinges 52
Step 3. Define cross-sections for bars .. 53
Step 4. Define loads on design model .. 54
Strength of RC diaphragm in earthquake loads 297
Strength of reinforced concrete butt joint in shear 152
Strengthening with composite materials 148
T
Table of reinforcement 134
Timber structures 205
Toster (Chapter) 331
Truss 41
V
Visualization of soil model 386
W
Wall-beam 68
Wind loads 258