# TABLE OF CONTENTS

# *1. PROGRAM INSTALLATION*

## 1.1. Software/Hardware Requirements

For the ***CBS Pro*** program to be installed, the following requirements have to be fulfilled by the computer:

- Windows 2000 (+SP 4) / XP Pro (+SP 2) / XP 64 bit / Vista operating system
- complete IBM PC compatible computer (with at least Pentium processor)
- 1024x768 monitor resolution or higher.

*NOTE:* *For 3D graphical presentation the* **CBS Pro** *program employs DirectX technology, which is accessible in all operating systems, except for Windows NT system. Considering this, in the above mentioned system, 3D graphical presentation is not complete (surface rendering is not provided); complete graphical presentation is available once the Open GL option is chosen.*

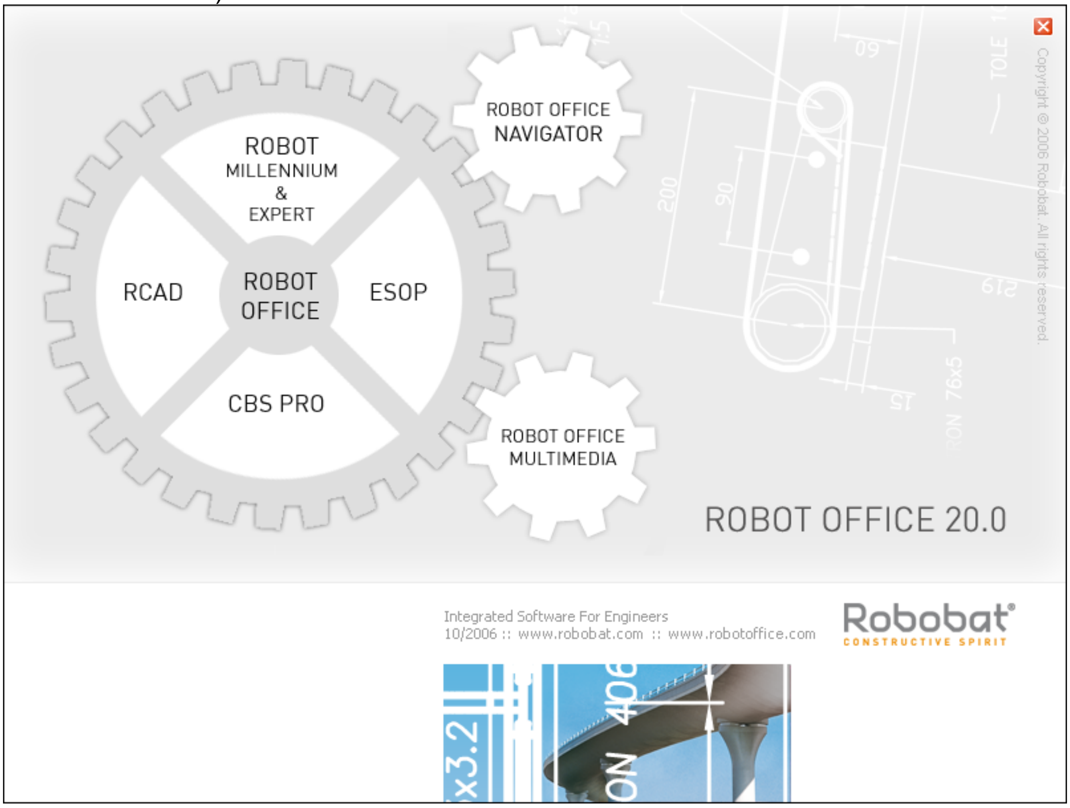
The requirements imposed by DirectX technology include:

- one of the following operating systems is required: Windows 2000 or Windows XP
- it does not work correctly in the following operating systems: Windows 95 or Windows NT 4.0.

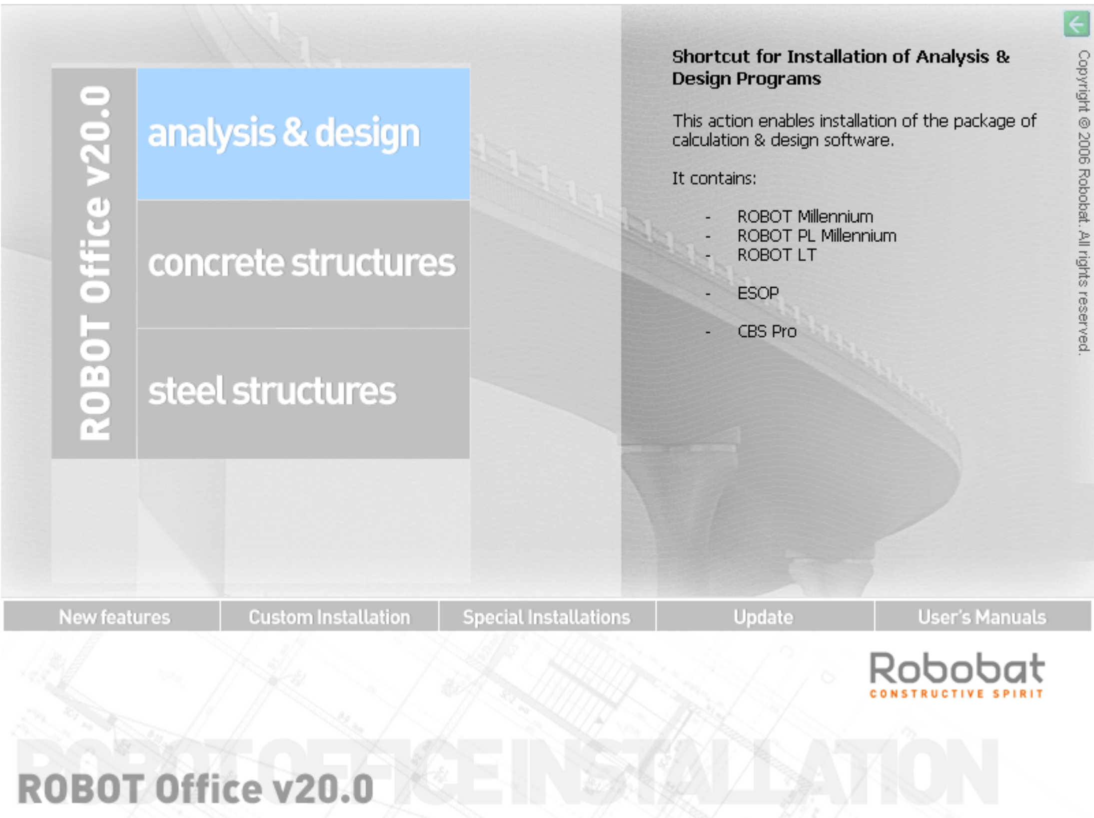

## 1.2. Installation Program

The ***CBS Pro*** program is equipped with a program for automatic installation. It enables quick and successful program installation on a user's computer. During program installation a few simple questions will appear on the screen and they will have to be answered by the user.

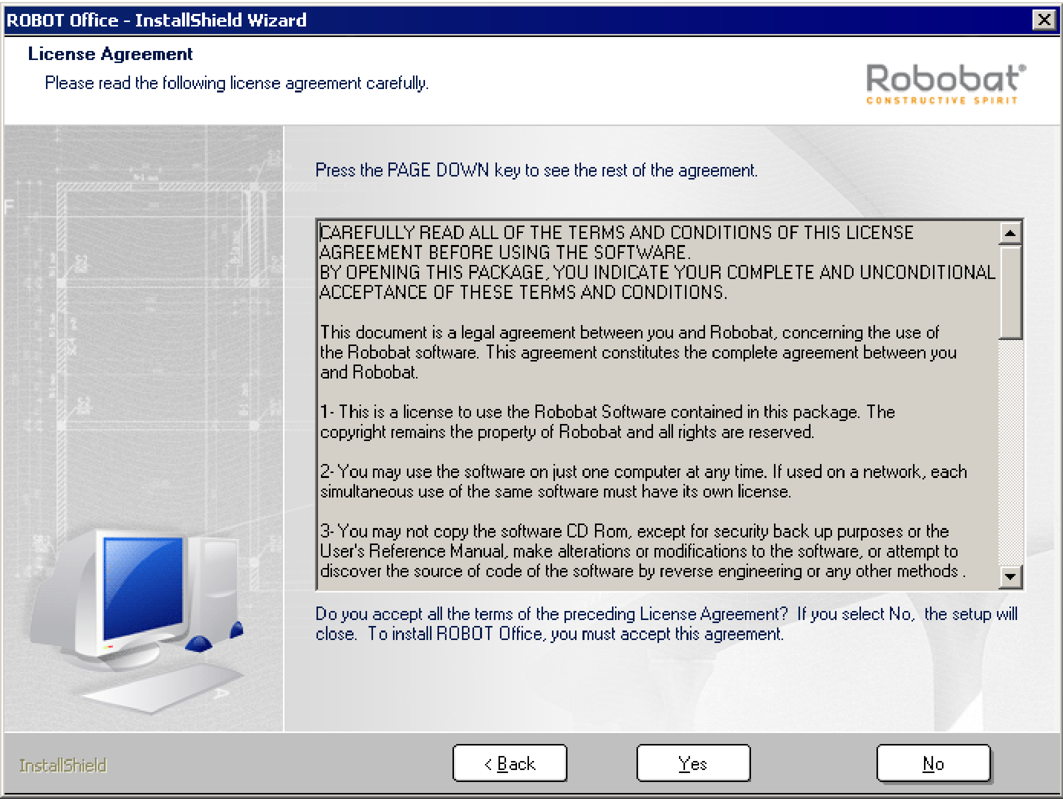
To begin with, the compact disk with the ***CBS Pro*** program installation should be inserted into the appropriate drive. The screen will show the start installation window of the system (in the case of a DVD drive).



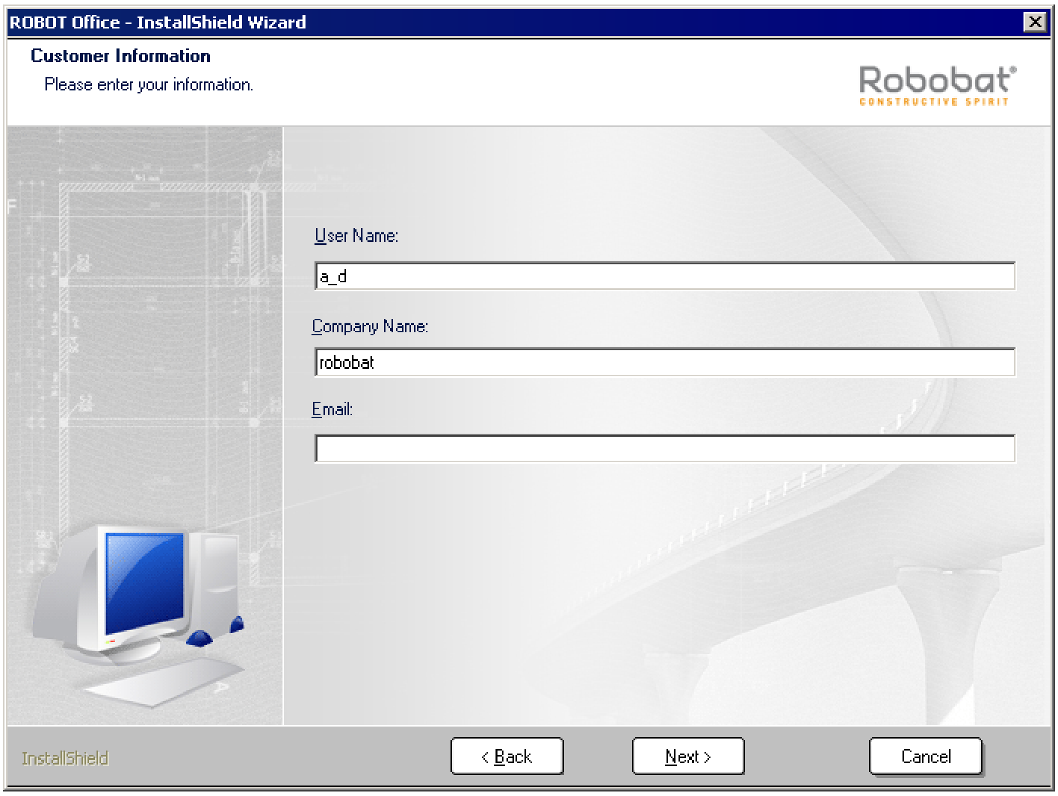

After selecting the *ROBOT Office* option, the window for selection of an installation type appears on the screen.



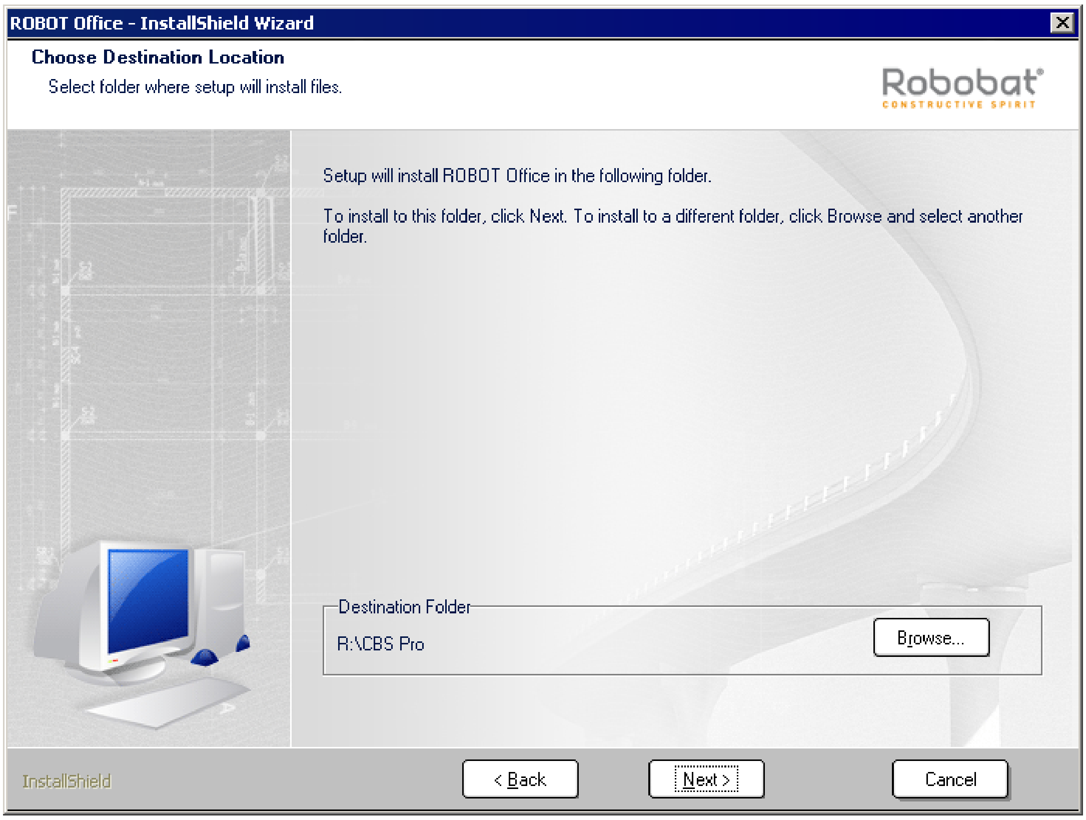
Presently, the ***CBS Pro*** program is installed as one of the components of the *ROBOT Office* package. Launching the installation of *ROBOT Office* in the *Custom Installation* mode (by pressing the **Custom Installation** button = see the drawing above) allows installation of required *ROBOT Office* components.
Below are presented successive stages of installation of the ***CBS Pro*** program (custom installation - after pressing the **Custom Installation** button).
First, the dialog box with the LICENSE AGREEMENT of the ***CBS Pro*** program appears; to continue the installation, the user, once familiarized with the terms of the agreement, should press the **YES** button (acceptance of the license agreement terms). The next dialog box shows the remarks related to work of the ***CBS Pro*** program; the installation will continue on pressing the **NEXT** button.




Then, the dialog box is displayed on the screen, wherein basic user information should be specified (see the drawing below); the following data should be given there: the name (initials) of the user, the company name, and optionally the e-mail address; press the **NEXT** button to continue.

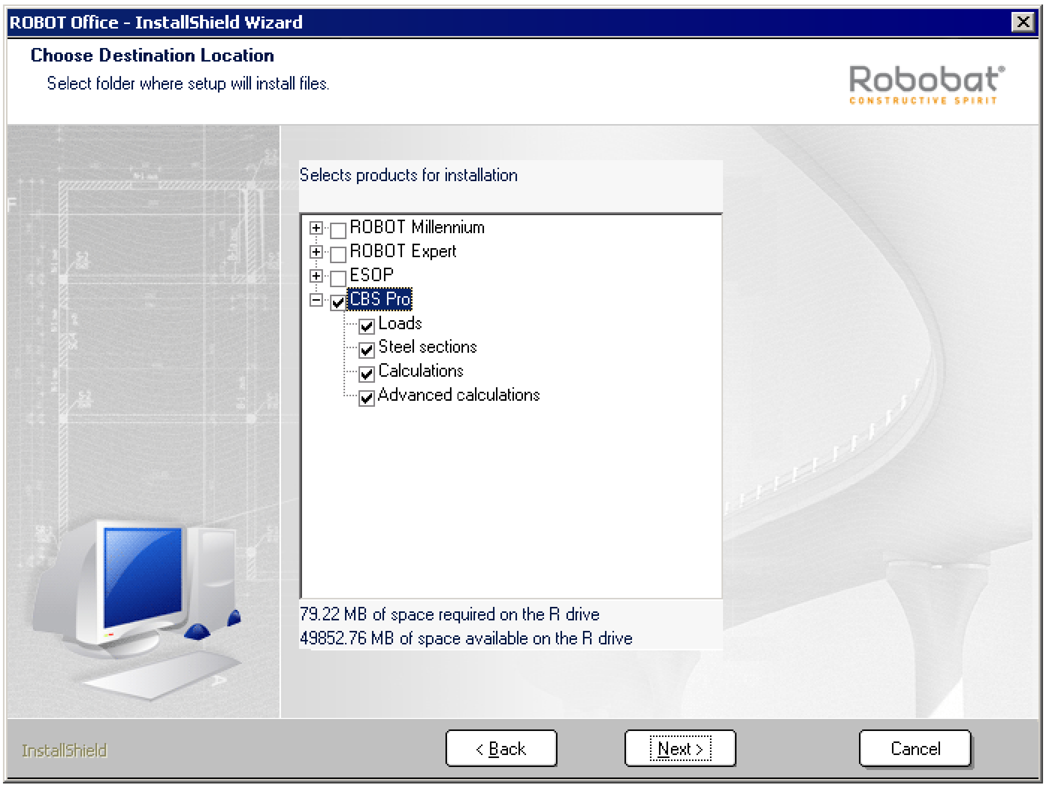



Afterwards, there appears the dialog box where the user may indicate a destination folder in which the program will be installed on disk (see the drawing below). The location may be entered directly from the keyboard or selected using the **BROWSE** button provided in the bottom part of the dialog box; if the indicated folder is not on disk, the installation program will create a folder with a name specified by the user; to continue, press the **NEXT** button.



Next, there appears the dialog box for selection of the products of the Robobat company to be installed on disk (see the drawing below); once the ***CBS Pro*** program only is chosen, it is possible to select components of the installation; the following components are accessible in the current program version:

- definition of loads (the *Loads* component that enables definition of loads in a structure)
- calculations – a component that enables analysis of a structure with the use of simplified methods (the trapezoidal and triangular method or the simplified FEM)
- advanced calculations - a component that enables structure analysis by means of FEM for the whole structure)
- steel sections.




Once the **NEXT** button is pressed, the screen shows the dialog box for determining a name of the group in which the ***CBS Pro*** program will be installed; a default name of the group, which can be changed by the user, is displayed on the screen.
Finally, the dialog box confirming the information entered hitherto (program location, name) is shown on the screen; pressing the **NEXT** button starts program installation.
Once the ***CBS Pro*** program is installed on disk, it is necessary to restart the WINDOWS 95/98/NT/2000 system. Therefore, a warning is displayed on the screen informing that the computer should be restarted before the program activation; moreover, one of the options listed below may be chosen in the dialog box (but only in the case the program is installed after copying the installation version from the Internet or directly from the installation CD once only the ***CBS Pro*** program installation is chosen):

- connect with the Internet and check for updates of the installed version on the software producer page
- complete installation without checking for updates of the installed program.

After installing the program, the ***CBS Pro*** program menu includes the *Help / Check for updates* option, which when run, opens the internet page of the program producer or dealer;

from this page it will be possible to download the program updates (Maintenance Pack, information about a new version, etc.).
The ***CBS Pro*** system can be run by:

- double-clicking on the system icon located on the Windows system desktop
- selecting the ***CBS Pro*** command in the group created in the START menu.

# 2. PROGRAM DESCRIPTION

## 2.1. General Program Description

The ***CBS Pro*** program is intended for initial generation of a structure model and estimation of structure costs. The current version of the program (due to sections available in the program) is applied mainly in the case of RC and timber structures. The program may be used as:

- tool for preparing tender bids (quick modeling, cost assessment)
- easy modeler generating automatically a calculation model in the ***ROBOT Millennium*** program
- program for approximated calculation of a structure and design of the structure's RC elements
- program enabling import of ready-to-use models from architectural programs.

Basic functions of the ***CBS Pro*** program include:

a) structure definition with automatic generation of a calculation model in the ***ROBOT*** program
   architectural support - export / import - DXF 2D (with the possibility to restore a structure on the basis of layers), export / import - IFC 3D, architectural presentation of the loaded structure
   definition of structure loads
   approximated calculation of a structure and design of the structure's RC elements

b) access to data needed for cost estimation (the link with the program ***ESOP Cost Estimates***).

## 2.2. List of Shortcuts

| **In order to:** | **Press:** |
|---|---|
| select all | **Ctrl + A** |
| copy a text or a drawing | **Ctrl + C** |
| open a new project | **Ctrl + N** |
| open an existing project | **Ctrl + O** |
| start printing | **Ctrl + P** |
| save the project | **Ctrl + S** |
| cut a text or a drawing | **Ctrl + X** |
| repeat the operation | **Ctrl + Y** |
| paste a text or a drawing | **Ctrl + V** |
| paste vertically | **Ctrl + W** |
| undo the operation | **Ctrl + Z** |
| delete selected elements | **Num Del** |
| display a structure axonometric view (3D XYZ) | **Ctrl + Alt + 0** |
| project a structure on the XZ plane | **Ctrl + Alt + 1** |
| project a structure on the XY plane | **Ctrl + Alt + 2** |
| project a structure on the YZ plane | **Ctrl + Alt + 3** |
| zoom in the structure view on the screen | **Ctrl + Alt + A** |
| display the initial view of a structure | **Ctrl + D** |
| zoom in the structure view using the window | **Ctrl + Q** |
| rotate the structure (selection mode) in the 3D view | **Shift** + **RMC** (right mouse button click) |
| | **Alt (right)** + **LMC** (left mouse button click) |

| | |
|---|---|
| copy contents of the current window to the clipboard | **Ctrl + Alt + Q** |
| open the ***Properties*** dialog box | **Alt + Enter** |
| zoom out the structure view on the screen | **Ctrl + Alt + R** |
| display the structure architectural view | **F10** |
| display the structure structural view | **F9** |
| display the structure computational view | **F8** |
| display a structure model with rendering | **Ctrl + F10** |
| display the structure skeleton model | **Ctrl + F9** |
| open the ***Display*** dialog box | **Space** |
| open the ***Reports*** dialog box | **F7** |
| start export of a structure to the ***ROBOT*** program | **Ctrl + F7** |

## 2.3. Stages of Work in the CBS Pro program

***CBS Pro*** is a program which allows defining a structure model and performing structure calculations (with the use of simplified methods or the advanced method) along with the possibility to design RC structure elements.
Creation of the structure model, definition of loads and structure calculations / design may be split into the following stages:

*1. model definition / model loading from other programs*

definition consists in creating a building model, i.e. determining locations of structural elements of the building, such as: beams, columns, walls, slabs, foundations, etc. The basic options used for this purpose include:
Loading a structure model from other programs
Structural axis grid
Story
Objects available in the program (beams, columns, walls, slabs, etc.)
Material database
Section database

*2. definition of loads*

definition consists in specifying locations and values of forces applied to individual elements of the structure model; the method of load definition has been presented in the following topics:
Rules of load definition
Types of loads
Definition of loads
Default loads
Load combinations

*3. structure calculations*

once loads are applied, structure calculations may be performed; selection of the method of calculation and result presentation have been presented in the following topics:
Calculation options
Calculation results

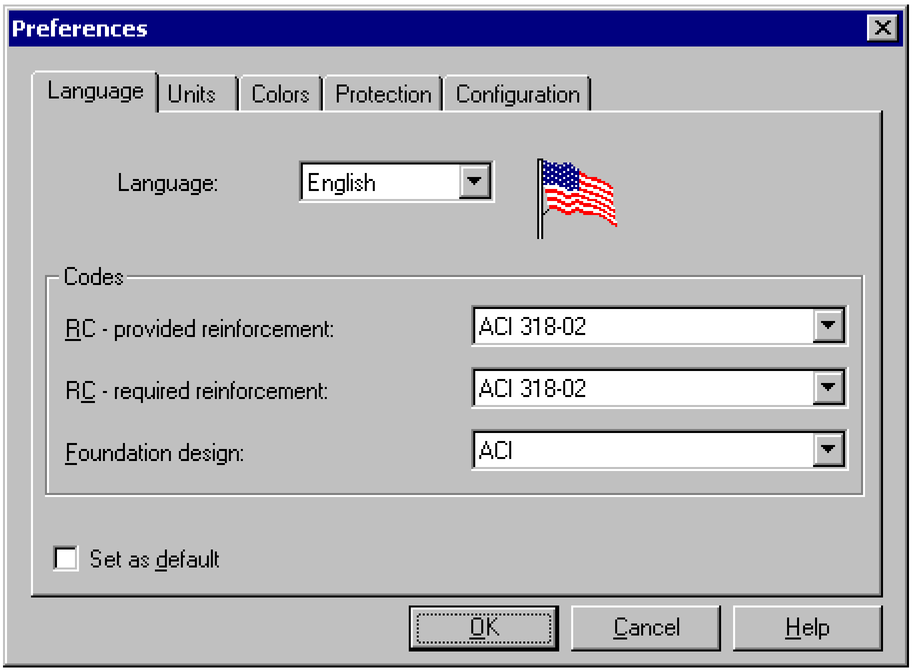
*4. design of RC structure elements*

once structure calculations are over, it is possible to design RC elements of a structure; the design method has been presented in the following topics:
RC element design – calculation options
RC element design – general parameters
RC element design – estimated calculations
RC element design – provided calculations

# 3. CONFIGURATION

## 3.1. Preferences

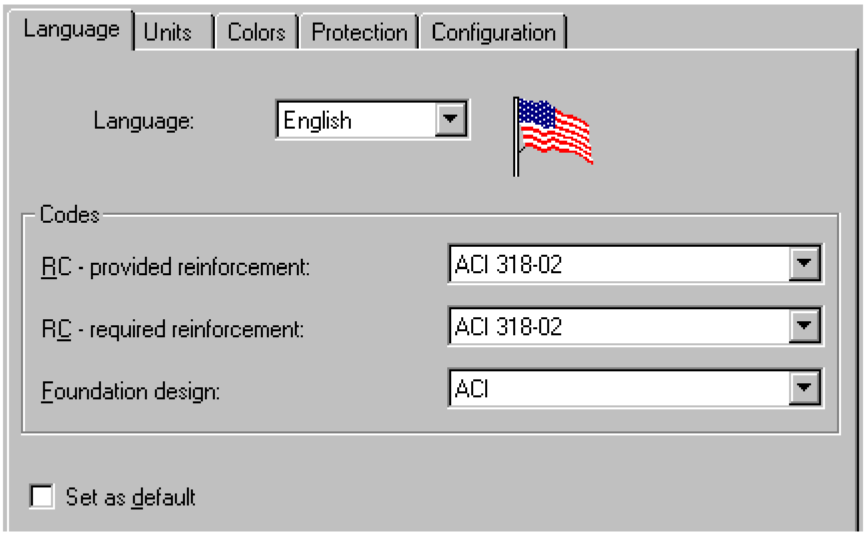
The option enables adopting basic parameters applied in the ***CBS Pro*** program. The option is available from the menu by selecting the *Tools / Preferences* option.



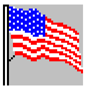

The ***Preferences*** dialog box consists of five tabs:
*Language*
*Units*
*Colors*
*Protection* – the options available on the *Protection* tab of the ***Preferences*** dialog box provide information and are used for program protection support; the tab is used for contact with the technical support
*Configuration*.

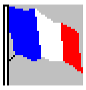
## 3.2. Language

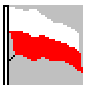
On the *Language* tab in the ***Preferences*** dialog box the user may select a language version of the ***CBS Pro*** program.

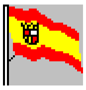


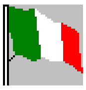

In the current program version the following language versions are accessible:

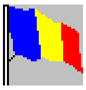

English

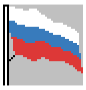

French

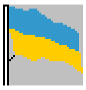

Polish

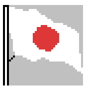

Spanish


Italian


Romanian


Russian


Ukrainian


Japanese.

Options in the lower part of the dialog box allow selection of RC codes that will be used in the program to perform structure calculations as well as design of RC elements and building foundations (the settings refer to advanced calculations and provided RC element design):

- *RC code (calculation of required reinforcement)*

  codes for calculation of the required reinforcement of slabs, walls and raft foundations
  available codes:
  American codes: ACI 318/99, ACI 318/99 (metric), ACI 318/02, ACI 318/02 (metric), ACI 318/02 (Thailand)
  British code BS 8110
  Canadian code CSA A23.3-94
  Eurocode2 (ENV 1992-1-1:1991) with several national application documents (Italian, Belgian, Dutch, Finnish, French, German NADs)
  French codes: BAEL 91 and BAEL 91 mod. 99
  Spanish codes: EH 91 and EHE 99

Dutch code NEN6720 (VBC 1995)
Polish codes: PN-B-03264 (2002) and PN-84/B-03264
Russian code SNiP 2.03.01-84
Romanian code STAS 10107/0-90
Norwegian code NS 3473: 2003
Italian code DM 9/1/96
Singaporean code CP65
Chinese code GB 50010-2002
Japanese code AIJ 1985

- *RC code (calculation of provided reinforcement)*
codes for calculation of the provided reinforcement of all types of RC structure elements
available codes:
American codes: ACI 318/99, ACI 318/99 (metric), ACI 318/02, ACI 318/02 (metric), ACI 318/02 (Thailand)
British code BS 8110
Canadian code CSA A23.3-94
Eurocode2 (ENV 1992-1-1:1991) with several national application documents (Italian and Belgian NADs)
French codes: BAEL 91 and BAEL 91 mod. 99
Dutch code NEN6720 (VBC 1995)
Spanish code EHE 99
Polish codes: PN-B-03264 (2002) and PN-84/B-03264
Russian code SNiP 2.03.01-84
Norwegian code NS 3473: 2003
Italian code DM 9/1/96
Romanian code STAS 10107/0-90
Singaporean code CP65
Chinese code GB 50010-2002

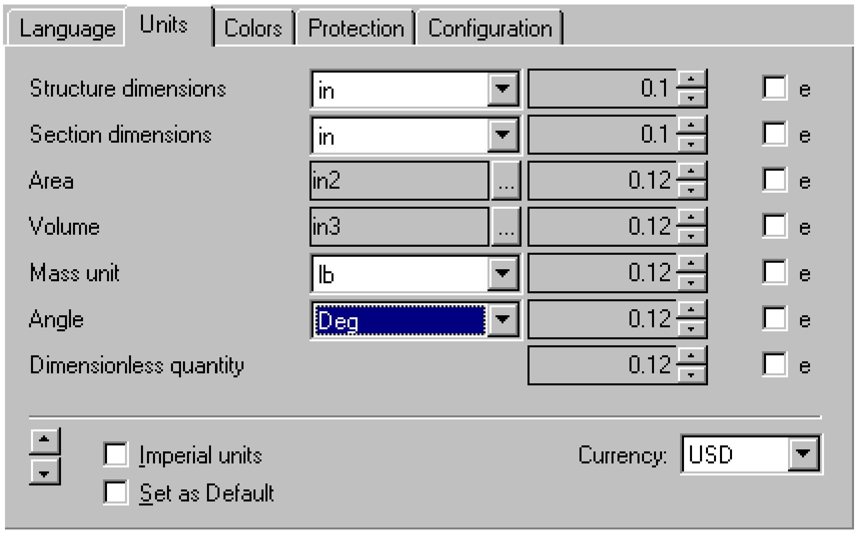
- *RC code (foundation design)*
geotechnical codes which are the basis for foundation design
available codes:
ACI
BS 8004:1986
CSA
DTU 13.12
Eurocode 7 (ENV 1997-1: 1994)
Fascicule 64 Titre V
PN-81/B-03020
SNiP 2.02.01-83.

Code settings are important in the case of structure calculations with the use of the advanced method and provided RC element design; if the estimated design method is chosen, they have no effect on results.

If the *Set as default* option is activated, a selected language and codes are saved as a default set.

## 3.3. Units

On the *Units* tab in the ***Preferences*** dialog box the user may select units applied in the ***CBS Pro*** program:




The dialog box above enables the following operations on the units used in the program:

- change of units of the quantities presented in the dialog box
- change of the display accuracy for individual quantities
- selection of the manner of presenting the values (decimal or exponential system)
- change to the system of units used in the United States (inches, pounds, etc.)
- saving the defined units as a default set
- change of the currency and conversion of all the prices entered by the user (once a new currency is chosen, the program displays a question asking about the conversion factor).

The units have been split into the following categories:

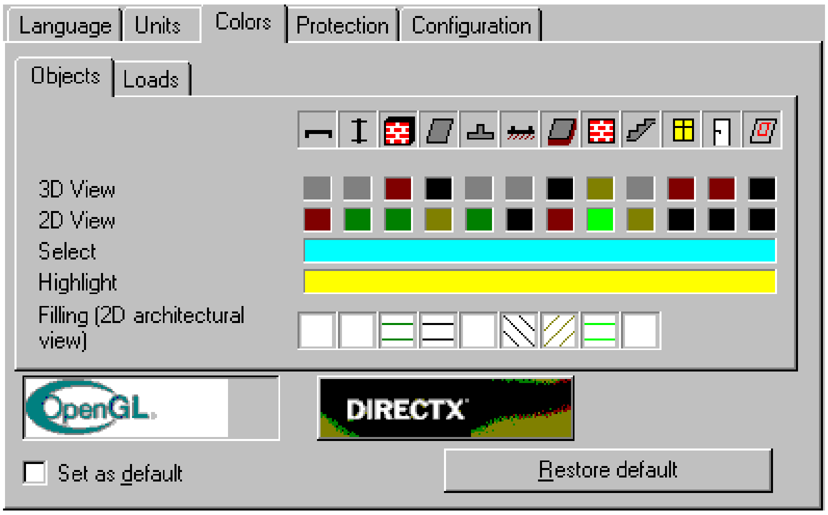
- structure dimensions
- cross-section dimensions
- area
- volume
- mass unit
- stress
- force unit
- force density (unit weight)
- angle
- moment
- displacement
- reinforcement area

- reinforcing bar diameter
- dimensionless quantity.

Not all the units are presented on the tab simultaneously, therefore, the bottom left corner holds the buttons ▲ and ▼, which are used to change the tab contents (after pressing one of the mentioned buttons, selection of other units becomes possible on the tab).
Units are chosen from the drop-down list available for each of the categories. For all the units, the method of presenting the number format for the quantities listed may be changed. In these fields the user may determine a number of decimal places for each of the quantities. To change the number of decimal places, the user should click with the left mouse button on the arrows (the number of decimal places increases or decreases, respectively). The unit precision will be reflected in reinforcement descriptions, dimensions, tables, etc. Switching on the *e* option enables representation of a number value in the exponential form; switching this option off restores the decimal form of a number.
To select a composed unit (e.g. a unit of stress or force density (unit weight)), the (**...**) button should be pressed; then a small dialog box opens, in which units of force and length may be chosen.
Activating the *Imperial units* option results in setting the units for all the above-listed units in conformity with the ones used in the United States (feet, inches, pounds, etc.). If this option is switched off, SI units will be set. To change units in the program (e.g. to imperial ones), it is necessary to close the program and restart it.
After switching on the *Set as default* option, selected units are saved as a default set.

## 3.4. Colors

On the *Colors* tab in the ***Preferences*** dialog box all objects may be ascribed their own colors. It is also possible to select a library assisting in 3D presentation with shading.




The following two tabs are available in the above dialog box: *Objects* and *Loads.*

*Objects* tab
Colors may be ascribed separately to objects presented in:
- 3D view
- 2D view.
Moreover, it is possible to determine colors used for elements selected in structure views and highlighted in structure views.
For elements presented in the 2D architectural view, a color of filling of these elements can be determined.

*Loads* tab
Load colors may be modified; it is possible to select a load color in 2D and 3D views and a color of filling of a planar load in the 2D view.

Pressing the ***Restore default*** button restores the standard set of colors for the available objects. At the bottom of the dialog box a library assisting in 3D presentation with shading may be selected; there are two libraries available:

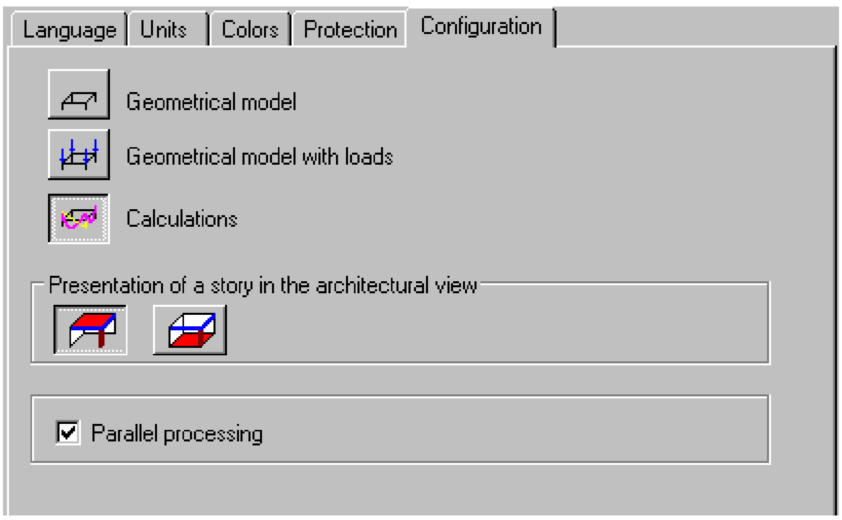
- DirectX
- OpenGL.

Users of Windows® 9x/Me/2000/XP systems may choose the DirectX® or OpenGL® library, whereas Windows® NT system users can only use the OpenGL® library.
After switching on the *Set as default* option, selected colors are saved as a default set.
Comments on application of the libraries assisting in 3D presentations:

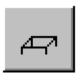
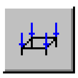
- Prior to changing a 3D-assisting library, close all the viewers with 3D views of a structure; after selecting an appropriate option and pressing the **OK** button, each newly-opened 3D viewer will be applying the selected library assisting in 3D display
- It is not recommended to use at the same time viewers showing 3D views generated by means of DirectX® and OpenGL®
- 3D-assisting library should be adjusted to the graphic card of the computer on which the ***CBS Pro*** program is used; the default 3D-assisting library is DirectX
- In case quality of presentation of a 3D view with shading is not satisfactory (in particular, for older types of cards), it may be helpful to switch off hardware acceleration of some graphic card options (available in the dialog box with advanced card properties in the display settings of the Windows® system). In some cases it may prove necessary to download the latest card drivers (available on the manufacturer's site).

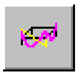
## 3.5. Configuration

The *Configuration* tab in the ***Preferences*** dialog box enables selection of the work mode for the ***CBS Pro*** program.



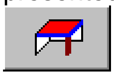

The options included in the above dialog box allow the ***CBS Pro*** program to be run as:

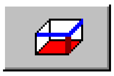
-  geometrical modeler
-  geometrical modeler with the possibility to assign loads

-  geometrical modeler with the possibility to assign loads and perform calculations for a defined structure along with design of RC elements in a structure.

*NOTE:* *If the geometrical modeler option is chosen only, then all the options concerned with loads and calculations in the* **CBS Pro** *program are unavailable; if the geometrical modeler option with the possibility to define loads is selected, all the options concerned with calculations are not accessible; selection of the program work mode requires restarting the program.*

In the lower part of the dialog box there are options used to select the way that a story is presented in the architectural view:

 - presents a slab positioned above the elements of a story; it displays a floor slab above the active story (as in the structural view)

 - presents a slab positioned at the bottom of a story, belonging to the story positioned below; it displays a floor slab at the bottom of the story (architectural views of buildings); elements positioned above the section through the story (beams) are drawn with a dashed line.

In the lower part of the dialog box there is also the *Parallel processing* option; it allows optimization of different operations (link with ***ROBOT***, calculations, loading and display of calculation results) for computers with at least double-core processors; if this option is activated, it allows parallel processing of processors.

## 3.6. Program Personalization

The ***CBS Pro*** program may be adapted to the user's needs by means of several options available in the program (settings of the below-listed options are remembered by the program and set automatically during next program start-up):

- **Working language** – the *Tools / Preferences / Language* option
- **Units** - the Tools / Preferences / Units option
- **Colors** (3D display) - the *Tools / Preferences / Colors* option
- **Default values** – the option refers to sections, materials, names, loads and texts
- **Window layout** – this option is concerned with a number of windows available in the program as well as their mutual position, defined grids, projections, display mode (structural, architectural, computational) and display of objects; these settings may be saved by means of the *Window / Save window layout* option accessible in the menu
- **Print options** – enables setting the printout scale and range
- **Ruler** – enables switching on or off the ruler available on the screen.

# 4. STRUCTURE READING / SAVING

## 4.1. Reading and Saving Options

Basic files of the ***CBS Pro*** program are given the *.geo extension.
In the program the following options allowing file reading or saving are available (the options are included in the *File* menu):

New - opens a new project

Open - opens an existing project (file)

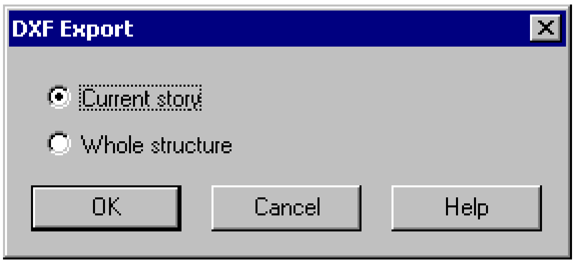
Close - closes the current project (file)

Save - saves the current project (file)
Note should be taken that only the sections and materials used in an example are saved to a file.

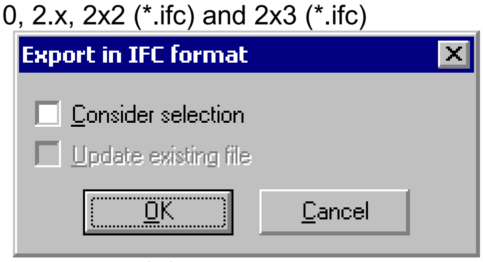
Save as - saves the current project (file) under a selected name

Import - opens a file saved in a different format; the following formats are available:
DXF (*.dxf)
IFC v.1.5, 2.0, 2.x, 2x2 (*.ifc) and 2x3 (*.ifc)
ROBOT CBS (*.rhg)
Adcof (*.add)

Export - saves a file in a different format; the following formats are available:
DXF (*.dxf)
When exporting to a DXF file, all the stories may be exported simultaneously; after selecting the DXF format, the additional ***DXF Export*** dialog box (see the drawing below) is displayed on the screen; there the user may choose whether the export concerns only the current story or all the stories at the same time (if the latter option is selected, then all the stories are placed in one drawing)




IFC v.1.5, 2.0, 2.x, 2x2 (*.ifc) and 2x3 (*.ifc)




Presently, export in the IFC format is possible:
**Consider selection:** only for selected objects (only selected objects will be exported to IFC)
**Update file:** to update a file when saving to an existing file (the existing file will be updated, not replaced).

ROBOT CBS (*.rhg)
Adcof (*.add)

Screen capture - remembers contents of the active window in the clipboard

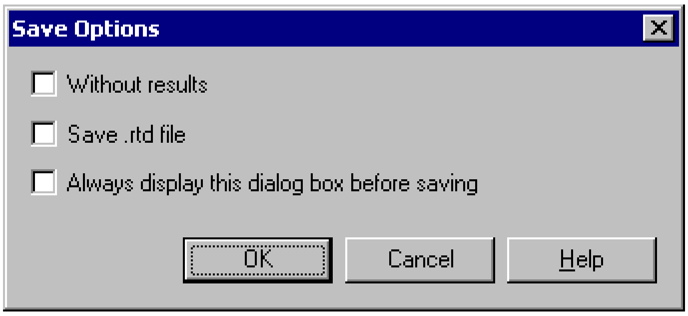
Print - prints the window contents according to the settings in the Print options dialog box

| | |
|---|---|
| Print options | - sets printing parameters (see description of the Print options dialog box) |
| Print preview | - displays a view of the printout before actually printing it |
| Printer settings | - printer parameter settings |
| Exit | - ends work in the program. |

*NOTE:* *The* **CBS Pro** *program enables work in many projects at the same time. When using options allowing advanced structure display simultaneously in many projects, problems with the memory may occur in case of weaker graphic cards. Then it is recommended to close a few projects.*

## 4.2. Save Options

Options provided in the ***Save Options*** dialog box allow selection of elements to be saved in a file. The dialog box opens after selecting the menu command: File / *Save Options.*
Then the dialog box shown in the drawing below appears on the screen.




In the current ***CBS Pro*** version it is possible to save a file with the RTD extension; it is a file generated during advanced calculations with the use of ***ROBOT Kernel*** or ***ROBOT Millennium,*** which allows calculation of the RC panel reinforcement from the level of ***CBS Pro***.
The above dialog box holds the options as follows:

- *Without results* - if this option is selected, a file with the GEO extension is saved without calculation results
- *Save .rtd file* - selecting this option results in saving a file with the GEO extension and a file with the RTD extension of the same name as the file with the GEO extension; an RTD file may be used for calculation of a reinforcement area of RC panels
- *Always display this dialog box before saving* - if this option is selected, each time before saving the file the above dialog box will be displayed on the screen.

First two options have also been added to the ***Save As*** dialog box.

*NOTE*:
In order to enable the design of RC slabs and walls (after closing the file with the GEO extension), the GEO file has to be saved together with the RTD file.

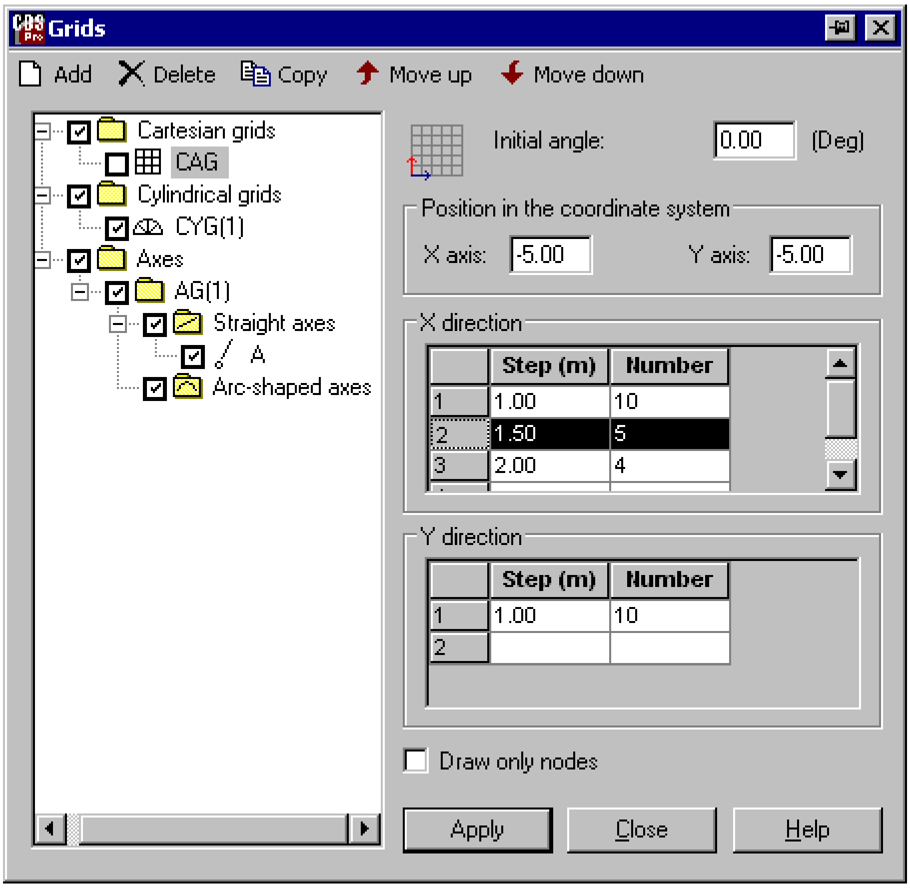
*NOTE*:
1. If calculations have not been performed in ***CBS Pro***, then the *Without results* option is not available
2. If advanced calculations have not been performed in ***CBS Pro***, then the *Save .rtd file* option is not available.

# 5. EDIT OPTIONS / VISUALIZATION

## 5.1. Grids/Axes

The option enables defining Cartesian grids, cylindrical grids, straight axes and arc-shaped axes on the screen. The option is available from:

- the menu, by selecting the *Edit / Grid/Axes* command
- the toolbar, after pressing the *Grid/Axes* icon.




There are the following icons provided in the top part of the dialog box:

- *Add new grid* – pressing this icon adds a new grid or new axes; a grid/axis type depends on the cursor position (i.e. on a selected grid/axis type) before pressing the icon; a grid / axes is/are added to the list of available grids or axes (a default name is assigned to them)
- *Remove grid* – pressing this icon deletes a selected (highlighted) grid or axis.
- *Copy* - pressing this icon copies a selected (highlighted) grid or axis
- *Move up*, *Move down* - pressing these icons moves a table row up /down; the options are available only then if a table includes at least 2 rows and if any row is highlighted in a table.

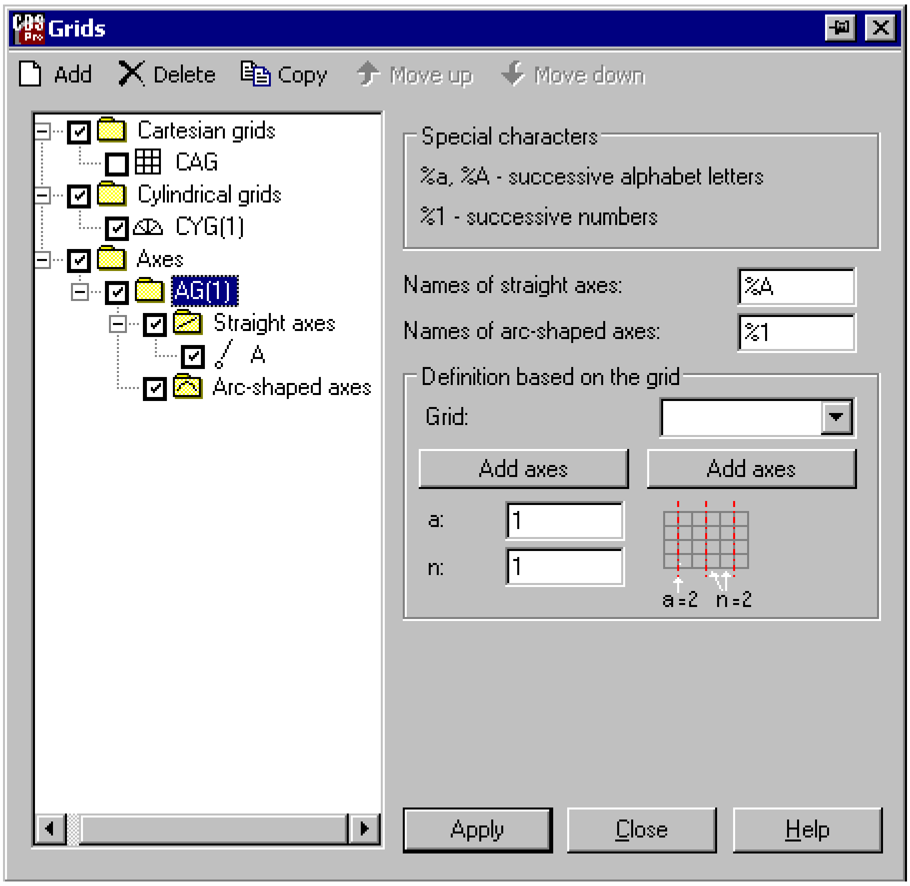
**Cartesian grids**
The following data must be determined for a Cartesian grid:

- *Initial angle* – an angle by which the grid is to be rotated with respect to the global coordinate system
- *Position in the coordinate system* – the bottom left apex of the grid in the global coordinate system
- *Step* – the distance between grid nodes correspondingly in the direction X and Y
- *Number* - the number of grid 'cells' correspondingly in the direction X and Y.

If the *Draw only nodes* option is switched on, then the grid will be presented on the screen in a form of isolated points in the places of nodes. Once the option is switched off, the grid nodes are connected with lines. It is recommended to switch the option off for grids with large cells and to switch this option on for grids with small cells (e.g. 0.1m).

**Cylindrical grids**
For cylindrical grids the same parameters as for Cartesian grids have to be determined and additionally, instead of the step, the angle value (in cylindrical distribution) is defined, in other words, these are the distances between grid nodes correspondingly in cylindrical and radial directions.

**Axes**
Axes have been split into groups. In each group straight and arc-shaped axes can be defined. The user may define axes individually or may generate a group of axes based on the existing grid.




For a given group the user may set generation of automatic name definition – independently for straight axes and arc-shaped axes. The following special characters are allowable:

%a, %A - axis names will be successive alphabet letters (small or capital letters)
%1 - axis names will be successive numbers.
For example, to name Cartesian axes X1, X2, etc. and Y1, Y2, etc., the user should define 2 axis groups and set for straight axes in the first group the name X%1 and for straight axes in the second group the name Y%1.
In the case of axis definition based on the grid, the user should choose a grid from the list of defined grids and next, press an appropriate button (e.g. **Add X axes**) depending on the direction of axis generation (e.g. generation of X or Y axes for a Cartesian grid or straight or arc-shaped axes for a cylindrical grid). Moreover, the user may specify a number of a node in a given direction from which axis generation should start (the *a* edit field) and a number of grid cells between the successively generated axes (the *n* edit field).

*Definition of a single axis*
When defining straight axes, coordinates of two points (x1, y1) and (x2, y2) should be specified. If only one axis is selected (e.g. the *X axis* option is switched on), then only one coordinate is determined.
For definition of arc-shaped axes, the user should specify coordinates of the arc center (the *xc* and *yc* edit fields), value of arc radius, value of initial angle (with respect to the global coordinate system) as well as value of arc angle.
To ensure better readability of final drawings, the user may - for each of the axes independently, switch on display of a name on the left or on the right side of the axis.

Axis names may be changed by the user. They are not subjected then to the mechanism of automatic name definition, which results in refreshing all axis names after each change. If a name is changed in such a manner so that the relevant field is left empty, it causes a given axis to be included again in the algorithm of automatic name definition.
There is an available mechanism of graphical definition of selected grid elements (e.g. grid translation with respect to the origin of the coordinate system, grid initial angle, straight axes). To define that, the user should set the cursor in the edit field and using the mouse define a value of a selected quantity on the screen.

*NOTE:* *While defining axes, the (global or local) coordinate system is considered. As a result, axes defined in one system as parallel to X and Y may, in another system, be axes in an arbitrary direction (e.g. rotated by a given angle).*

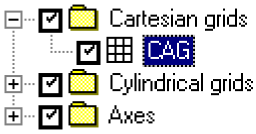
The current program version allows optimization of structural axis definition (quick graphical definition). When defining axes graphically, sequential definition of axes is enabled; in this mode, after defining the last axis point on the screen by means of the mouse, the program accepts automatically the position of the defined axis. Once the axis position is accepted, the next axis can be defined.

## 5.2. How to Define a Grid and Axes

*NOTE:* *The example below illustrates how to define a grid using the options provided in the* **Grids** *dialog box. The program also offers a mechanism for graphic definition of selected elements of grids (e.g. translation of a grid in relation to the origin of the coordinate system, initial angle of a grid, straight axes). To define e.g. an axis graphically on screen, set the cursor in the appropriate edit field (e.g. x1 or x2 for axes) and using the mouse define the position of an axis on screen.*

To define structural grids and axes, follow the steps below:
Definition of Cartesian Grid

- select the menu command *Edit / Grid/Axes* or press the *Grid/Axes* icon
- unfold the *Cartesian grids* list (by pressing the ⊞ symbol), switch on the CAG option (the ☑ symbol appears) – see the drawing below




- for the CAG grid enter the data as follows:

  *Initial angle*: 0.00
  *Position in the coordinate system - X axis*: 0.00
  *Position in the coordinate system - Y axis*: 0.00
  *X direction* - see the drawing below

| | Step (m) | Number |
|---|---|---|
| 1 | 1.0 | 1 |
| 2 | 2.0 | 4 |
| 3 | 1.0 | 1 |

  *Y direction* - see the drawing below

| | Step (m) | Number |
|---|---|---|
| 1 | 1.0 | 6 |

Definition of Cylindrical Grid

- switch on the *Cylindrical grids* option and press the **Add** icon in the top menu

- switch on the *CYG(1)* option and enter the data as follows:

  *Initial angle*: 90
  *Position in the coordinate system - X axis*: 0.00
  *Position in the coordinate system - Y axis*: 0.00
  *Cylindrical direction* - see the drawing below

| | Angle: (De | Number |
|---|---|---|
| 1 | 180.00 | 9 |

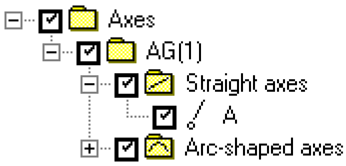
  *Radial direction* - see the drawing below

| | Step (m) | Number |
|---|---|---|
| 1 | 1.0 | 6 |

Definition of Axes

- switch on the *Axes* option and press the **Add** icon in the top menu

- select the field A as in the drawing below




- enter the data as follows:

  *X axis:* ☑
  *y1:* 6.00
  *Description on the right:* ☑

- press the **Add** icon in the top menu and enter the data as follows:

  *Y axis:* ☑
  *y1:* 0.00
  *Description on the right:* ☑

- press the **Add** icon in the top menu and enter the data as follows:

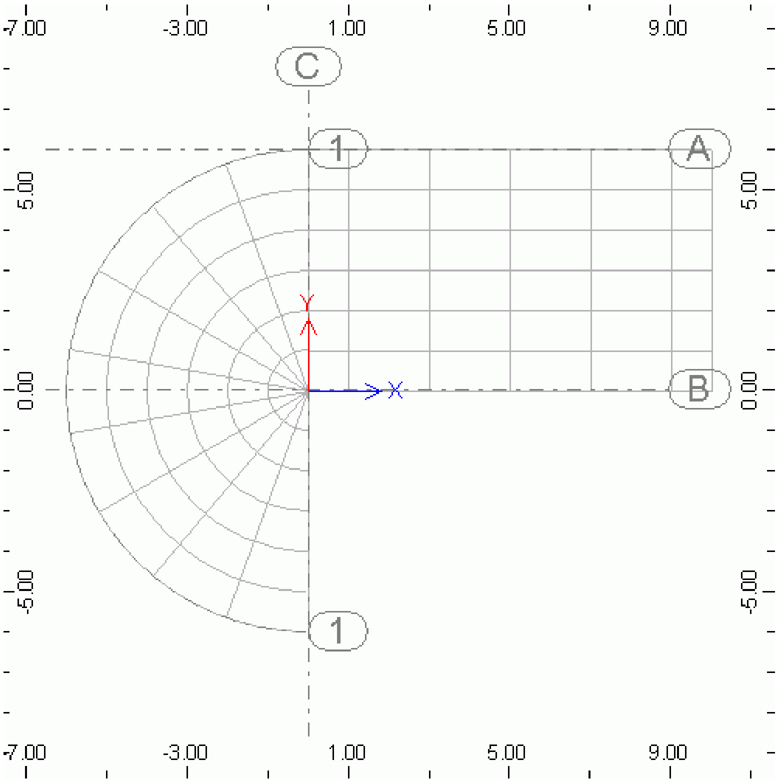
  *Y axis:* ☑
  *x1:* 0.00
  *Description on the right:* ☑

- select the position on the *Arc-shaped axes* list and press the **Add** icon

- select the field 1 and enter the data as follows:

  *xc:* 0.00
  *yc:* 0.00
  *Radius:* 6.00
  *Initial angle:* 90.00
  *Angle:* 180.00
  *Description on the left:* ☑
  *Description on the right:* ☑

- press the **Apply** button; the generated grid composed of the Cartesian and cylindrical grids as well as several axes is presented in the following drawing.




## 5.3. Edit Options

The ***CBS Pro*** program enables the advanced method of work which allows executing certain commands while other commands are under way. In particular, in the course of defining any object, the user may modify the position of the coordinate system, move back to the previous phase of a given command (e.g. move back to the previous point in definition of a slab contour), change current section or material, complete its definition in a different dialog box, open dialog boxes needed. Apart from that, the user may combine modes of manual and graphical data definition.

The are the following edit options available in the program (they are located in the *Edit* menu):

- *Undo* – the option allows the user to go back to the previous stage of structure modeling; last 10 stages are available (10 steps back)

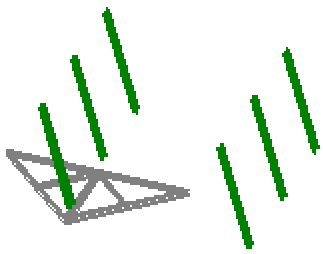
- *Redo* – the option repeats the 'undone' operation; the option is associated with the *Undo* option

- *Cut* – deletes selected objects; the objects are stored in the clipboard so that the user may copy them by means of the *Paste* option to any place of a structure being modeled (on any story)

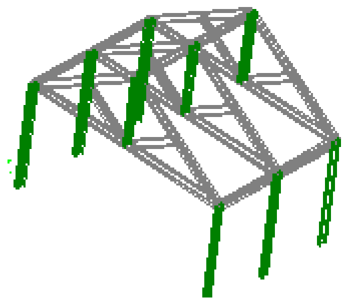
- *Copy* – copies selected elements to the clipboard; they may be copied to any place of a modeled structure by means of the *Paste* option (on any story); while copying to the clipboard, the program takes account of the selection mode (current story or whole structure); after copying a story, the last-defined story is set as a current one

- *Paste* – copies elements from the clipboard to any place of a modeled structure (on any story); the *Paste* option results in pasting a structure copied to the clipboard in the following way: the lowest story in the clipboard is ascribed to the current story in a structure; if the *Paste vertically* option is used, only the lowest story is pasted

- *Paste Vertically* – pastes a structure part from the Clipboard and places it in the vertical plane (see the drawings below); using the *Insertion point* option enables the user to determine the height at which it is inserted; the *Insertion point* option is accessible from the context menu of the ***CBS Pro*** program (as soon as any structure element is copied to the Clipboard and the *Paste Vertically* option is selected)



before executing the operation



after executing the operation

- *Delete* – deletes selected objects

- *Grid/Axes* – the option enabling definition of grids and structural axes; after activating this option, the Grids dialog box opens, in which the user may define Cartesian grids, cylindrical grids as well as straight axes and arc-shaped axes

- *Ruler* – switches on/off the ruler presented on the screen

- *Snap* – the option allowing the user to steer the cursor position while defining a structure model; if this option is switched on, the cursor is snapped to the object snap points

  intersections in grid nodes
  intersections of structural axes
  object ends
  object centers
  axis intersections
  architectural points (points at intersections of lines generated in an architectural view)
  intersections of element axes with grid and structural axes
  graphical objects (from import of an dxf file, but also graphical objects from import of an IFC file); characteristic points include: center, intersection of graphical objects, object ends; if the snap to DXF graphical objects option is switched off, then the cursor will not be snapped to these graphical objects

*Modes* – a set of options which enables different work modes:

- *Drag* - the option concerns edition of objects that require defining two points (beams, continuous footings, walls); if this option is switched on, then the program assumes by default the last point of the previously-defined element to be the first point of a new element; pressing the **Esc** button completes definition

- *Keyboard* – use of the keyboard in definition of beams, continuous footings, slabs, raft foundations or walls causes display of the dialog box which allows entering values from the keyboard

- *Snap to center* - the option concerns edition of objects that require defining two points (beams, continuous footings, walls); while defining the second point of an object, the program locates automatically the center of an object positioned the closest to the cursor

- *Snap – perpendicular* - the option concerns edition of objects that require defining two points (beams, continuous footings, walls); while defining the second point of an object, the program generates automatically a segment perpendicular to an object positioned the closest to the cursor

- *Snap* - the option concerns edition of objects that require defining two points (beams, continuous footings, walls); while defining the second point of an object, the program trims automatically the element being defined to fit an object positioned the closest to the cursor

- *Select* – the option allows object selection; once the cursor moves closer to an object, it is singled out with a different color; there are 2 selection modes available:

  *Point selection*
  A left mouse button click on a chosen element stands for its selection (if additionally, the **Shift** key is pressed, then the element is added to the current selection); if a given element has been earlier selected (highlighted), then the left mouse button click on such an element with the **Ctrl** key pressed at the same time causes the element to be no longer selected.
  *Window selection*
  To perform window selection, the user should – holding the left mouse button pressed (the moment when the button is pressed determines the first point) – move the cursor to any point; the line connecting these two points is the diagonal of a selection window; if the selection window has been defined 'from left to right', then only these objects that are entirely contained in the selection window are selected; if the selection window has been defined 'from right to left', then the objects that have a common part with the selection window, are selected. During the window selection the **Shift** key operates in the same way as for point selection.
  Selection is possible in the 2D view and 3D view; it allows selecting objects using a point (indication with the mouse cursor) and using a window; work of the Shift and Ctrl keys:
  Selection with the Shift key pressed down results in adding an element to the selection (elements already selected earlier still remain selected)
  Selection with the Ctrl key pressed down changes the object selection status: if an object was selected earlier, it stops being selected.

- *Select – current story*, *Select – whole structure*
  Objects can be selected either on the current story or in the whole structure; NOTE: in the current program version the window selection operates on the current story; once the selection is made, the user may:
  - change parameters of selected objects in the dialog boxes: ***Object properties*** and ***Fix section modification***
  - delete selected objects

- design objects on a given story or in the whole structure, depending on the selected option.

If the *Select – whole structure* option is switched on, then in the Criterion of Selection dialog box the program makes accessible the *Story filter* option which enables the operations of selection on the defined stories.

*NOTE:* *The operation of selection is ascribed to a selected view; after selecting objects in the whole structure and switching to other structure view, the selection option chosen for that view is applied.*

*NOTE:* *After opening a new project in* **CBS Pro** *, the 4-function mode: rotation, 2D rotation, zoom and pan is active by default in the 3D view. To switch to the selection mode in this view, the* **Esc** *key should be pressed.*

There is also the *Add Specially* option available in the program (in the *Objects* menu or the *Add Specially* icon on the toolbar). It enables quick definition of slabs in typical situations; if the user selects this option and then clicks with the left mouse button on the area limited by walls or beams, then the program generates a slab on this contour.

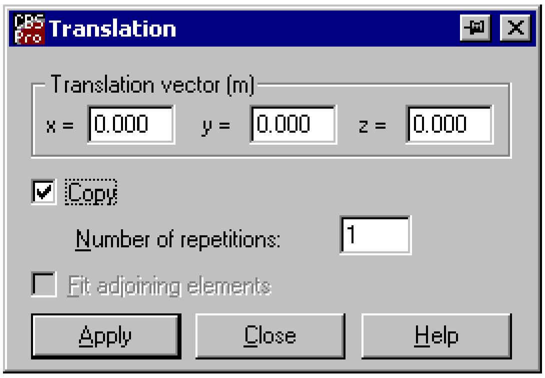
## 5.4. Edit Operations (Translate, Rotate, Mirror, Fit, Trim, Extend)

The ***CBS Pro*** program is equipped with a great number of useful edit tools, which make the user's work in the program easier while defining and/or modifying a designed structure. These options include: translation, rotation, horizontal mirror, vertical mirror, axial symmetry, trimming, extension.

The *Translate* option is used to translate prior-selected nodes / elements of a generated structure. The option is available:

- after pressing the *Translation* icon
- after selecting the *Edit / Operations / Translate* command from the menu.

The dialog box shown in the drawing below appears on the screen.

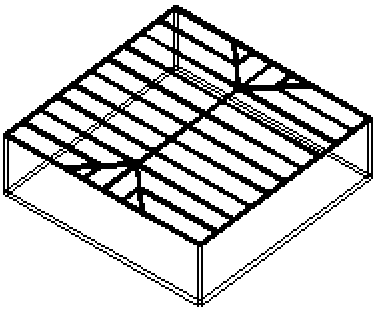


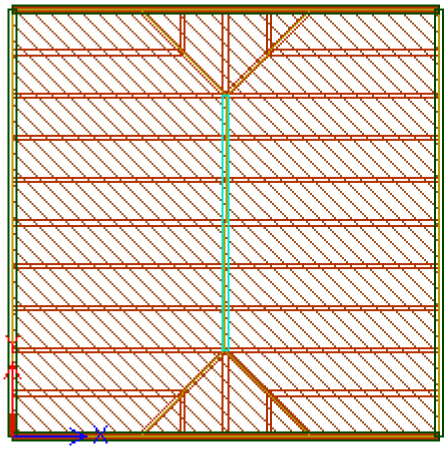

In the fields: *x*=, *y*= and *z*= coordinates of the translation vector should be defined.

If the *Copy* option is turned off, then the elements selected will only be translated. If this option is turned on, then the elements selected will be copied the number of times specified in the *Number of repetitions* field.

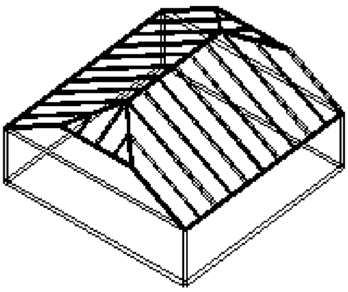
The *Fit adjoining elements* option is available only when the *Copy* option is turned off. If the *Fit adjoining elements* option is turned on, then dimensions and positions of all the objects adjoining the translated object will be fitted to a new position of the translated object. For example, translation of a structural wall results in automatic fitting of structural walls, partition walls, beams, columns adjoining it, to which in turn spread footings are adjusted. This option may also be used for generation of roof surfaces. To do that, the user should:

- define roof surfaces in XY projection
- model a rafter framing (NOTE: also in XY projection) – if the rafter framing is not modeled, then temporary beams located in place of roof ridges should be modeled.

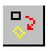




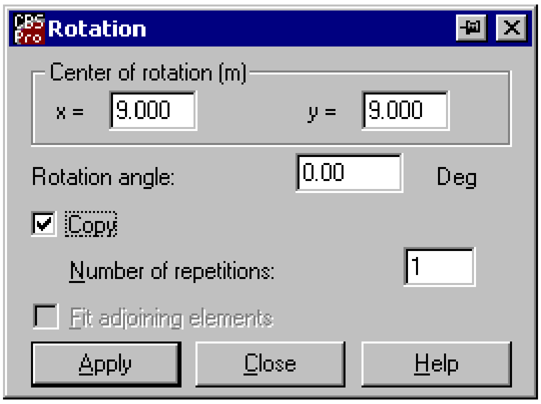
- translate roof ridges to a selected level with fitting of adjoining elements; the roof obtained is illustrated in the drawing below.



The *Rotation* option is used to rotate prior-selected objects in a generated structure. The option is available:

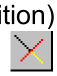
- after pressing the  *Rotation* icon
- after selecting the *Edit / Operations / Rotate* command from the menu.

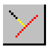
The dialog box shown in the drawing below appears on the screen.




In the fields: *x=* and *y=* coordinates of the center of rotation should be defined, whereas in the *Rotation angle* edit field a value of an angle by which the selected object will be rotated, should be specified.
If the *Copy* option is turned off, then the elements selected will only be rotated. If this option is turned on, then the elements selected will be copied the number of times specified in the *Number of repetitions* field.
The *Fit adjoining elements* option is available only when the *Copy* option is turned off. If the *Fit adjoining elements* option is turned on, then dimensions of all the objects adjoining the rotated object will be fitted to a new position of the rotated object.
The logic of this operation is identical as in the case of the *Translation* operation.

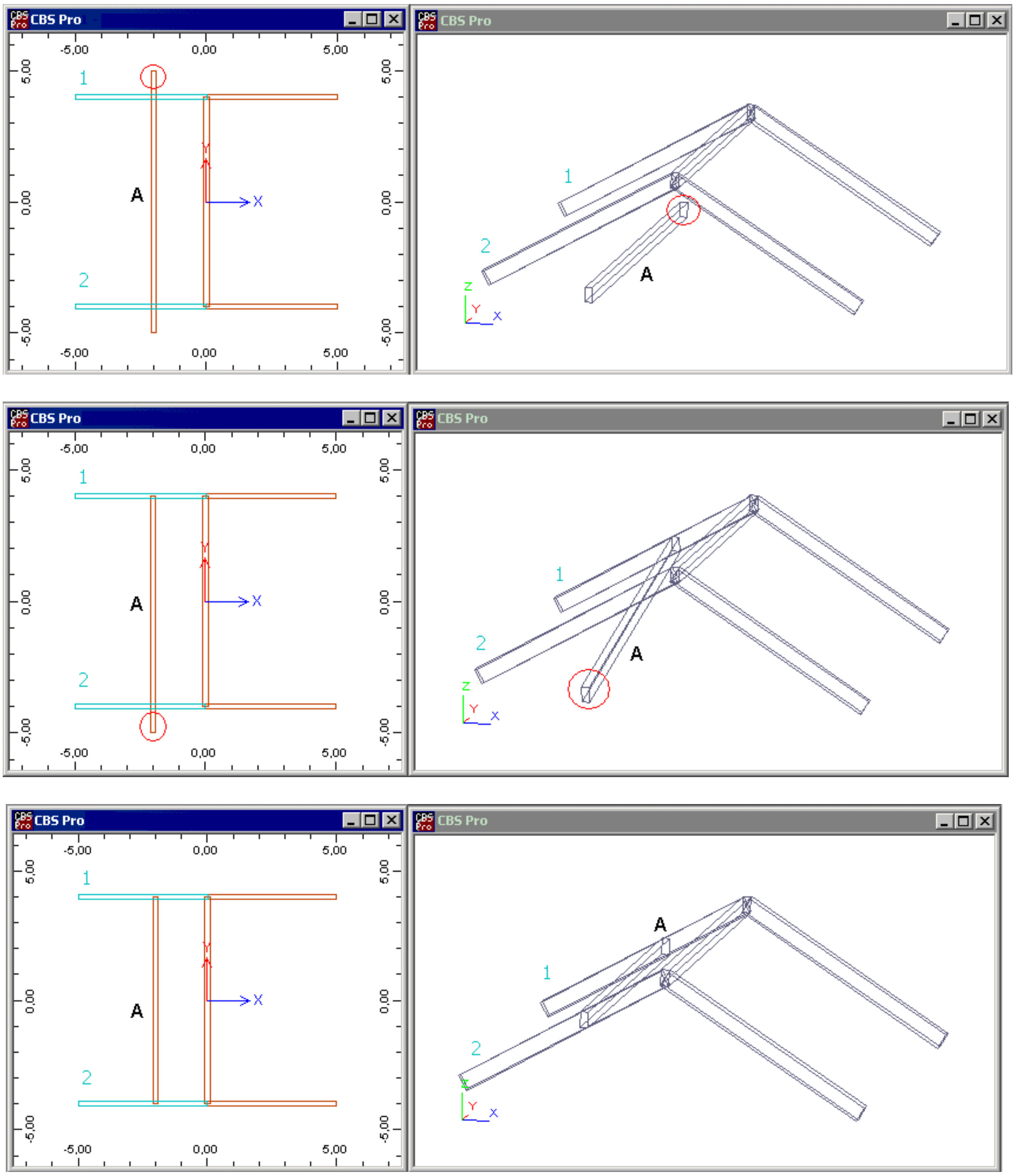
The options: *Vertical mirror*, *Horizontal mirror* and *Axial symmetry* are used for copying a selected structure part with respect to a defined axis (vertical, horizontal or axis in any position).

The  *Trim* option is applied to trim a defined element in such a manner so that it fits other currently selected elements. Once the *Trim* option is chosen, the element part to be removed should be indicated (with the mouse click).

The  *Extend* option is used to extend a defined element in such a manner so that it fits other currently selected elements. Once the *Extend* option is chosen, the element to be extended should be indicated (with the mouse click).

*NOTE:* *It is possible to extend and trim structure objects so that they fit graphic objects (objects resulting from import of a DXF file).*

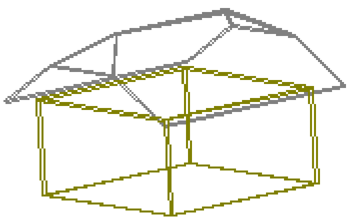
*NOTE:* *If the* Trim *or* Extend *operation is carried out on objects (beam, continuous footing) positioned on different levels, i.e. on objects that do not intersect (objects are selected in an object projection), then besides trimming or extending an object, the program performs the operation of shifting the trimmed / extended object to the plane where the objects are positioned, with respect to which the given object is being trimmed / extended. The example of the Trim operation is illustrated in the drawings below – beam A is trimmed to fit beams 1 and 2.*



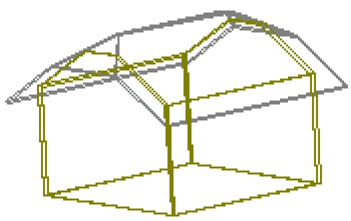

The *Fit Vertically*  option is used to adjust vertically the elements so that they fit the intersections with the selected structure elements. Below are described possible ways of using this option for individual types of structure objects.

**1. Walls, partition walls**

They may be trimmed to fit the planes formed by slabs and beams; as regards beams, each of them constitutes an independent plane made up of the beam itself and two horizontal straight lines, perpendicular to the beam, passing through its ends (see the drawings below).



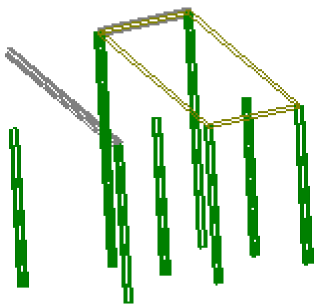
before executing the operation



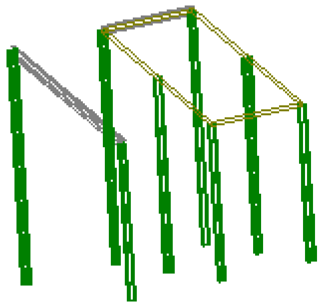
after executing the operation

**2. Columns, beams, continuous footings**

They may be trimmed to fit the planes formed by slabs and beams (see the drawings below).



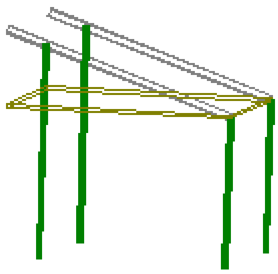
before executing the operation



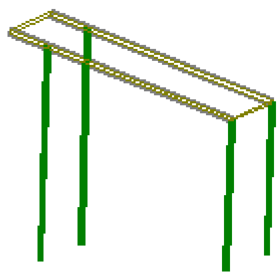
after executing the operation

**3. Slabs, raft foundations**

They may be projected onto the plane formed by the selected elements (see the drawings below).



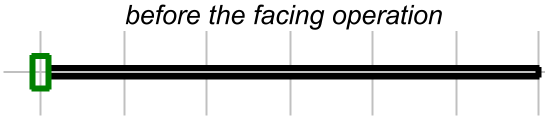
before executing the operation



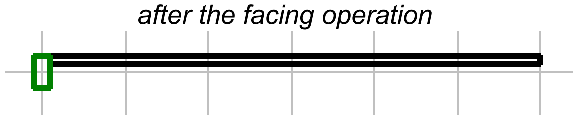
after executing the operation

The *Face* option is used to fit an element to a structure element indicated earlier. NOTE: It is important on which element's side the user clicks with the mouse on the element fitted to a structure element indicated earlier. For example, the drawing below illustrates the operation of facing a wall to a column.

*before the facing operation*



*after the facing operation*



The facing operation causes translations of an object with respect to the axis; such a translation can be viewed in the architectural view, whereas in the real model, translations are not included, which can be seen in the structural and computational views.

*NOTE:* *In the current program version, the facing operation does not cause changes in the position of objects in the structural and computational models; they are still positioned on their axes.*

Facing has effect on the position of continuous footings under walls and spread footings under columns; if continuous footings and spread footings are defined in a structure model, then the position of continuous footings and spread footings changes automatically if the facing is modified.

Translations can also be defined (or modified, as regards the existing ones) in the ***Object Properties*** dialog box on the *Position* tab.

*NOTE:* *The edit operations:* Trim, Extend, Fit Vertically *and* Face *are available both in the 2D view and 3D view. If these options are switched on without selecting earlier objects in the structure view, then first the base object should be indicated (the one with respect to which objects should be modified), and next, an object to be modified.*

Other edit operations available in the program (the *Edit / Operations* menu) are the *Group* and *Ungroup* options. They allow grouping / ungrouping selected structure objects of the same type (beams, continuous footings, walls); objects joined into groups must touch each other.
Grouped objects may be displayed in the 2D and 3D views by means of the Display Object Groups option (the menu *View / Display Object Groups*). Grouped objects are displayed and selected as one object.
To display results of static calculations in the ***Object Properties*** dialog box, the user should switch off display of element groups for beams, columns and continuous footings.
How the option works for individual object types:

- when beams are grouped, the program creates a multi-span beam (if there are more than 2 supports) or a beam composed of several segments that may be ascribed different sections or materials. It is significant if a structure is calculated by means of simplified methods (method 1: trapezoidal and triangular method) and also makes selection of a beam easier.

- when grouping columns that are positioned on different stories (not adjoined to by floors or beams), grouped columns are treated as uniform objects as regards carrying of horizontal forces in simplified calculations ; NOTE: it is possible to calculate the provided reinforcement taking account of the buckling length and internal forces within a whole group of columns, instead of individual segments. A group of columns is presented as a whole in the 3D view of the entire model, while views of individual stories show individual columns belonging to the current story; a name of the column group does not include numbers of stories (it is presented if object groups are displayed in the 3D view), in the remaining cases the program displays names of individual columns included in a group, whose names comprise by default a story number

- when walls are grouped, it is possible to take account of rigidity of the wall group as regards carrying of horizontal forces; the program my display internal forces carried by the whole wall group or by individual component walls; NOTE: to keep continuity of the diagram of moments due to horizontal forces in the wall group, walls between stories should be grouped

- when continuous footings are grouped, it is possible to take account of grillage work of continuous footings; it contributes to reduction of dimensions of continuous footings as regards carrying of moments due to horizontal forces induced by seismic impact or a wind load.

If display of object groups is switched off, then forces in component walls are presented for simplified calculations.

*NOTE:* *The* Group *option creates a group of objects for purposes concerned with calculations and edition. The other type of grouping possible in the program refers to the design of RC elements and may be activated in the* **General Parameters** *dialog box for selected RC elements.*

Orientation of linear elements (beams, walls, partition walls and continuous footings) is defined by the beginning and end of an object; thus it depends on the order of defining individual nodes. The *Change orientation* option is used to change the direction of selected linear elements, which consequently may lead to the global uniforming of the direction for linear elements.
After activating the option, the user should define a direction by indicating two points whose order determines the orientation. Coordinates of all selected linear objects are modified in such a way so that their projection on a defined axis is positive. A perpendicular object should be projected on the axis perpendicular to the defined axis (sense: to the left of the defined axis).

The *Distance* option is used to determine distances between indicated points. Once the option is switched on, the user should select two points between which the distance should be determined. The program draws a line between the selected points and displays a distance between them. Both ends of the segment connecting the beginning and end points are marked with lines perpendicular to the line connecting the two points. Moreover, a distance is presented on the status bar.

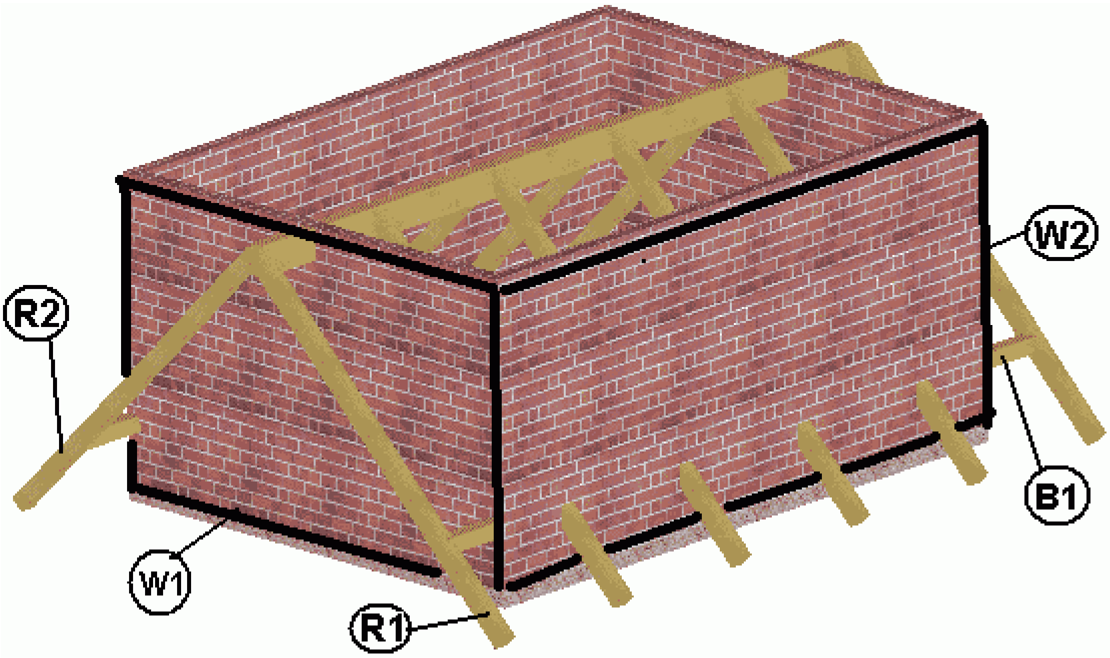
*NOTE:* *It is possible to measure distances to characteristic points of objects:*
*- center of a linear object / slab edge*
*- perpendicularly to a linear object / slab edge*
*- point of intersection with a linear object / slab edge*
*after activating the relevant edit mode (snap to center, snap – perpendicular, snap). When determining a characteristic point, the user should follow the principles analogous to those applied when creating new objects with the use of a selected snap mode.*

Clicking with the left mouse button changes the beginning point; the size of fonts used to present distances may be changed by pressing the keys **PgUp (**to increase the font size) or **PgDn** (to decrease the font size) on the keyboard.

*NOTE:* *The* Distance *option is active only in the 2D view.*

## 5.5. How to Fit Walls to the Roof

To fit walls to the roof structure (see the drawing below), follow the instructions presented below:

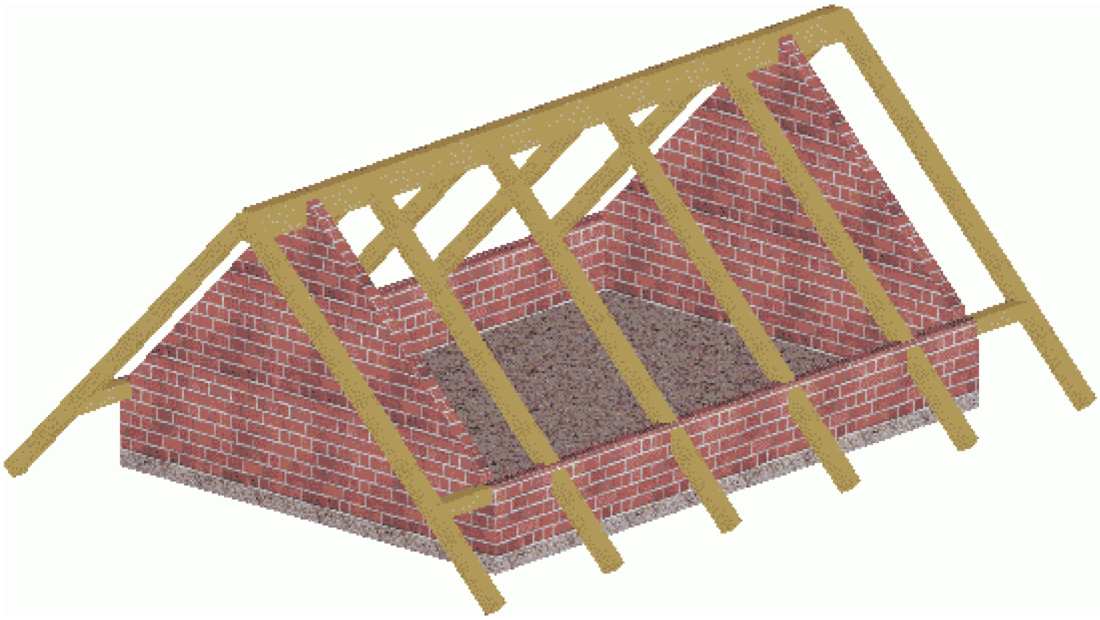



- press the right mouse button and choose the *Select* option from the context menu
- holding the **Ctrl** key down, select two rafters (R1 and R2)
- select the menu command Edit / Operations / Fit Vertically or press the Fit Vertically icon
- select the wall W1 to be fitted to the position of the rafter
- press the right mouse button and choose the *Select* option from the context menu
- select the tie beam B1

- select the menu command Edit / Operations / Fit Vertically or press the Fit Vertically icon

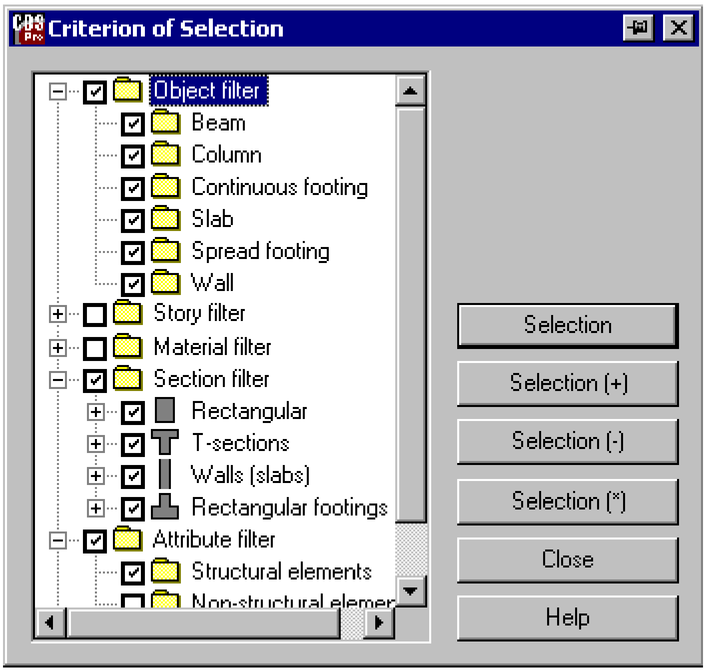
- select the wall W2

- repeat the action for the successive walls; the walls fitted to the roof structure are presented in the drawing below.



## 5.6. Selection Criteria

The option is used to define criteria of selection. The option is available after choosing the following command from the menu: *Edit / Selection criteria*. Then the dialog box shown in the drawing below is displayed on the screen.




Clicking with the mouse cursor on the selection field next to a given filter (√ symbol appears) results in activating the selected criterion of selection. To define a detailed condition of

selection, the user should expand a chosen filter and switch on the options in the selected fields (√ symbol appears again). For example, to select all the beams defined in a structure, the user should expand the *Object filter* option by clicking on the '+' symbol and next, switch on the *Beams* option (√ symbol appears).
Once the *Name filter* option is turned on, the user should enter a new filter name in the edit field that appears. The following special characters may be used:
* - application of this character substitutes any character string
? - application of this character substitutes a single character.

In the program the following selection modes are enabled (they are activated by pressing an appropriate button):

- **Selection** – after pressing this button, elements that satisfy determined criteria are selected

- **Selection(+)** - after pressing this button, elements that satisfy determined criteria are added to the current selection

- **Selection(-)** - after pressing this button elements that satisfy determined criteria are subtracted from the current selection

- **Selection(*)** - after pressing this button, elements that constitute a common part of the current selection and the elements satisfying determined criteria, are selected.

For example, to select all the objects whose material is different than concrete, the user should:

- select objects, for which material can be defined (all the objects except for dimension lines, rooms, etc.) and press the **Selection** button

- switch off the previous criterion, set the *Material filter* criterion with the material *Concrete* chosen and press the **Selection(-)** button.

## 5.7. How to Define a Selection Criterion

To select the timber – pine beams of rectangular section 20x20 and all the concrete columns in the current story of a building by applying selection criteria, follow the steps below:

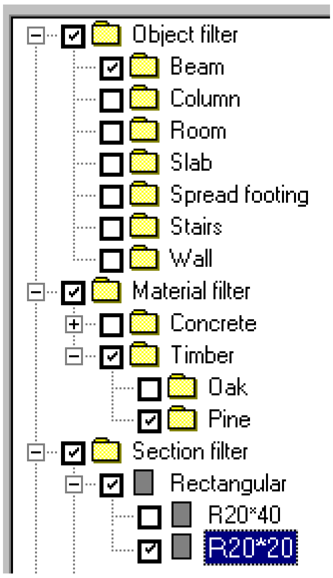
- select the *Edit / Select - Current Story* command from the menu

- select the menu command Edit / Selection Criteria

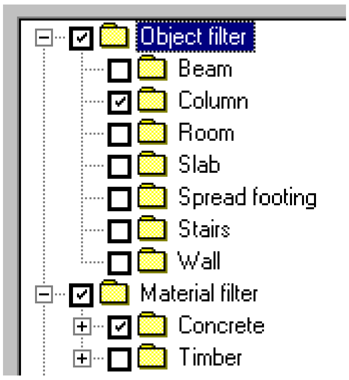
- unfold the Object filter list (by pressing the ⊞ symbol) and switch on the Beam option

- unfold the Material filter list, the Timber sublist and switch on the Pine option

- unfold the Section filter list, the Rectangular sublist and switch on the R20*20 option




- press the **Selection** button; it has resulted in selection of the timber – pine beams of rectangular section 20x20
- select again the menu command *Edit / Selection Criteria*
- unfold the *Object filter* list (by pressing the ⊞ symbol) and switch on the *Column* option
- unfold the *Material filter* list and switch on the *Concrete* option




- press the **Selection (+)** button; all the concrete columns have been added to the timber beams already selected.

## 5.8. Model Presentation on Screen (Views, etc.)

The program offers access to the following options that enable changing presentation of structure model elements (the options are available in the *View* menu):

- *Synchronize views* – if this option is switched on, then all the windows are ascribed parameters concerned with the active window, i.e. story number, projection, display mode (structural view, architectural view) and settings with regard to grids and object visibility
- *Structural view* - if this option is switched on, then objects (beams, walls, slabs, etc.) are presented as in a calculation model (i.e. in axes); representation of a single object does not depend on its position with respect to other objects
- *Architectural view* - if this option is switched on, then architectural requirements are considered in the object presentation, in particular:

- graphical representation of a given object depends on its position with respect to other objects (e.g. intersection of walls)
- drawing in the XY plane is generated automatically as a section, which results in application of different line thicknesses for elements in view and in cross-section
- descriptions needed are added automatically (e.g. descriptions of windows, doors)
- rooms are described automatically (number, name, area, finishing layer)

- *Computational view* – if this option is switched on, then computational requirements are considered in the object presentation, in particular:

  - a 3D view presents data concerning the structure (e.g. loads) and calculation results
  - a 2D view presents division of multi-span beams into spans of the beams

- *Toolbars* – the most frequently used options are available on toolbars; icons have been grouped on the following toolbars:

  Standard - system options
  Story - options concerned with stories
  Objects - options enabling definition / modification of available objects
  3D - options concerned with 3D presentation
  Edit - edit options
  Edit-modes - edit options connected with the available modes of element definition
  Properties - options concerned with sections and materials
  Conversion of lines - options that allow converting lines to selected objects (walls, beams, etc.)
  Dimension lines - options that allow defining structure dimension lines
  Coordinate system - options that enable performing operations on the coordinate system
  Snap - options that manage the cursor position during definition of a structure model
  Zoom - options that manage structure zoom
  Loads - options that enable definition of structure loads
  Calculations - options that enable calculation of the structure and design of the RC elements in the structure

- *Status bar* – if this option is switched on, then a line is displayed in the bottom part of the program window, where different information may be presented (coordinates of the cursor position, height of the current story, parameters of the object currently highlighted, etc.)

- *Zoom* - options managing structure zoom

  *Zoom window* - defines degree of structure window zoom
  *Zoom in* - zooms in a structure view
  *Zoom out* - zooms out a structure view
  *Zoom all* - returns to the initial view (fits the current zoom in such a way so that the whole structure is shown in the program window)

NOTE: *If the mouse has the wheel button, then it is possible to zoom in a structure view by means of the mouse wheel; a view can be zoomed in 'to the point' at which the mouse cursor is positioned (the mouse cursor is a "target positioner")*

The options above may be also activated from the context menu or using shortcut keys.

- *Projection* – allows setting a required view (projection); the following projections are accessible in the current program version:

  XY - standard work plane
  3D (building) - 3D display of the entire structure without the possibility of edition
  3D (story) - 3D display of the current story without the possibility of edition

- *Display* – selection of this option opens the dialog box in which display parameters may be set

- *3D View* – options managing 3D view

  XY projection - projection onto XY plane
  XZ projection - projection onto XZ plane

YZ projection - projection onto YZ plane
3D projection - display of a structure in any position
Model with rendering - display of a structure with object rendering switched on
Skeleton model - objects are presented only by means of edges
3D view comprises 3 default structure projections: ZX, XY, YZ corresponding to the front view, top view and side view; the options are available in the menu *View / 3D View / Projection*, on the 3D View toolbar and in the context menu on the 3D layout / Projection (there are also keyboard shortcuts accessible: CTRL+ALT+1, CTRL+ALT+2, CTRL+ALT+3 as well as return to the initial projection CTRL+ALT+0)

*Parallel projection* – switches on the axonometric structure view (the option is only available for 3D view) – a view in parallel projection (a view without perspective compression)

*Perspective projection* - switches on the perspective structure view (the option is only available for 3D view); it is a structure view involving perspective compression; this compression is equivalent to the compression for wide-angle lenses in photo or video cameras (a perpective view enables 'entering' inside the object and recording a 3D movie presentation)

- *Coordinate system* – options for managing a position of the coordinate system

*Translate* – translates the origin of the coordinate system by means of the mouse, the origin of the coordinate system is translated to the point indicated by the user on the screen (the mouse-click point)

*Rotate* – rotates the coordinate system by means of the mouse; the rotation angle is calculated based on positions of two points defined with the mouse (the axis of the coordinate system is rotated in such a way so that the X' axis creates with the X axis the angle defined by the user on the screen)
NOTE:
Calculation results for slabs, walls, raft foundations, that are available in the ***Properties*** dialog box, are presented in the current coordinate system (if the system is translated, the coordinates are translated as well, and if it is rotated, the slab is rotated in the graphical viewer).

*Global system* – restores the initial (default) position of the coordinate system

*Define* – option that enables defining a position of the coordinate system using the options provided in the ***Coordinate system*** dialog box; data determining translation or rotation of the coordinate system may be entered into appropriate edit fields located in the dialog box or defined graphically on the screen (to do so, the user should set the cursor in the edit field and enter data using the mouse)

*According to object* – sets the coordinate system according to the local system of the selected object; an object should be selected with the mouse; position of the coordinate system depends on a mouse-click point - the system will be adopted on this object end that is positioned closer to the point selected with the mouse

- *Previous / next story* – options managing display of a story view

When defining new elements (beam, column, etc.) it is possible to insert them at characteristic points (see options available in the *Snap* toolbar) indicated on elements from the previous / next story, analogously as for components from the current story; additionally, they may be snapped to intersections of objects from the current and previous / next story.
Remarks on the options *Previous / Next story*:
1) after activating the option, the √ symbol appears next to the option's name (these options cannot be activated simultaneously)
2) elements from these stories are shown in a different color (identical to the grid color)
3) these elements cannot be selected
4) elements are displayed together with descriptions (if descriptions are switched on).
Characteristic points of elements from the previous and next stories are recognized by the program except when the *Dimension Lines / Automatic* option is active (according to

remark no. 3, these elements cannot be selected; after selecting objects and moving to the adjacent story, the automatic dimensioning operation will be performed for the elements selected in the current story).

## 5.9. 3D View

3D view can work in one out of five modes:

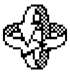
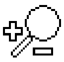
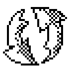
- four simple modes: rotation, 2D rotation, zoom and pan
- one multi-functional mode.

*NOTE:* *After opening a new project in* **CBS Pro**, *the 4-function mode is a default work mode in the 3D view. To switch to the selection mode in this view, the* ***Esc*** *key should be pressed.*

The user may switch between the work modes by choosing an appropriate menu option *View / 3D View* and on the *3D View* toolbar. Once the work mode is selected, a movement with the mouse (with its left button pressed) causes the relevant modification of the 3D view:

- Rotate – structure rotation in all planes
- Rotate 2D - structure rotation in the plane that is parallel to the plane of the screen
- Zoom – movement 'deep down' the view – structure zoom in / zoom out with respect to the plane of the screen
- Pan – movement in the plane of the view (structure pan with respect to the center of the screen).

The multi-functional mode (Rotate / Zoom / Pan) enables work with all the modes at the same time. The 3D view layout is split into quarters and to each of them one mode is ascribed:

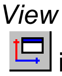
-  top left: rotation
-  top right: pan
-  bottom left: zoom
-  bottom right: 2D rotation.

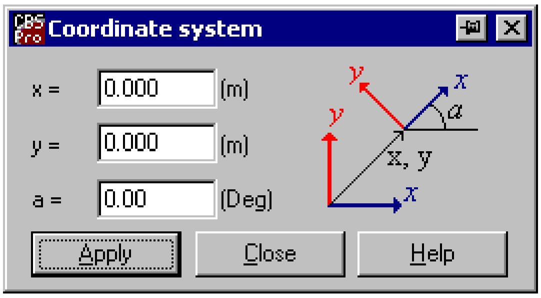
After positioning the cursor in an appropriate quarter, the cursor shape changes (see the icons above).

Moreover, it is possible to activate structure rotation in the 3D view using shortcut keys and the mouse: to do it, the user should press:
- Alt (on the keyboard) and LMC (**l**eft **m**ouse button **c**lick)
- Shift (on the keyboard) and RMC (**r**ight **m**ouse button **c**lick).

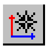
## 5.10.Coordinate System

The option enables defining a position of the coordinate system. The option is available from:

- the menu after choosing one of the options included in *View / Coordinate system / Define*
- the toolbar after pressing the Define coordinate system  icon.




Data determining translation or rotation of the coordinate system may be entered into appropriate edit fields located in the dialog box or defined graphically on the screen (to do so, the user should set the cursor in the edit field and enter data using the mouse). In the case of translation, the origin of the coordinate system is translated to the point indicated by the user (the mouse-click point); if rotation is performed, the axis of the coordinate system is rotated in such a way so that the X' axis creates with the X axis the angle defined by the user. NOTE: Calculation results for slabs, walls, raft foundations, that are available in the ***Properties*** dialog box, are presented in the current coordinate system (if the system is translated, the coordinates are translated as well, and if it is rotated, the slab is rotated in the graphical viewer).
There are also the following options available in the program:

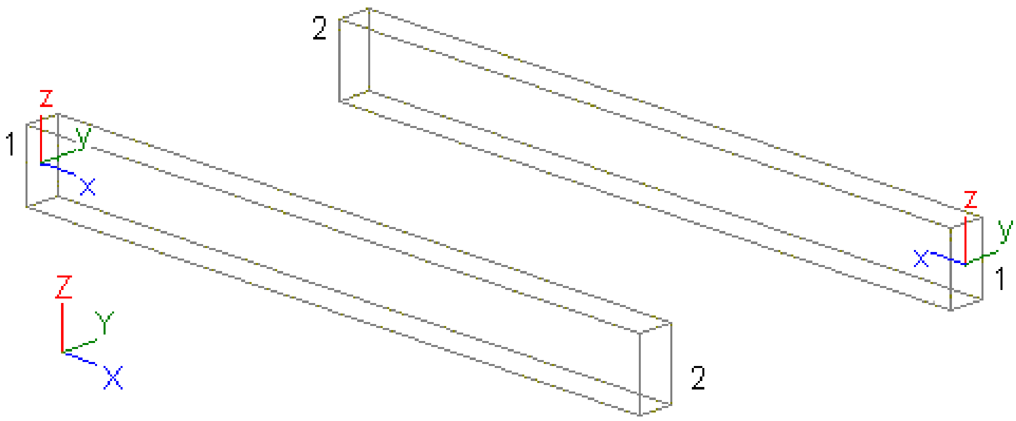
 *Global system* – restores the initial (default) position of the coordinate system

 *According to object* – sets the coordinate system according to the local system of the selected object; an object should be selected with the mouse; position of the coordinate system depends on a mouse-click point - the system will be adopted on this object end that is positioned closer to the point selected with the mouse.

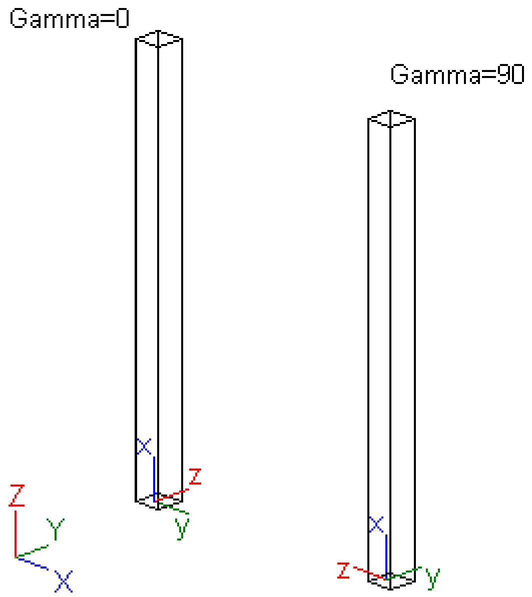
## 5.11. Local Coordinate Systems of Objects Defined in CBS Pro

When defining an object in the ***CBS Pro*** program, it is ascribed a local coordinate system. the local system depends on an object type and orientation:

**1. Beam, continuous footing**
The local system is presented in the drawing below.



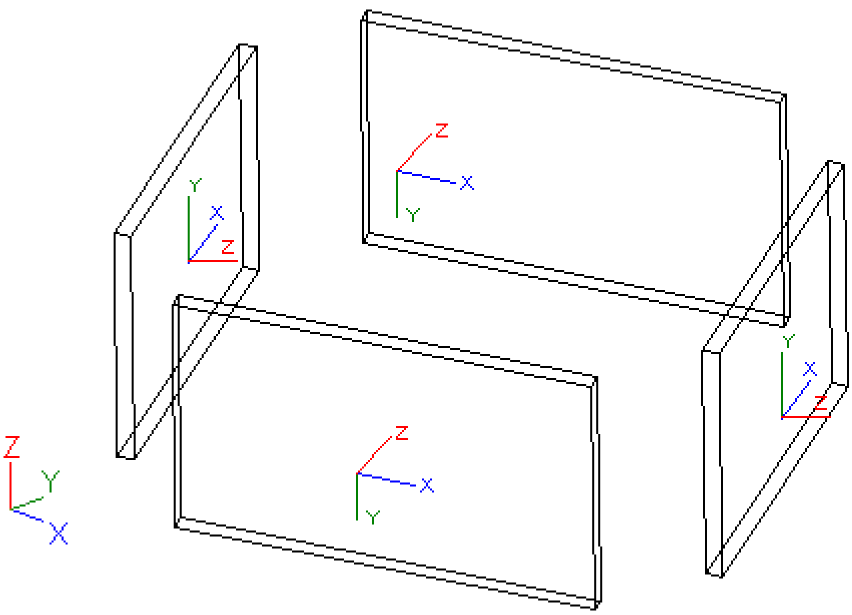

**2. Column**
The local system is presented in the drawing below.




*NOTE:* *In the case of columns, orientation of local systems differs from that in* **ROBOT;** *the local system of a column defined in* **ROBOT** *is congruous with the local system of a column rotated by the GAMMA angle = 90 degrees in* **CBS Pro***.*

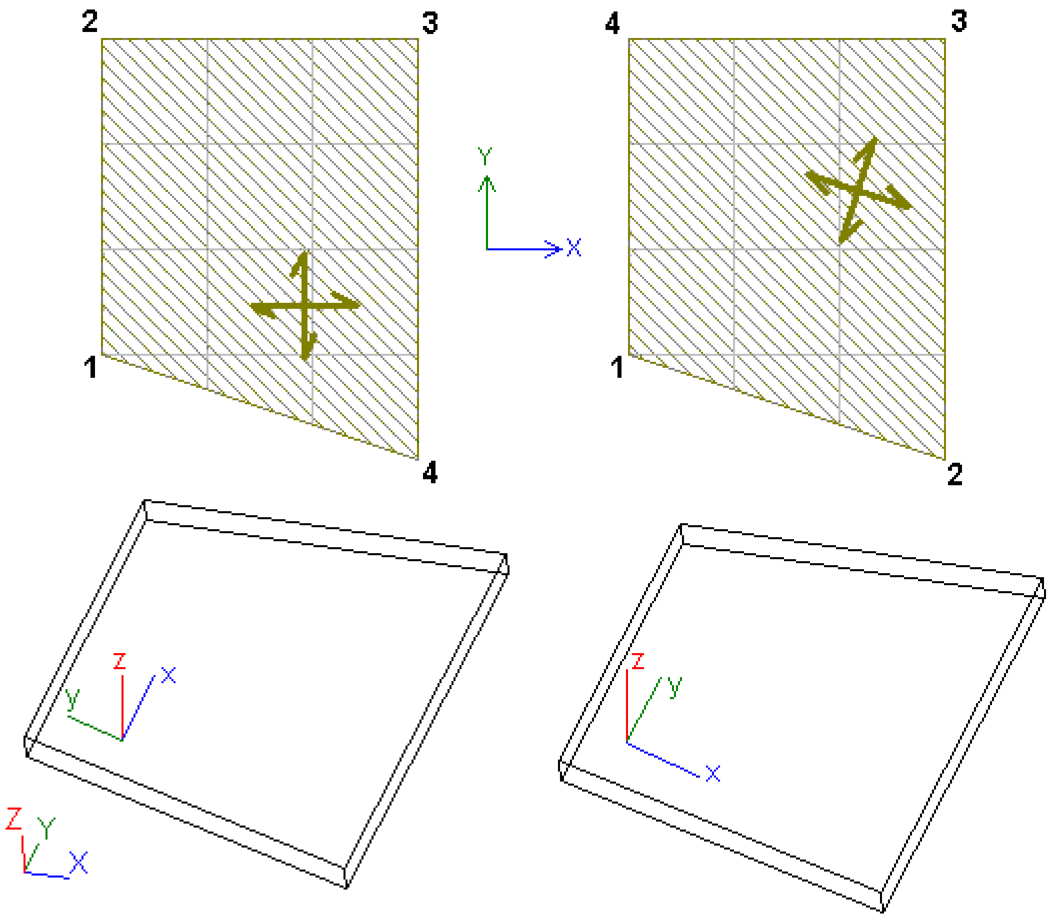
**3. Wall**

The axes x and y of the local system lie in the plane of a wall. The x axis is horizontal, while y vertical. Orientation of a wall has no effect on the sense of the x axis (always in the positive direction of the X axis of the global system).
Examples of walls and definitions of local systems are presented below.




**4. Slab**

The x axis of the local system is defined by positions of first two apexes of a slab. The order of defining apexes determines the sense of the local x axis.



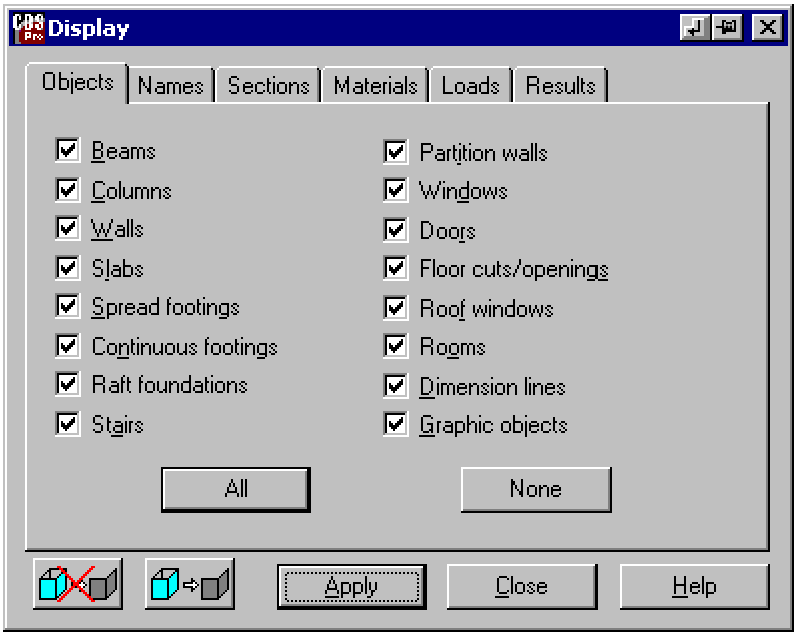

**5. Raft foundation**
Analogous principles as for a slab are applied here, however, the Z axis is pointed down (oppositely to the Z axis of the global system).

**6. Stairs**
The local system as for slabs is adopted.

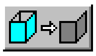
**7. Spread footing**
The local system congruous with the global coordinate system is adopted. If a spread footing is rotated by a GAMMA angle, the local system is rotated too.

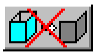
## 5.12.Display

The option enables setting display parameters. The option is available from the menu by selecting the option: *View / Display*.

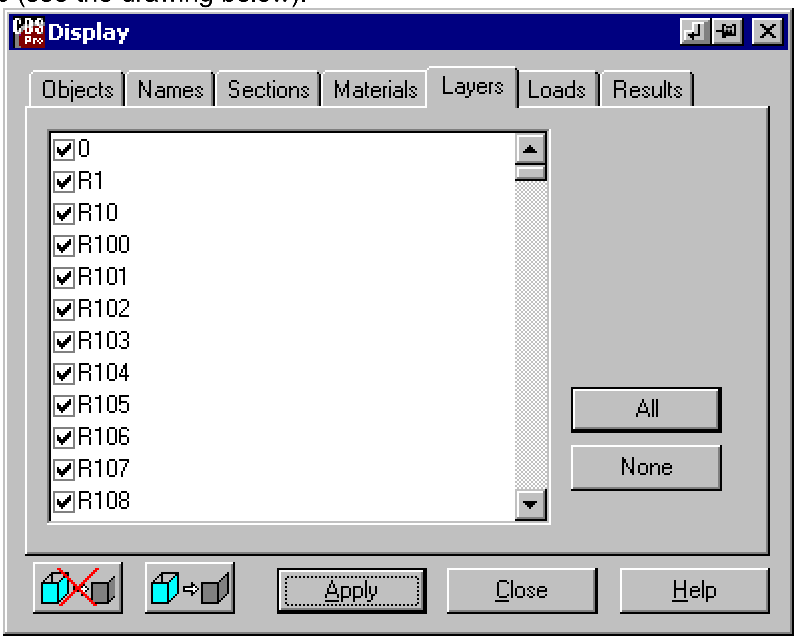



In the above dialog box, the user may select the model elements to be presented on the screen; these may include objects available in the program, sections assigned to them, materials, names, results or loads (they are selected independently for each object type).
At the bottom of the dialog box there are two icons:

 - pressing this icon results in displaying only selected elements in a chosen structure view (NOTE: at least 1 object has to be selected in a structure view); objects will be displayed in a structure view without being highlighted

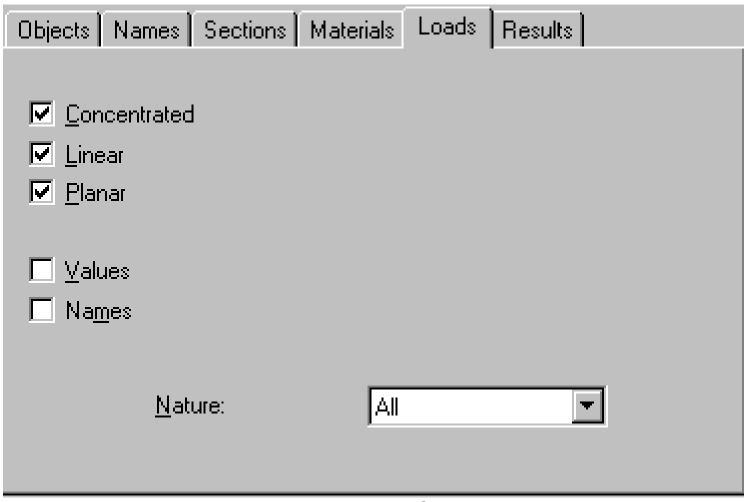
 – pressing this icon restores the visibility of all elements (not only selected ones) in a chosen structure view; this is an operation opposite to displaying selected elements.

Moreover, when importing a DXF format file, display of individual model layers may be set on the *Layers* tab (see the drawing below).



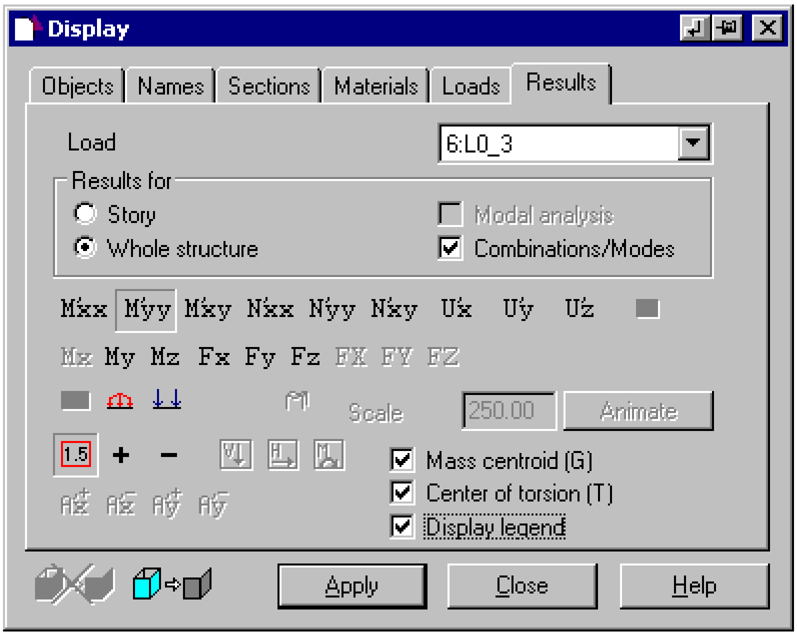

The tabs: *Objects*, *Names*, *Sections* and *Materials* are identical. Below are described the remaining two tabs.

The *Loads* tab




The above dialog box makes possible selection of the loads to be presented on the screen; the following loads may be chosen for presentation: concentrated, linear or planar. Moreover, for loads there is a possibility to show their values and names on the screen. As an additional load filter a load nature can also be used; all load natures or a selected nature available in the current regulations may be displayed.

The *Results* tab




The options on the tab above enable presentation of calculation results or load distribution in a 3D view, depending on the selected calculation method.

If there are walls in a structure model that was calculated using one of the advanced methods, it is possible to display values of the reduced forces V, H, M for walls in points defined in the ***Reduced forces*** dialog box.

For continuous footings the forces FX, FY and FZ (reactions for walls calculated in ***ROBOT***) may be presented – see Default options.

*NOTE:* *Results may be presented in the 3D view, once the computational view is chosen (the* View / Computational View *option in the menu or the F8 key shortcut). For the options in the* **Display** *dialog box to be accessible, the 3D view, with the computational view switched on, must be active (after switching to a different view the options in the dialog box are inaccessible; therefore, after e.g. selecting objects, the 3D view should be activated).*

Objects defined in a structure are presented as contours and lines; the following types of presentations are possible in the program:

- Maps – display of maps or load distribution (the trapezoidal and triangular method) on planar elements

- Load distribution – is accessible only for the trapezoidal and triangular method and the simplified FEM method, for the remaining methods there are maps (Note: in the case of the simplified FEM method maps are available only for slabs)

- Diagrams – display of distribution of loads (concentrated forces) onto columns or walls.

The above dialog box contains the *Results for combinations* option. If this option is switched off, the load case list includes, apart from simple cases, only extreme combinations (ULS+, ULS-, SLS+, SLS-, ALS+, ALS-). After activating the *Results for combinations* option the list of cases also comprises all components of code combinations and combinations defined manually which are marked as active in the ***Combinations*** dialog box.
If the *Display legend* option is switched on, then a 3D structure view will present, apart from diagrams or maps, a scale for the displayed quantity.
Switching on the options *Mass centroid (G)* or *Center of torsion (T)* allows presenting positions of points (with their coordinates) at which the mass centroid G or the center of torsion T are located; the options are available in the dialog box in the computational views (2D and 3D) after performing seismic calculations (simplified or advanced). The 2D view presents the mass centroids and the centers of torsion with coordinates for the active story, while the 3D view displays the mass centroids and the centers of torsion for all stories (at the midpoint of the story height).
The 2D view also enables presenting distribution of loads: concentrated forces and linear loads (loads parallel to the Z axis may be presented in the 2D view as well as in an axonometric view) – see also: Presentation of results in a 2D view.

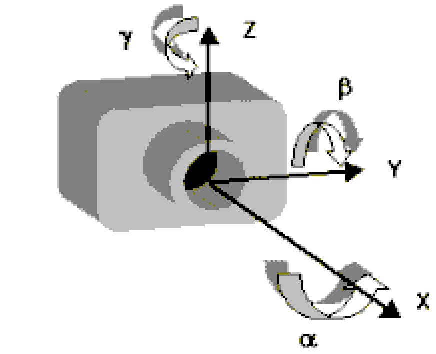
*NOTE:* *After the calculation is performed in the* **ROBOT Millennium** *or* **ROBOT Kernel** *program, the option that enables presenting results for a selected story is inactive.*

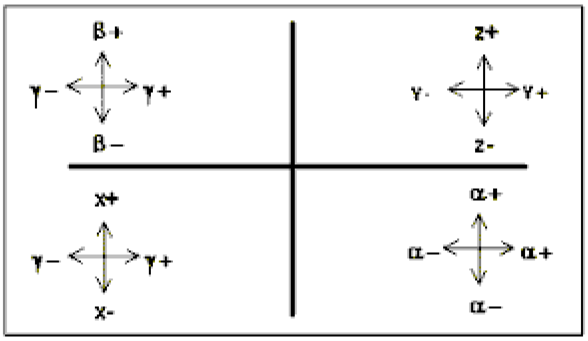
In the ***CBS Pro*** program there is also the *Display Selected Elements* option provided. It is available from the context menu in a selected structure view. After switching on this option only selected elements are visible on the screen (after switching on the *Display Selected Elements* option objects are presented without being highlighted).
*NOTE:* *In a view in which the* Display Selected Elements *option is switched on, selection of objects is not possible (the mouse cursor changes – the cursor is presented in gray then* *).*

# 5.13.3D Perspective Display

## 5.13.1. Perspective View Support

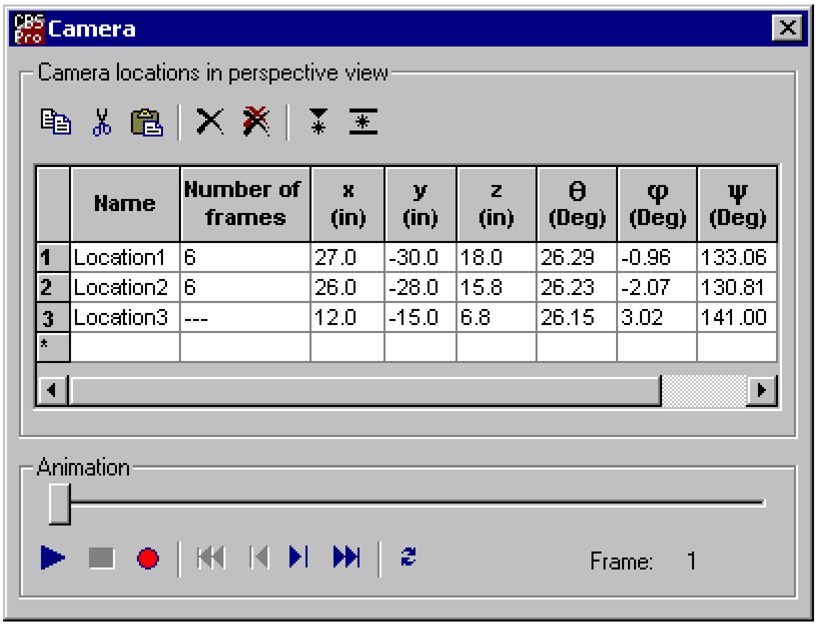
The perspective view is available from the menu after selecting the *View / 3D View / Perspective Projection* option or from the 3D View toolbar in every viewer with a 3D structure view.
When using the perspective projection, the camera movement options linked with the mouse movements on screen refer to the camera's local system (see drawing below).



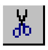




### 5.13.2. Camera Location

The *Camera location* option is accessible from the context menu in any 3D viewer. It enables preparing and recording animations. Once it is activated, the following dialog box appears on screen:




The top toolbar in the dialog box above contains options used for setting a camera location:

-  copies a camera location
-  cuts a camera location
-  pastes a camera location
-  deletes one camera location
-  deletes all camera locations
-  adds a new camera location
-  adds a camera location based on the current view.

The table presents settings of the current camera locations. Individual table columns show the following information: first column specifies a name of camera location, the next one determines a number of frames between the current location and the next one, the subsequent six columns defines a camera location. A camera location in the above dialog box is presented in the global system.

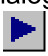
The bottom part of the dialog box (the *Animation* field) includes options used to steer animation watching and recording .

### 5.13.3. Watching a Presentation

Once the *Play* option is selected in the *Animation* field in the Camera dialog box, the program plays an animation in an active 3D viewer.

*NOTE:* *If this is a 2D viewer or a 3D viewer without perspective projection switched on, that is active, an animation is not played.*

For animation support there are the following options provided in the lower part of the ***Camera*** dialog box:

-  plays / pauses an animation
-  stops an animation (returns to the first frame)
-  records an animation
-  jumps to the previous location
-  jumps to the previous frame
-  jumps to the next frame
-  jumps to the next location
-  plays an animation continuously.

### 5.13.4. Recording a Presentation

Once the *Record* option is selected in the *Animation* field in the Camera dialog box, the program activates animation saving to file. To record a presentation, the user should:

- indicate name of the file where to an animation will be compressed; NOTE: animation files may come in large sizes, particularly for not compressed formats or low-compression ones, thus, take note if the adequately large amount of free space is left on disk
- select video compression (a codec type); all the video compression codecs installed in the system are accessible in the dialog box.

*NOTE:* *Some of the codecs installed in the system may not enable saving a file in a chosen format, they only allow reading (decoding). In such a case saving in a selected format is impossible and when trying to do so, an appropriate message appears.*

*NOTE:* *For some codecs there are compression options available in the dialog box for selection of compression. They depend on a codec type, a description of their application is provided on the manufacturer's Internet sites.*

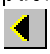
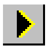
After selecting compression parameters, an animation is prepared for saving. Pressing the *Play* option saves the animation to a file. Stopping the animation ends recording. While saving all the options of presentation watching are available.

# 6. DEFINITION OF A STRUCTURE MODEL

## 6.1. Story

The story is one of the basic terms in definition of a structure model. By standard, a structure is defined story by story (in the XY plane). For the user to be able to control a defined structure model, it is recommended to work in two windows with the XY projection set in one window and 3D projection (for a whole building or for the current story) set in the other.
A number of the current story may be changed by means of the *Story / Current story* option (it is also presented in the *Story* toolbar as a selection list). In each window the user may set a different story number (e.g. while working using two windows, the user may set the first floor view in one of them and the second floor view in the other).


The program also enables definition of a new story (moving to a not-existing story) using the options provided in the *Story* toolbar. After pressing one of the following icons:

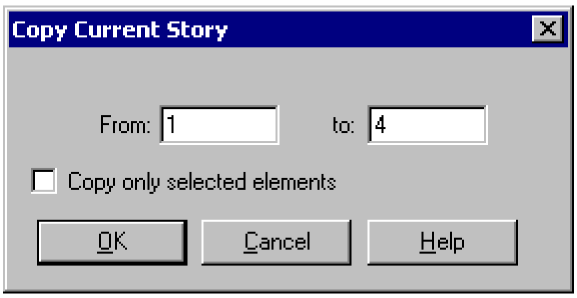
-  Previous story
-  Next story,

the message: "Do you want to add a new story?" will appear on screen. If the 'yes' answer is given, the program will create a new story and will move to the new story; if the **No** button is pressed, the story will not be created (the program will not move to a different story).

## 6.2. How to Copy a Story

To copy the current story e.g. five times, follow the steps below:

- activate the 2D viewer
- select the menu command *Story / Copy*
- in the ***Copy Current Story*** dialog box, in the *From:* edit field enter the value 1, in the *to:* field enter 4



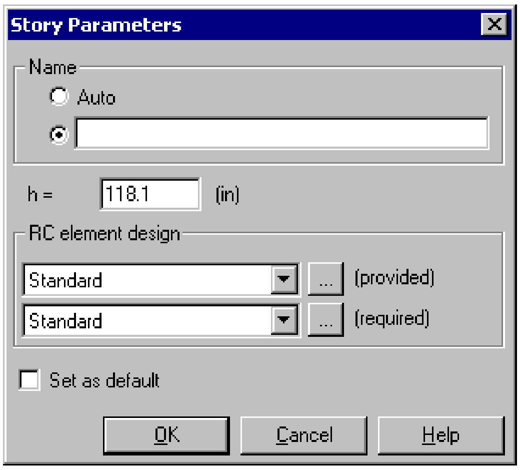

- press the **OK** button; a building consisting of five identical stories has been generated.

# 6.3. Story Parameters

Story parameters may be changed in the dialog box available from:

- the menu, by selecting the *Story / Parameters* option
- the toolbar, after pressing the  *Story parameters* icon.

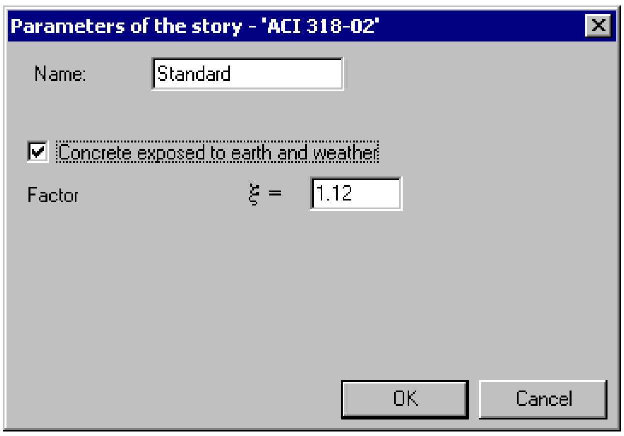
For each story a name may be defined; the user may also use the option of automatic name ascription to stories (the *Auto* option switched on). Apart from that, each story may be ascribed height independently.




Options in the lower part of the dialog box enable selection of a story parameter set (saved in a file available on the list or lists) used in calculation of the required and provided reinforcements of structure elements; to define new story parameters for a selected RC code, press the (**...**) button provided to the right of the selection list.

*NOTE:* *If the same code is used for calculation of the required reinforcement and the provided reinforcement, then only one selection list will be available in the above dialog box. If different codes are used to calculate the required reinforcement and the provided reinforcement, the dialog box will include two selection lists.*

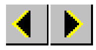
A list of story parameters that can be set in the ***CBS Pro*** program is conditioned by a selected code of the required and provided reinforcement design. Pressing the (**...**) button located to the right of the selection list opens the ***Parameters of the story*** dialog box for a selected RC code. In the dialog box below the user may assign a name to a defined parameter set and change story parameters to the ones appropriate for the RC element design code.



After switching on the *Save as default* option, the current parameters are saved as a default parameter set.

## 6.4. How to Modify Story Parameters

To modify the name and the height of the third story, do as follows:

- select story no. 3 by pressing the buttons *Previous story / Next story*  or by selecting story no. 3 from the *Story no.* drop-down list in the toolbar 

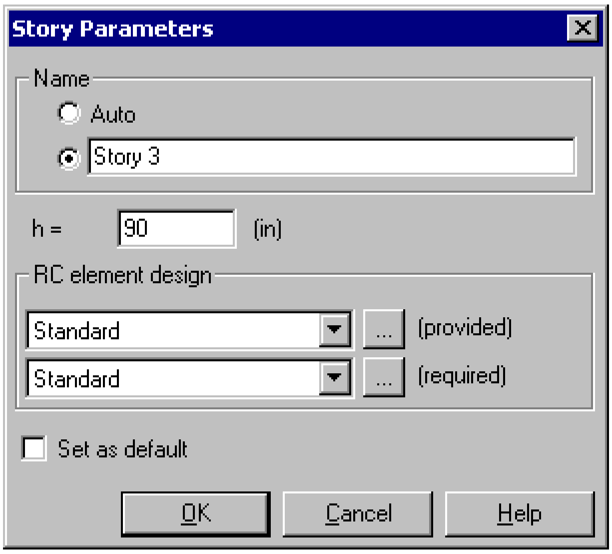

- select the menu command *Story / Parameters* or press the *Story Parameters*  icon

- select the blank field and enter there *Story 3*

- in the *h=* edit field enter 2.5

- leave the parameters of RC element design unchanged

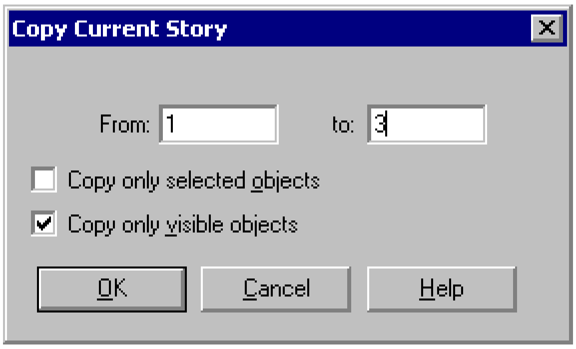
- press the **OK** button.



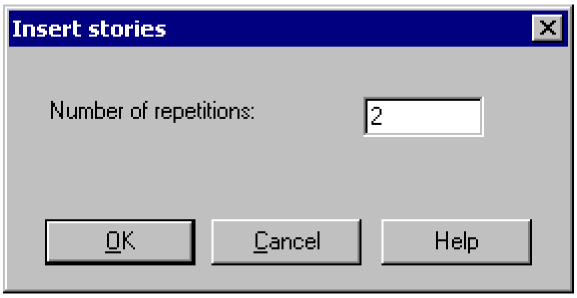

## 6.5. Edit Options (Story)

The following edit options are available for the story (they are located in the *Story* menu):

- *Copy* – the option enables copying the current story; the user should define story numbers (from, to) where to a given story should be copied; the user may copy all the elements from a given story as well as only selected elements (the *Story / Copy / Copy only selected objects* option); if the *Copy only visible objects* option is switched on, then only visible objects and loads will be copied; after copying a story, the last-defined story is set as a current one




- *Insert* - the option enables inserting any number of empty stories between the existing stories

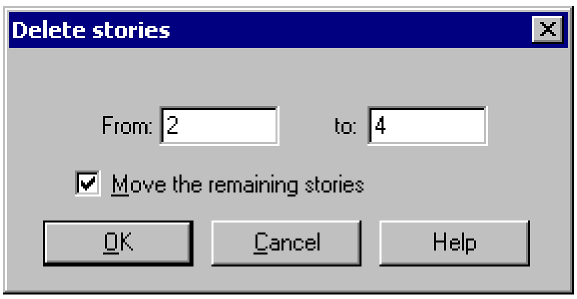



The program also enables definition of a new story (moving to a not-existing story) using the options provided in the *Story* toolbar. After pressing one of the following icons:

- Previous story
- Next story,

the message: "Do you want to add a new story?" will appear on screen. If the 'yes' answer is given, the program will create a new story and will move to the new story; if the **No** button is pressed, the story will not be created (the program will not move to a different story).

- *Delete* – the option enables deleting defined stories; if there are any stories above the stories currently deleted, the program may leave the empty stories or move the upper stories down (the *Story / Delete / Move the remaining stories* option).




## 6.6. Dimension Lines

In the program definition of dimension lines for a generated structure is enabled. There are options available which allow selecting a dimension line type (in the *Dimension lines* menu or icons in the Dimension lines toolbar).

– dimension lines parallel to axes of the global coordinate system

– dimension lines parallel to the dimensioned object

– automatic generation of dimension lines; all the displayed points (architectural points or axis intersections) are being used

– dimension lines with arrowheads

– dimension lines in the form of reference marks.

Moreover, the following options are available in the program:

– enables display of architectural points; points are displayed for the currently-selected elements.
The option is used during automatic generation of dimension lines; if this option is turned on, then while defining dimension lines automatically, the architectural points displayed are used in generation of a dimension line.

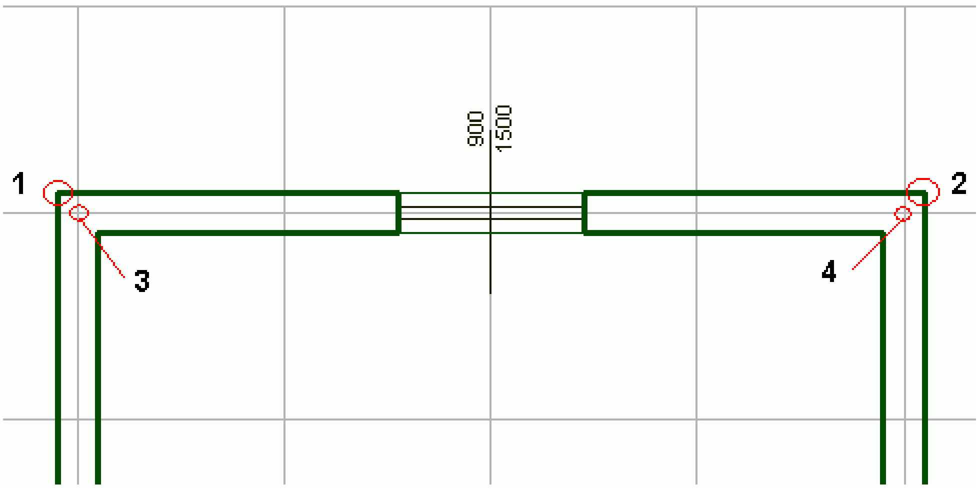
– enables display of axis points; points are displayed for the currently-selected elements.
The option is used during automatic generation of dimension lines; if this option is turned on, then while defining dimension lines automatically, the displayed points of axis intersections are used in generation of a dimension line.

If the *Drag* option is turned on, it enables placing successive structure dimensions on the same dimension line.

All the points positioned on a dimension line are available in the ***Object properties*** dialog box on the *Position* tab; this dialog box enables modification of points (adding, deleting). If a point used in definition of a dimension line is simultaneously a point belonging to any other object (e.g. wall end), then translation of such an object will result in automatic update of points on a dimension line.

## 6.7. How to Define Dimension Lines

To add a dimension line to outer contour edge for the wall (see the drawing below), do as follows:



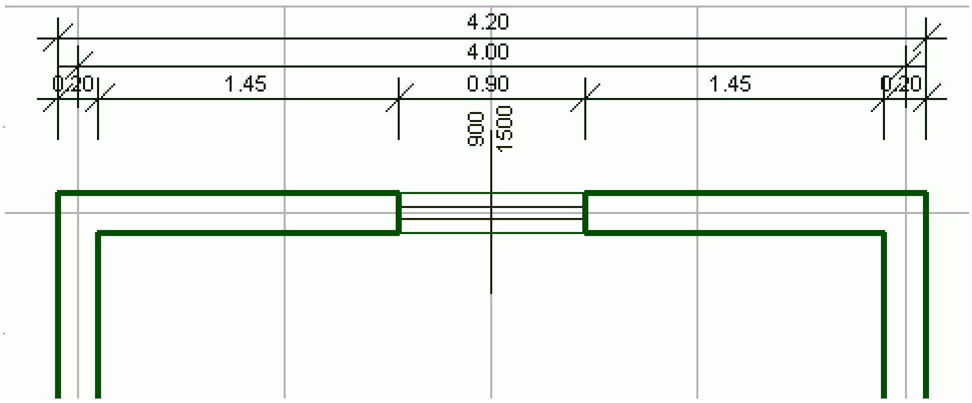

- select the command Dimension Lines / Orthogonal or press the Orthogonal icon
- activate the option Snap cursor to object architectural points by pressing the icon (other snap options may be active at the same time)
- in the 2D viewer click with the left mouse button on point 1, and next, on point 2 (see the drawing above)
- using the mouse determine the location of the dimension line and press the left mouse button.

To add a dimension line to axis for the wall, follow the instructions below:

- select the command Dimension Lines / Orthogonal or press the Orthogonal icon
- activate the option Snap cursor to intersections of object axes by pressing the icon (other snap options may be active at the same time)
- in the 2D viewer click with the left mouse button on point 1, and next, on point 2 (see the drawing above)
- using the mouse determine the location of the dimension line and press the left mouse button.

To add automatically dimension lines to outer contour edge for a wall with openings, do as follows:

- switch on the option *Display architectural points* by selecting the menu command *Dimension Lines / Display Architectural Points* or by pressing the icon on the *Dimension Lines* toolbar
- press the right mouse button and choose the *Select* option from the context menu
- select the wall being dimensioned (the wall becomes highlighted)
- press the *Automatic* icon
- in the 2D viewer, using the mouse determine the location of the dimension line and press the left mouse button; the created dimension lines are presented in the drawing below.




## 6.8. Objects Available in the Program

There are the following object types available in the ***CBS Pro*** program:

- beams – are defined by means of two points; beam section may be rectangular or T-shaped (for RC sections) or it may be any section from the steel section database (for steel sections)

  if the option for beam definition is selected (particularly important when defining steel members), then the following options are available in the menu:

  *Horizontal Beam* – if this option is chosen, then a beam is defined as horizontal on a selected story

*Inclined Beam - Up* - if this option is chosen, then a beam is defined as an inclined one between the stories (the beginning of a beam is positioned on the lower story, whereas the end - on the upper story)

*Inclined Beam - Down* - if this option is chosen, then a beam is defined as an inclined one between the stories (the beginning of a beam is positioned on the upper story, whereas the end - on the lower story)

*Lintel* - if this option is chosen, then a beam is defined as a lintel (the beam above window and door openings) – see the description of the lintel definition and assumptions adopted in the lintel calculations; the user should indicate windows or doors above which a lintel should be defined – a default lintel length is defined in the ***Default Options*** dialog box

*Ground beam* - if this option is chosen, then a beam is defined as a ground beam (a horizontal beam supporting objects located on the same story, such as columns, walls, partition walls, and transferring loads from a structure to foundations); a ground beam is faced (aligned) to the bottom of the current story - it may be presented in the architectural view or in the ***Object Properties*** dialog box.

In the current program version reactions from ground beams are applied axially to centers of foundations (the eccentricity from the edge to the center of the spread footing is ignored).

*NOTE:* *In advanced calculations of a structure, an equivalent load resulting from a partition wall or finishing layers of a wall positioned on a ground beam is not applied to the beam and the equivalent load should be applied manually (there is a similar case with a continuous footing if a wall bottom is not supported on a slab).*

- columns - are defined by means of one point – the other point (column end) is generated automatically based on the height of a given story; column section may be rectangular, T-shaped or round – see also: How to define a column

- spread footings - are defined by means of one point; spread footing section may be rectangular (the *rectangular footing* object in the section database) or trapezoidal (the *trapezoidal footing* object in the section database) – see also: How to define a spread footing

- continuous footings - are defined by means of two points; continuous footing section may be rectangular or T-shaped

- walls - are defined by means of two points; the user may assign thickness to them and define layered material (see the description of layered material in the material database); the program offers the possibility of assigning a shape other than rectangle to the wall (e.g. gable wall); to do that the user should open the ***Object properties*** dialog box (the *Position* tab), add a new point to the contour and modify its coordinates appropriately – see also: How to define a wall

- slabs - are defined by means of a broken line (polyline); the user may assign thickness to them and define layered material (see the description of layered material in the material database); by default they are positioned on the top level of the current story – see also: How to define a slab

the *Objects* menu also contains the *Add Specially* option - it enables quick definition of slabs in typical situations; if the user selects this option and then clicks with the left mouse button on the area limited by walls, then the program generates a slab on this contour

- raft foundations - are defined in the same manner as slabs – the difference is that by default they are positioned on the bottom level of the current story

- partition walls - are defined in the same manner as walls – the difference is that the *Structural element* option is switched off automatically; if a structure is exported to the ***ROBOT Millennium*** program, it results in modeling this type of objects as linear load on a slab

  The following rules apply to non-structural elements in the program:
  - continuous footings / spread footings are generated under unsupported columns / non-structural walls
  - the weight of disregarded non-structural elements is not included in the structure self-weight
  - non-structural elements are disregarded during design of structure elements.

- windows – are defined by means of one point which determines a position of the window center; window section may be rectangular (the *rectangular opening* object type in the section database); NOTE: windows may not exist independently, they are always associated with the *wall* object type; it means that a window position is always defined in the local coordinate system of a wall and that a window is automatically deleted when the wall to which it belongs is deleted

- doors - are defined in the same manner as windows - the difference is that they are automatically positioned on the bottom level of the current story

- floor cuts/openings – are defined by means of a polyline (for any shape) or by means of one point (for a rectangular shape); floor cuts/openings may not exist independently, they are always associated with the *slab* object type; it means that a position of a floor cut/opening is always defined in the local coordinate system of a slab and that a floor cut/opening is automatically deleted when the slab to which it belongs is deleted

- stairs – are defined by means of a quadrangle (it should be remembered that edges at both ends of a flight of stairs must be parallel); in the current program version only straight stairs are available (the *single-flight stairs* object type in the section database); parameters of a flight of stairs (number of steps, their width and height) may be defined by the user or determined automatically by the program (to calculate the number of steps the following approximate formula is applied: 2*h + s = 63, where h – step height, s – step width); height of a flight of stairs is recognized automatically based on the slab positions with which the stairs have been connected

  NOTE: The self-weight and loads of stairs are taken into account during calculations of a structure by means of the advanced method; stairs are treated as a slab, while all stair loads as slab loads
  NOTE: In the case of structure calculations by means of the triangular and trapezoidal method or the simplified method, the self-weight and loads of stairs will be taken into account if supports are defined for flights of stairs.

- rooms - are defined by means of polylines; their shape may be defined by the user or determined automatically by the program based on the analysis of geometry of a given story (detection of closed contours); rooms are auxiliary (non-structural) objects used for room description in the architectural presentation and when preparing summary tables of finishing materials

- texts - are defined by means of a point; this point indicates the bottom left corner of the entered text; at the defined point the program inserts a default text, whose parameters are determined in the dialog box opened after selecting the option: *Edit / Default values / Texts*; a text or its parameters may be changed in the ***Object properties*** dialog box.

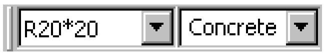
After defining an object of a given type, the program expects – by default – definition of the next object of the same type. Pressing the **Esc** button switches on the selection mode.

## 6.9. How to Define a Beam (Horizontal and Inclined)

To define a horizontal, concrete beam of section 20*20 cm, follow the steps presented below:

- select the menu command *Objects / Horizontal Beam* or drop down the menu at the *Beam*  icon and select the *Horizontal Beam* option

- on the toolbar choose the R20*20 section (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Concrete material




- in the 2D viewer click with the left mouse button on the beginning point of the beam, and next on the end point of the beam

- the beam will be automatically generated on the upper level of a given story.

To define an inclined beam of timber – pine section of dimensions 20*20 cm, follow the instructions below:

- select the menu command *Objects / Inclined Beam - Down* or drop down the menu at the *Beam*  icon and select the *Inclined Beam - Down* option

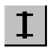
- on the toolbar choose the R20*20 section (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Pine material


- in the 2D viewer click with the left mouse button on the beginning point of the beam (upper, outermost level of the story), and next on the end point of the beam (lower, outermost level of the story)

- the beam will be generated as an inclined one, with the beginning and end points positioned respectively, on the levels of the highest point and the lowest point of a given story.

## 6.10.How to Define a Column

To define a concrete column of section 20*20 cm, follow the steps below:

- select the menu command *Objects / Column* or press the *Column*  icon

- on the toolbar choose the R20*20 section (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Concrete material


- in the 2D viewer click with the left mouse button on the insertion point of the column

- the height of the inserted column is by default equal to the height of the story.

# 6.11. How to Define a Wall

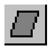
To define a 20cm-thick wall of solid brick, follow the steps listed below:

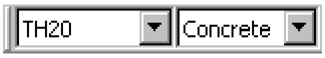
- select the menu command *Objects / Wall* or press the *Wall*  icon
- on the toolbar choose the TH20 section (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Solid brick material


- in the 2D viewer click with the left mouse button on the beginning point of the wall, and next, on the end point of the wall
- the height of the inserted wall is by default equal to the height of the story.

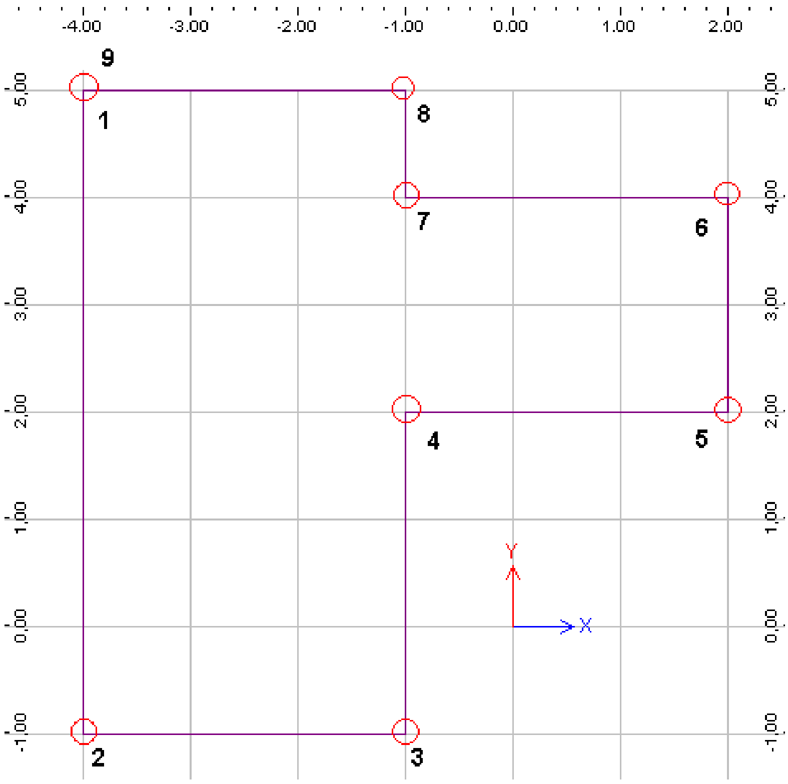
# 6.12. How to Define a Slab

To define a 20cm-thick concrete slab, follow the below-listed steps:

- select the menu command *Objects / Slab* or press the *Slab*  icon
- on the toolbar choose the TH20 section (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Concrete material




- in the 2D viewer click with the left mouse button on points 1 to 9 one by one (see the drawing below), given that the coordinates of point 9 are the same as for point 1 (thus the contour determining the shape of a slab closes).

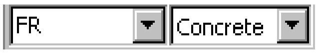


NOTE: *If a closed contour is already generated, a slab can also be defined in a very simple way using the Add Specially option; selection of this option and the left-mouse-button click on the closed contour, limited with walls, results in automatic generation of* a slab on this contour.

## 6.13.How to Define a Spread Footing

To define a (concrete) spread footing, follow the instructions below:

- select the menu command *Objects / Spread Footing* or press the *Spread Footing* icon
- on the toolbar choose the section FR – rectangular footing (if this section is not displayed on the list, add it using the *Default Sections* command*)* and the Concrete material




- in the 2D viewer click with the left mouse button on the insertion point of the spread footing
- the inserted spread footing is positioned by default, on the level of the lowest point of a given story.

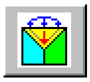
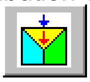
## 6.14.Description of the Lintel Definition and Assumptions Adopted in the Lintel Calculations

A lintel is an object of the beam type, yet it does not participate in general load distribution and is calculated independently.

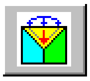
The following assumptions concerning the lintel definition have been adopted in the program:

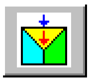
- a lintel is always associated with an opening (window, door); after deleting the opening, the lintel is deleted as well
- the lintel is created automatically at the height equal to the half the height of the section above the upper edge of the opening
- the ***Object Properties*** dialog box enables modification of the x and y coordinates (when extending the lintel above the neighboring windows)
- the lintel length considered in calculations equals the width of the opening + half the lengths of overhangs on both sides
- in the architectural view the lintel is presented with the entire overhang
- in the ***Properties / Calculation options*** dialog box the lintel overhang is adopted as a support width
- if the lintel is defined above two openings, then the width of the middle support equals the distance between the openings (NOTE: if openings touch, then they are treated as one opening).

Calculations of the lintel in the ***CBS Pro*** program are performed adopting the following assumptions:

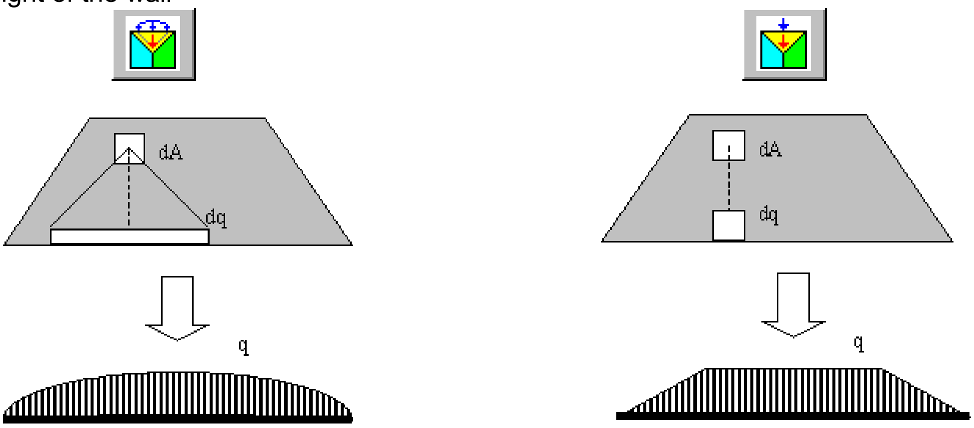
- distribution of loads onto the lintel considers the manner of load transfer; the icons  and  are used for that purpose – they are provided in the ***Calculation Options*** dialog box for distribution according to the triangular and trapezoidal method

- the weight of the wall above the opening should be assumed as a lintel load – in the form of:

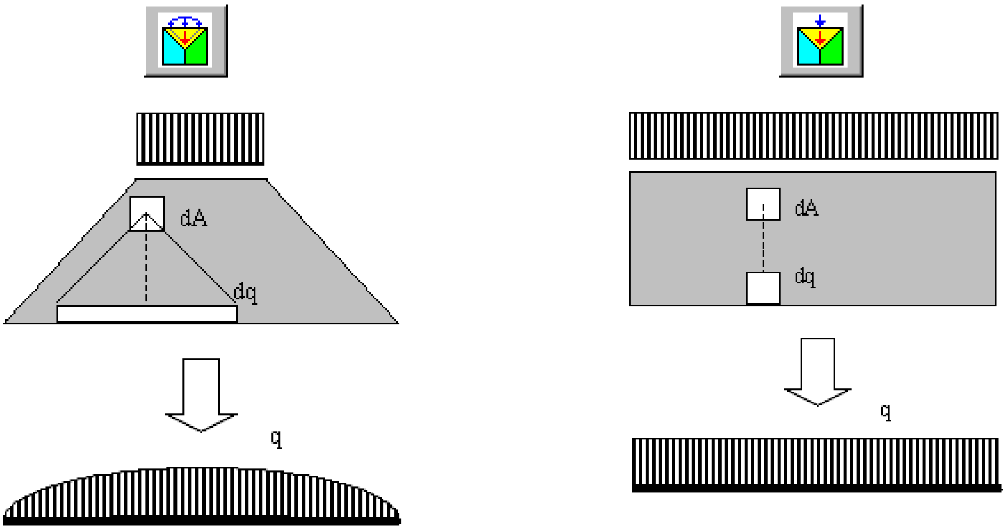
  - a triangle or a trapezoid formed by drawing straight lines at an angle of 45 degrees from the opening edge if the  icon is selected; if the trapezoid is formed, it is also necessary to add loads from the upper surface of the wall from the length of the trapezoid side

  - a rectangle above the lintel beam if the  icon is selected.

Schemes of the load distribution are presented in the drawings below.
1. Self-weight of the wall




2. Loads from the floor



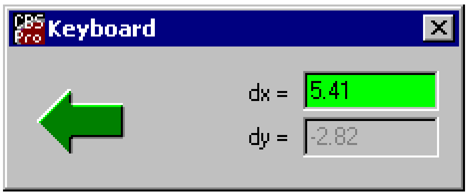

The lintel design in the ***CBS Pro*** program is carried out adopting the following assumptions:

- width of the outermost supports equals the lintel overhang

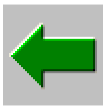
- width of intermediate supports equals the distances between openings (if openings touch, they are treated then as one opening and the width equals zero).

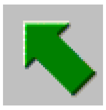
# 6.15.Entering of Data from the Keyboard

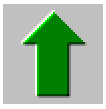
The ***CBS Pro*** program enables definition of linear and planar elements through the keyboard. If the keyboard is used in definition of beam, continuous footing, slab, raft foundation or wall, the program displays the dialog box allowing definition of values from the keyboard.

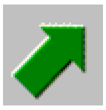


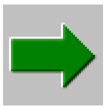

The coordinates dx and dy, that can be defined in the above dialog box, are the relative coordinates referred to the last-defined point. The icons located in the left part of the dialog box indicate direction of value increment:

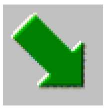
 – to the left

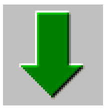
 – to the left-up

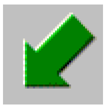
 – up

 – to the right-up

 – to the right

 – to the right-down

 – down

 – to the left-down.

The direction can be determined by pressing the arrows on the keyboard; pressing two arrows simultaneously allows definition of both increments: dx and dy. By pressing an arrow, as well as by pressing the **Enter** key, the value entered is accepted.

# 6.16.Object Properties

The ***Object properties*** dialog box plays the following role:

- is used to present and modify data concerning all objects already defined; this data is displayed for one or for several currently selected elements

- options on the *Results* tab are used to present internal forces, load distribution (for the trapezoidal and triangular distribution) as well as results of design for elements of a structure model defined in the ***CBS Pro*** program

- options on the *Calculation options* tab are used to determine support conditions of individual structure elements as well as to define releases on beams, to change a support type or to change the local system for slabs in the case of structures calculated by the ***ROBOT*** program

- may be used to define a new object; in this mode the program changes the name of the dialog box to: ***Add object – object type***.

The ***Object properties*** dialog box opens after:

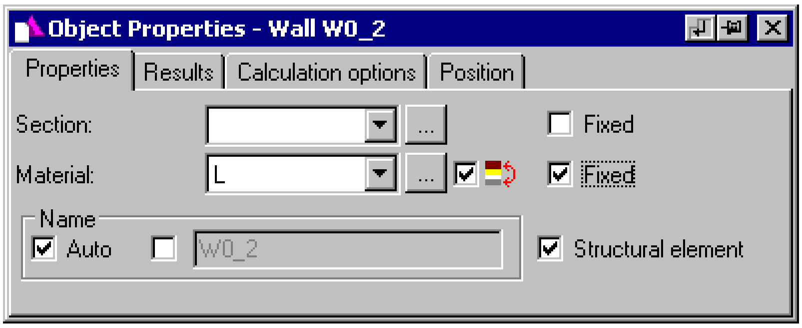
- selecting the menu command*: Edit / Properties*

- pressing the  icon

- pressing the **Alt** + **Enter** key combination

- selecting the *Properties* command from the context menu.

The tabs: *Results* and *Calculation options* are displayed for the following objects:
- *Results* tab – all objects except spread footings
- *Calculation options* tab – for beams, slabs and raft foundations.




The following information is presented on the *Properties* tab:

- *Section* – a section from the section database ascribed to a selected object; there is a list of all the defined sections of a given type available which enables changing a section; after pressing the (**…**) button located to the right of the field containing a section name, the ***Section database*** dialog box is opened

- *Material* - a material from the material database ascribed to a selected object; there is a list of all the defined materials available which enables changing a material; after pressing the (**…**) button located to the right of the field containing a material name, the ***Material database*** dialog box is opened

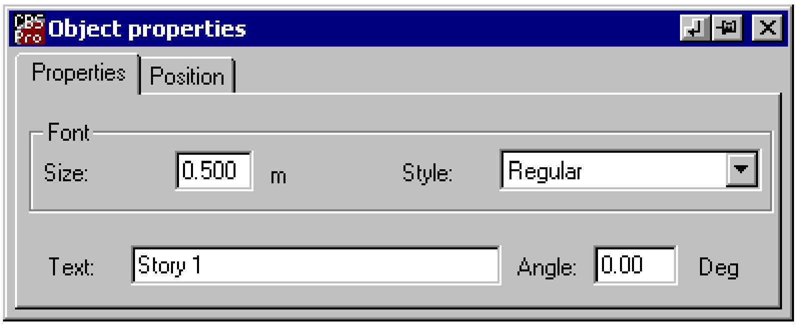
- In the case of a layered material, next to the button used for adding a new material there is an additional option for reversing the order of layers

- *Name* – a name ascribed to a selected object; a name may be ascribed automatically based on the default parameters set in the dialog box opened after selecting the option: *Edit / Default values / Names* (the *Auto* option is switched on then) or ascribed manually by the user (it should be entered to the edit field after switching off the *Auto* option)

- *Structural element* – the option which enables the user to determine if a given object is to be treated as a calculation element; if for a given object the option is switched on, then an equivalent of the object model (bar, support or panel) will be generated in the ***ROBOT Millennium*** program; if the option is switched off, then - in the ***ROBOT Millennium*** program, the object will be modeled as a load (e.g. for a partition wall, its weight will be changed to a linear load applied to the corresponding panel).

For the options: *Section* and *Material,* the *Fixed* option is also available. If this option is turned on, it 'freezes' a given parameter: e.g. if the user selects 2 beams of identical sections and for one of them the *Section fixed* option is turned on, then replacement of the section with a different one will not affect this particular beam.

The above data is accessible for all the elements except for texts; for texts the *Properties* tab looks as shown in the figure below.




The following options are available on this tab:

- *Size* – font size which is defined in structure units; it means that the text size is correlated with the current zoom
- *Style* – font style; the allowable styles include: regular, italic, bold and bold italic
- *Text* – entered text
- *Angle* – angle at which the text will be displayed; for horizontal texts the inclination angle equals 0 degrees.

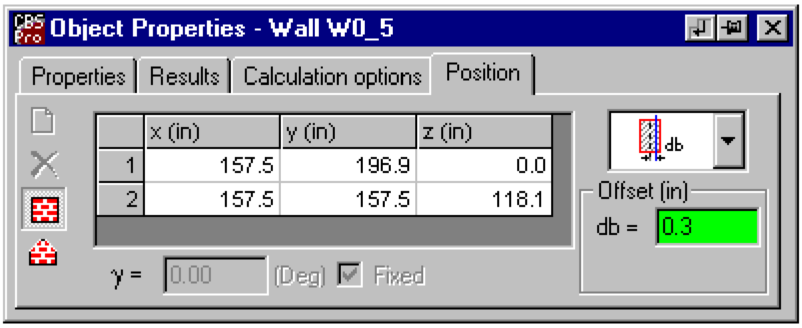
In order to obtain a required text on the screen, first a text should be defined by means of the option: *Edit / Objects / Text* (a default text with parameters set in the dialog box opened after selecting the option: *Edit / Default values / Texts* will be displayed on the screen) and afterwards it should be modified in the ***Object properties*** dialog box.

The *Position* tab shown in the figure below presents the following information:

- coordinates of the successive snap points of a given object (e.g. the beginning and end point for beams, columns, continuous footings, coordinates of the successive apices for slab or opening of any shape); in the case of slabs and openings the user may change the number of apices by means of the options: *New row* and *Delete row*
- $\gamma$ *(gamma angle)* – an angle by which a given object is rotated with respect to the local x axis; this option is active in the case of columns, spread footings, beams as well as continuous footings.

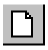
For the $\gamma$ *(gamma angle)* option there is the *Fixed* option is available. If this option is turned on, it 'freezes' a given parameter (just as in the case of the *Section* and *Material* options described above).

*NOTE:* *When defining a gamma angle for one object it is possible to use a value of the inclination angle of another object. To do it, the user should position the cursor in the field for definition of a gamma angle of an object whose section should be rotated, next highlight an object whose inclination angle should be read and after the angle value is displayed, indicate with the cursor the highlighted object or accept it with the* ***Enter*** *key.*




NOTE: *It is possible to change dimensions of the* **Object Properties** *dialog box on the* Position *tab.*

The top left corner of the dialog box holds the following options:

-  *New Row* – enables adding a new row to the table which describes positions of an object's characteristic points, and entering there coordinates of an object's point

-  *Delete Row* – enables deleting a selected row from the table describing positions of an object's characteristic points.

Comments on the option which enables adding/deleting a row to/from the table:

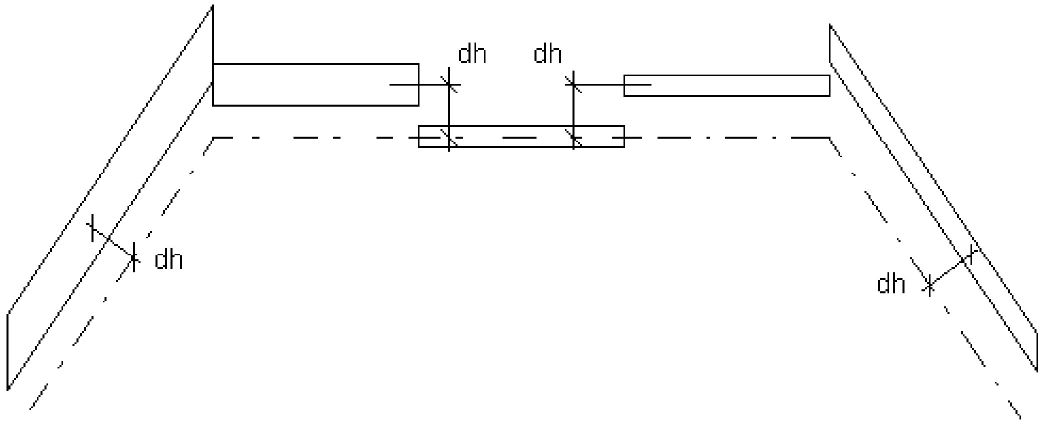
- for slabs – the option of row deletion stops to be active when there are only 3 rows (3 nodes of a slab) left on the list

- for walls: the options *New row* and *Delete row* are available when the *Detailed Wall Presentation* icon is switched on; additionally, the option of row deletion stops to be active when there are only 3 rows (3 nodes of a wall) left on the list.

At present, it is possible to modify a slab based on the wall with the simplified wall presentation turned on. The presentation of the trapezoidal gable wall should not be changed from *Detailed* to *Simplified*, since such a change will permanently modify the wall to a rectangular one (i.e. the nodes outside the rectangle contour will be lost).
The right-hand part of the dialog box holds a selection list which (depending on an object type) enables the user to choose horizontal translations of objects, and additionally, vertical translations of beams; the facing operation may refer to the following objects:
- facing a wall to walls
- facing a beam to walls
- facing a column to walls and walls to columns
- facing a wall to continuous footings and a continuous footing to walls.

NOTE: *The last-applied translation is set as a default one for a given type of object.*

NOTE: *When defining a translation by a given value dh, horizontal slabs and beams are translated vertically (with respect to the Z axis) by the value dh, whereas inclined slabs and beams are translated vertically in such a way so that the translation distance measured in the local object system equals dh (see the drawing below).*




The following positions of translations are possible:
*STANDARD*:
- facing to the left / right
- top / center / bottom (for beams and slabs)
*NON-STANDARD:*
- translation by the value db (horizontally) and dh (vertically - for beams and slabs).

Facing has effect on the position of continuous footings under walls and spread footings under columns; if continuous footings and spread footings are defined in a structure model, then the position of continuous footings and spread footings changes automatically if the facing is modified.

The *Position* tab also allows modifying positions of loads not assigned to an object (by changing coordinates of points where a load is applied).

In case of walls, the program allows for two modes of presentation:

- *Simplified* (refers to rectangular walls) – a wall is presented by means of two points (bottom left corner and top right corner)

- *Detailed* (refers to any type of walls) – all points of a wall contour are being presented.

To define e.g. a gable wall, the user should choose the detailed presentation, add point at the required location (it is added in the midpoint of the distance between the point currently highlighted and the previous point) and in the end, change the value of its z coordinate.
For rectangle-shaped openings, the program enables defining the reference point at the following points (see the drawing below):

- opening center

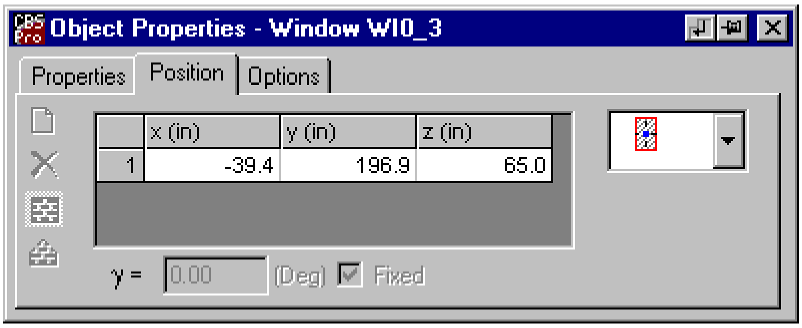
- bottom left corner of an opening

- bottom right corner of an opening

- top left corner of an opening

- top right corner of an opening

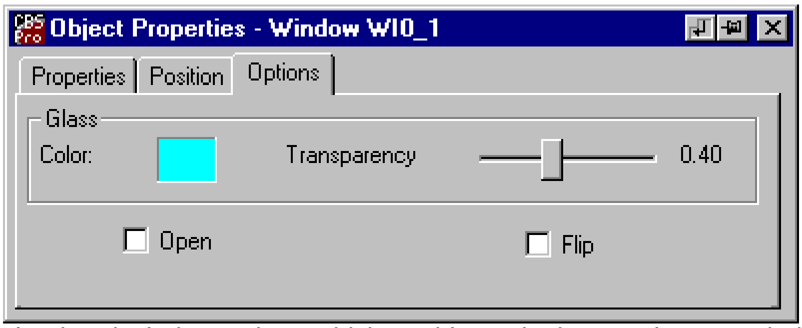
- centers of individual sides of an opening.




Coordinates of a new object may be defined manually or by using the combination of manual and graphical definition. For example, to define a 3 m-long wall attached at the point being the end a different wall and perpendicular to this wall, the user should:

- open the ***Add object – Wall*** dialog box on the *Position* tab
- translate the coordinate system to a required point using the option *Coordinate system – according to object* (orthogonally with respect to the existing wall)
- define first point with the mouse by clicking on the origin of the coordinate system
- define second point manually entering the coordinates x = 0.0, y = 3.0.

When the ***Object properties*** dialog box presents properties of a window, the additional *Options* tab is available.




The above dialog box includes options which enable assigning a color to a window pane and selecting a degree of transparency for presentation in the 3D view (the options refer only to the architectural view).
The lower part of the dialog box holds the options *Open* and *Flip*; they allow modifying the way a window is presented in the 3D architectural view:

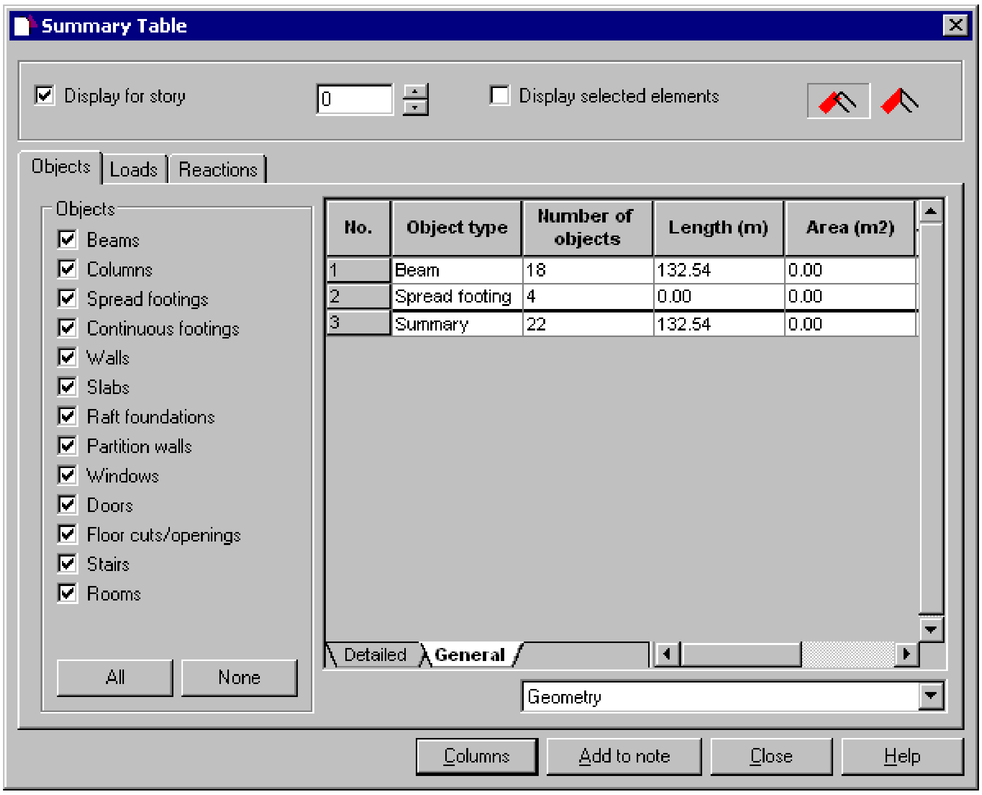
- *Open* - if this option is switched on, individual window panes are displayed as opened (casement window), pivoted open, slid open (double-hung window) according to the definition of the window in the database (for a given window type)
- *Flip* - if this option is switched on, an opened window (after activating the *Open* option) will be reversed by 180 degrees (a window opened to the inside will be presented as opened to the outside).

## 6.17.Summary Table

The *Summary Table* option enables presentation of data for a structure defined by the user. The option is available after:

- pressing the *Summary Table* icon
- selecting the menu command: *Edit / Summary Table*.

The dialog box shown in the drawing below appears on screen then.



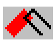

NOTE: *In the current program version it is possible to change the vertical and horizontal dimensions of the* **Summary Table** *dialog box. thus the length and height of the dialog box can be adapted to the user's needs.*

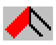
The upper part of the dialog box holds the following two options:

- *Display for story* – if this option is switched off, the table presents all objects from all stories; if this option is switched on, then the program makes accessible another edit field in which a story number may be specified (the table will be showing then only the objects from a selected story)

- *Display selected elements* - if this option is switched on, the table presents only the objects selected in the graphic viewer.

The upper part of the dialog box also holds two icons:

-  - if this icon is activated, then a volume and mass of objects will be calculated, and displayed in the table, based on structural dimensions (axial dimensions of objects)

-  - if this icon is activated, then a volume and mass of objects will be calculated, and displayed in the table, based on architectural dimensions (real dimensions useful in a cost estimate).

NOTE: *In a summary table it is possible to select all elements by clicking on the top left corner of the first table column; individual objects can also be selected by means of the summary table.*

The program allows a possibility to delete objects and loads directly from the summary table. Selected objects can be deleted independently of that if selection of one story or of a whole structure is activated. An object or a load is deleted on pressing the **Del** button.
If objects and loads are selected and the user wants to remove them through the summary table, then objects are deleted on the *Objects* tab, while loads - on the *Loads* tab.

The above dialog box consists of three tabs:
- Objects
- Loads
- Reactions.

At the bottom of the dialog box there are the following buttons apart from the standard ones **Close** and **Help**:

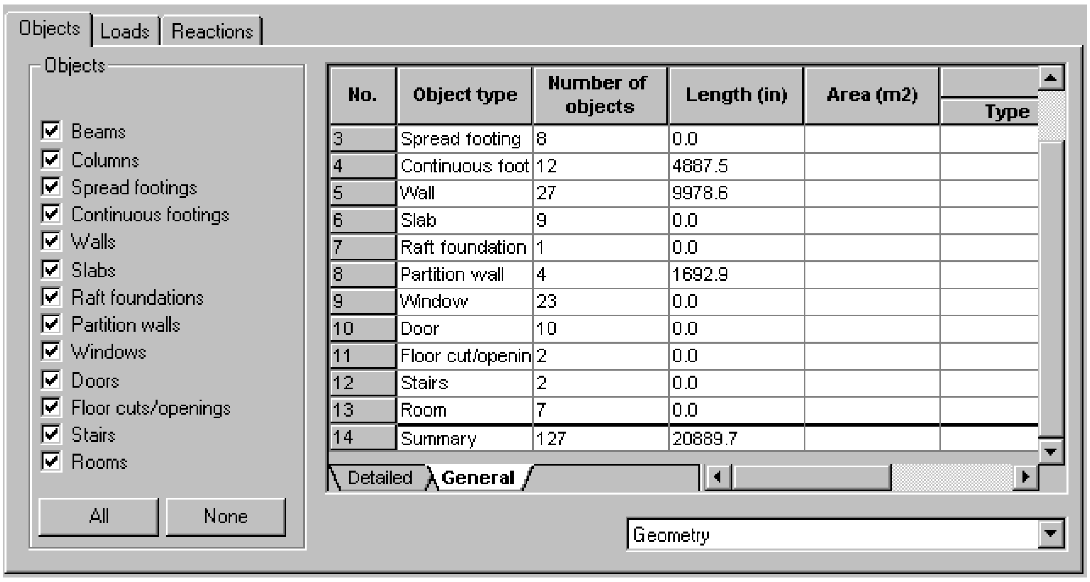
- **Columns** – pressing this button opens the ***Columns*** dialog box which shows the list of available and currently selected columns of the summary table of objects or loads (√ symbol appears next to the selected options); the right part of the ***Columns*** dialog box holds the two buttons ▲ and ▼ for modification of a column position in the summary table

- **Add to note** - after pressing this button, the table located in the ***Summary table*** dialog box will be added to the list of note components (see the description of the Printout composition dialog box); a name of the table saved in the ***Printout composition*** dialog box is specified in the ***Add to Note*** dialog box that appears on pressing the **Add to note** button

- **Print** – the option allowing printout of the table.

# 6.18.Summary Table - Objects

The option enables presentation of data for structure objects (beams, columns, slabs, walls, etc.). The data has been grouped on two tabs:
- *Detailed*
- *General*.




The table on the *Detailed* tab presents a list of individual objects (structure elements, finishing) in compliance with the selection made in the left part of the dialog box (√ symbol beside the object name indicates that the selected object type will be presented in the summary table, no symbol - the selected object type will not be presented in the table).
Pressing the **All** button results in selection of all the object types to be presented in the table; after pressing the **None** button, no object will be presented in the table.

The columns presented in the table:

- may be the columns defined in several templates available on the selection list at the bottom of the dialog box (under the table)

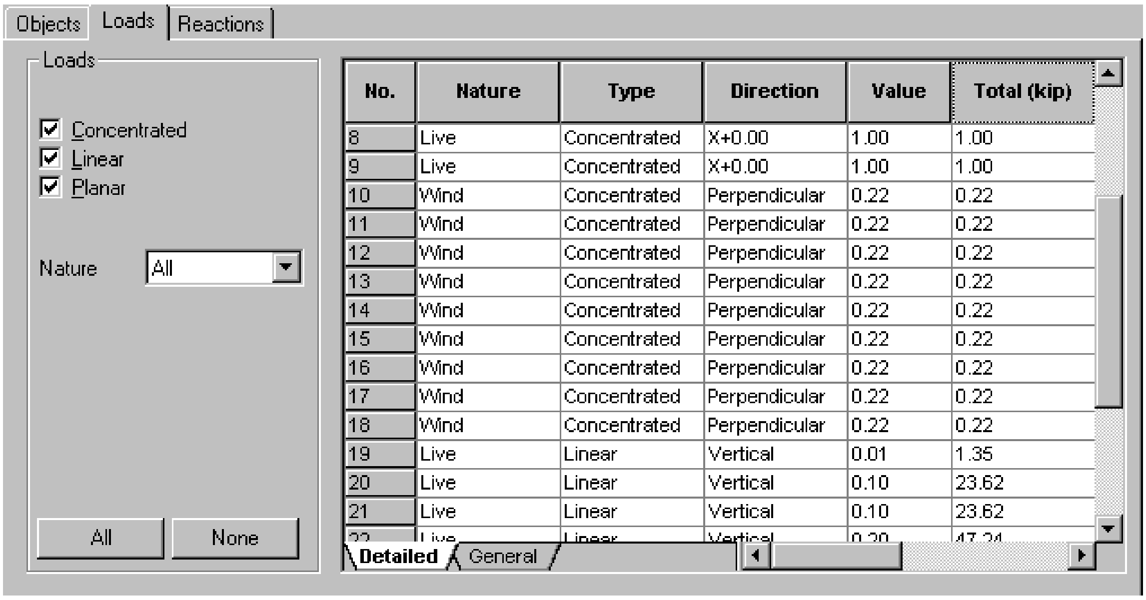
- may be freely selected and rearranged by the user after pressing the ***Columns*** button.

At present, the following templates are available in the program: *Geometry, Cost – RC structure, Cost – timber structure, Cost – steel structure*.
The *Geometry* template comprises a set of columns needed for basic data about objects in a model, whereas the remaining templates include columns with information about the amount and the cost of individual structure elements.
The table on the *General* tab presents individual types of objects used in a model. Similarly as on the *Objects - Details* tab, this tab also enables selection of a column set by applying a defined template (the list of available templates is analogous as that on the *Objects - Details* tab) or by pressing the ***Columns*** button.
This tab may present the data as follows: structure self-weight, weight of partition walls and dead load resulting from the finishing (it is not presented on the tabs for loads, since thanks to selection, this type of information may be easily shown on the tabs for objects).
The tabs *Detailed* and *General* (for the *Cost - RC structure* profile) include additionally the *Use* column. Steel use values are presented in the following units:

- spread footings, continuous footings - (%) of the cross section
- beams, columns – in the units chosen in the ***General Parameters / Optimization criterion*** dialog box.

## 6.19.Summary Table - Loads

The option enables presentation of data concerning the structure loads. The data has been grouped on the following two tabs:
- *Detailed*
- *General*.




The table on the *Detailed* tab may present the information as follows:

- load type: concentrated, linear or planar (√ symbol next to the name of a load type indicates that the selected type will be presented in the summary table, no symbol – a selected type will not be presented in the table); pressing the **All** button results in selection of all the load types to be presented in the table; after pressing the **None** button, no load type will be presented in the table
- load nature (also a subnature, in case there are several subnatures of a load) – the list includes all the load natures (subnatures) available in the regulations
- load direction:

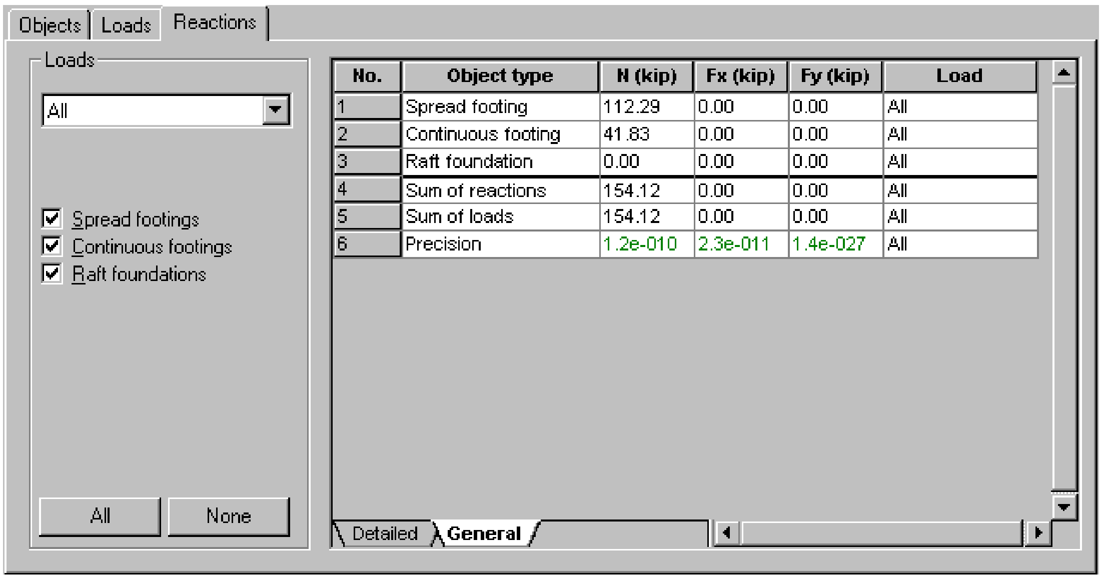
- vertical
- X - 135 (horizontal, forming an angle of -135° with the X axis)
  sign convention of an angle: positive values for the counter-clockwise direction
- perpendicular to an object to which it is applied (e.g. for a wind load)
- projection (e.g. for a snow load)

- load value
- total – total value of the applied load
- story – a story on which the load is applied
- name – load name
- object / name and object / type – name and type of an object to which the load has been applied (if the load is not assigned to an object, then a blank cell is presented in the table).

The table on the *General* tab comprises the columns as follows: direction, number and total. The main purpose of this table is to present loads for selected natures, types of loads and stories, therefore, all the selected loads are displayed for 3 directions:
- Z direction (vertical)
- directions X and Y (horizontal).
By default the table consists of the columns: *Direction* and *Total*, but it is also possible to turn on display of the *Number* column; this column shows the number of all the components for a given direction (not the number of loads as objects).

## 6.20.Summary Table - Reactions

The option enables presentation of reactions for loads applied to structure elements. The data has been grouped on the following two tabs:
- *Detailed*: presenting reaction forces in individual structure elements
- *General*: check if the reaction sum is congruous with the sum of all loads applied a structure.




The left-hand part of the dialog box holds a list of all defined subnatures of structure loads (cases) for which reaction values may be presented. Below are three options (*Spread footings, Continuous footings, Raft foundations*) whose activation means that reaction values will be presented for selected support types.
Pressing the **All** button activates all the support types (i.e. the options *Spread footings, Continuous footings, Raft foundations* will be switched on), while pressing the **None** button means that all the support types will be switched off.

In the right-hand part of the dialog box there is a table with reaction values:

- the *Detailed* tab shows rows with values of forces for individual supports

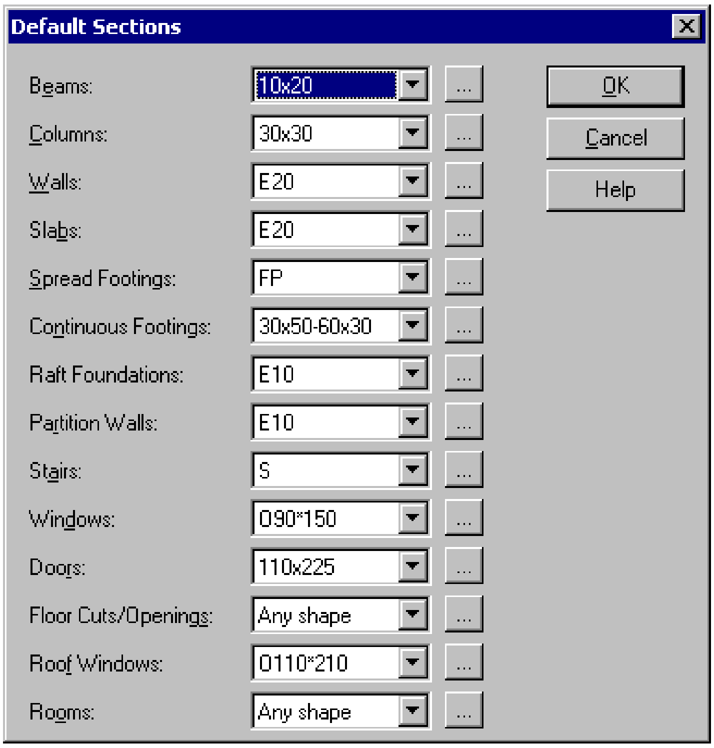
- the *General* tab shows rows with sums of reaction forces for individual support types (spread footings, continuous footings, raft foundations) which have been activated in the left-hand part of the dialog box, and the bottom line of the table displays the total sum of reactions; if all the support types are switched on in the left-hand part of the dialog box, then additionally, the table includes a row with a sum of loads and a value of the difference between the sum of loads and the sum of reactions (precision); if values of the sum of reactions and the sum of loads differ, a precision value is displayed in red, and in the ***Reports*** dialog box the message 'Imbalance of reactions and loads' is displayed.

# 6.21.Default Values

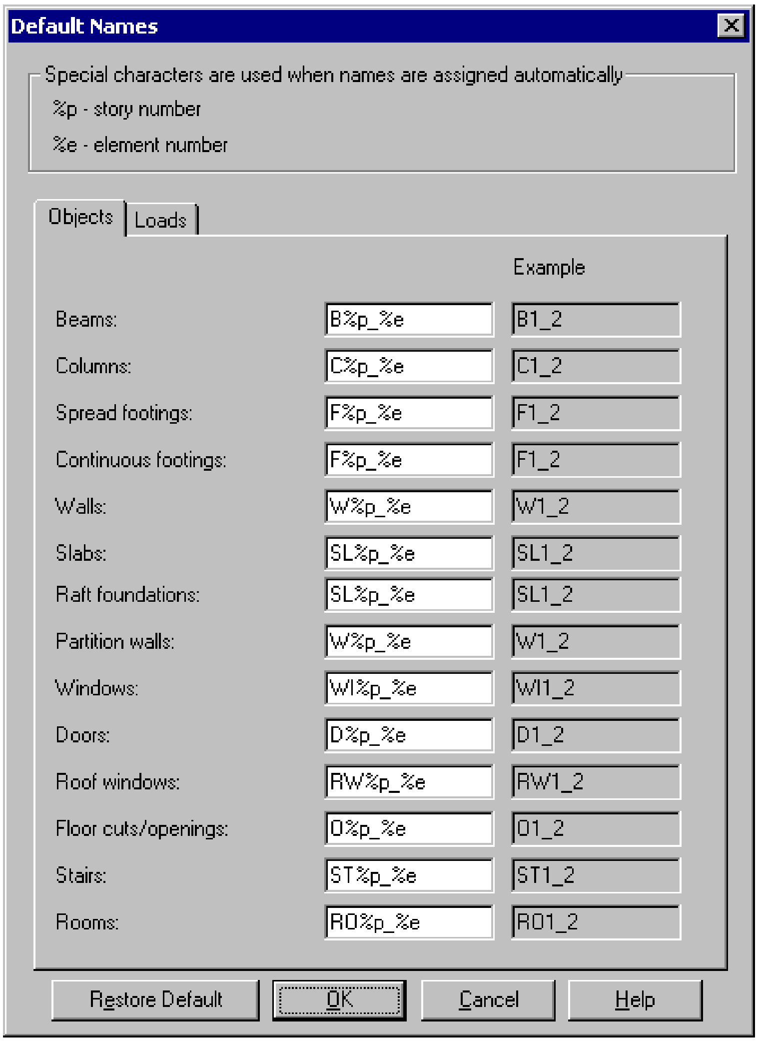
The option enables setting default parameters of sections, materials, names and texts. The option is available from:

- the menu, by selecting one of the options contained in *Edit / Default values*

- the toolbar after pressing the icons: *Default sections* and *Default materials*.




The ***Default sections*** dialog box allows setting a default section for each of the objects (for beams, columns, slabs, etc.). The selection list for each object type contains all the sections defined hitherto, which may be attributed to a given object (e.g. rectangular and T-shaped sections for beams). Pressing the (**...**) button located to the right of the selection lists opens the Section database dialog box, in which section database can be edited (i.e. change of parameters of the existing sections, adding a new section).
The ***Default materials*** dialog box enables setting a default material for each of the objects (for beams, columns, slabs, etc.). The selection list for each object type contains all the materials defined hitherto, which may be attributed to a given object. Pressing the (**...**) button located to the right of the selection lists opens the Material database dialog box, in which material database can be edited (i.e. change of parameters of the existing materials, adding a new material).




In the ***Default names*** dialog box on the *Objects* tab the user may set a default name for each of the objects (for beams, columns, slabs, etc.); options on the *Loads* tab enable determining default names for natures of loads defined in a structure (dead, live, wind, snow, etc.). While defining a name, special characters may be used, which can make automatic numbering easier:

- %p – if these characters are added, it means that in place of these characters a number of the current story will be inserted automatically
- %e - if these characters are added, it means that in place of these characters a number of the successive element from a given group (of beams, columns) on a given story will be inserted automatically

Pressing the ***Restore default*** button restores default values of names proposed in the ***CBS Pro*** program.

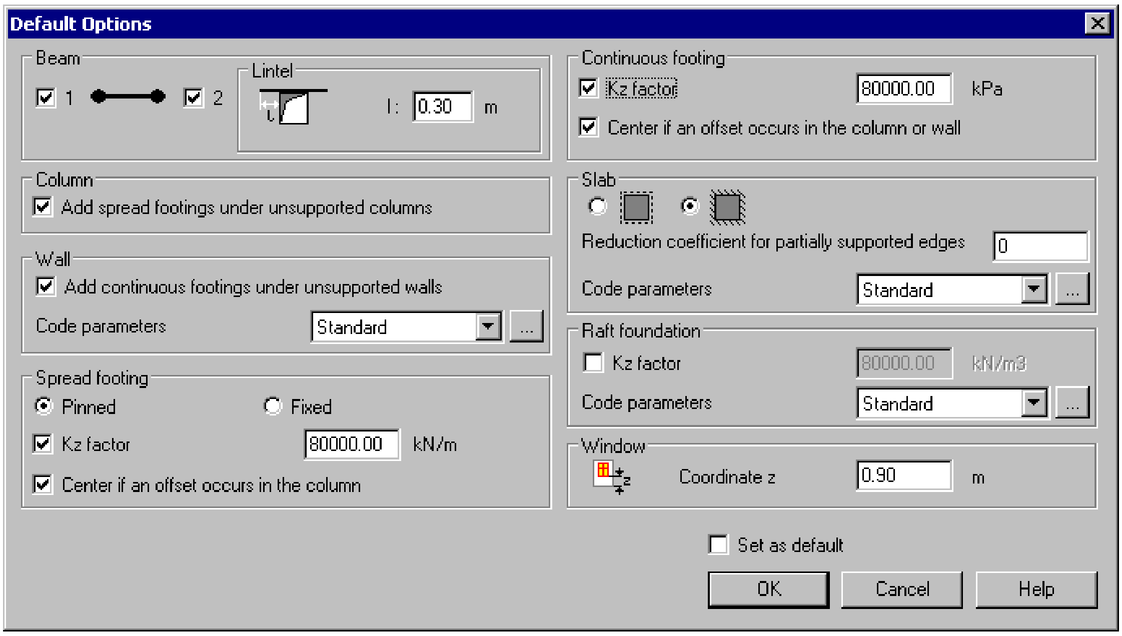
The ***Default text*** dialog box enables setting default parameters for texts.

- *Size* – font size
- *Style* – font style
- *Text* – default text entered while defining a text
- *Angle* – angle, at which the text will be displayed.

Parameters of the text entered may be changed in the Object properties dialog box.

# 6.22.Default Options

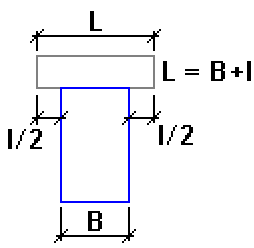
The option enables setting default parameters of the options used in the program while defining a structure and performing calculations. The option is accessible from the menu after selecting the *Edit / Default values / Options* option.




The following options may be selected in the above dialog box:

- *Beam* field: after switching on the option, pinned supports are defined on the beam ends (left, right)
- *Beam / Lintel* field: for definition of a default length of the lintel overhang l; the total length of the lintel L equals the sum of the opening (window, door) width B and the overhang length (L = B + 2*l/2) - see the drawing below

  see also: Description of the lintel definition and assumptions adopted in the lintel calculations




- *Column* field: after switching on the option *Add spread footings under unsupported columns,* spread footings are added to the structure in course of structure calculations

- *Wall* field: after switching on the option *Add continuous footings under unsupported walls,* continuous footings are added to the structure in course of structure calculations; the selection list contains defined types of reinforcement parameters and pressing the (**...**) button opens the ***Code parameters*** dialog box; the options provided in this dialog box depend on a selected RC design code

- Spread footing field:

  the options *Pinned* and *Fixed* determine the support type provided by a spread footing to the column; it may be modified in the ***Object properties*** dialog box (the *Calculation options* tab)
  it is also possible to define a value of the elastic factor of the soil Kz; it expresses the total reaction of the soil to the spread footing that would occur for the unit deformation of the soil
  if the *Center if an offset occurs in the column* option is switched on, then in the 2D and 3D architectural view, the axis of the spread footing will be positioned exactly on the column axis; when the option is switched off, the foundation position is not modified automatically if the facing operation is performed on columns

- *Slab* field: it is possible to select a default support of a slab (pinned or fixed support) along the entire length of the slab contour; the selection list contains defined types of reinforcement parameters and pressing the (**...**) button opens the ***Code parameters*** dialog box; the options provided in this dialog box depend on a selected RC design code

- *Continuous footing* and *Raft foundation fields:* it is possible to define a value of the elastic factor of the soil Kz; in the case of a continuous footing this value expresses the reaction of the soil to 1 m of the continuous footing that would occur for the unit deformation of the soil, whereas in the case of a slab – it expresses the reaction of the soil to 1 $m^2$ of the slab that would occur for the unit deformation of the soil;

  NOTE: calculations of continuous footings for which the Kz option is not switched on, differ depending on a selected calculation method:
  *1. simplified structure calculations*
  a wall is supported on the continuous footing

  *2. advanced structure calculations*
  for a wall supported on the continuous footing linear (pinned) supports are generated; this information is transferred to ***ROBOT Millennium*** or ***ROBOT Kernel*** (a continuous footing is loaded to ***ROBOT***, but it is disregarded during calculations); once calculations are performed, reactions calculated in ***ROBOT*** are transferred to ***CBS Pro***; these reactions are applied to the wall and the continuous footing and may be presented in ***CBS Pro*** (forces FX, FY and FZ - Result display)

  for raft foundations: the selection list contains defined types of reinforcement parameters and pressing the (**...**) button opens the ***Code parameters*** dialog box; the options provided in this dialog box depend on a selected RC design code
  for continuous footings: if the option *Center if an offset occurs in the column or wall* is switched on, then in the 2D and 3D architectural view, the axis of the spread footing / continuous footing will be positioned exactly on the axis of the column / wall; when the option is switched off, the foundation position is not modified automatically if the facing operation is performed on walls / columns.

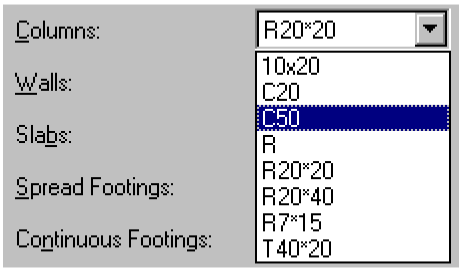
*NOTE:* *In the current program version, the facing operation does not cause changes in the position of objects in the structural and computational models; they are still positioned on their axes.*

In the lower part of the dialog box there is an option which allows determining (in the *Coordinate z* field) a default value of the distance between the lower edge of a window and the lower edge of a wall.

# 6.23.How to Define Default Values (Sections, Materials, Names)

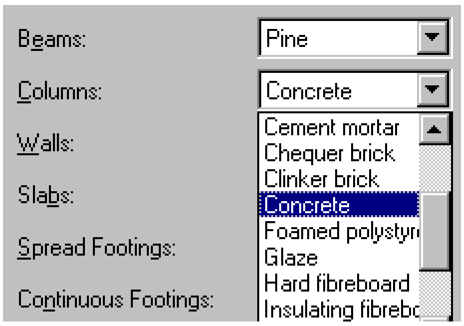
To set a default section of the column as a round one of 50-cm diameter, do as follows:

- select the menu command Edit / Default Values / Sections or press the Default Sections icon
- press the (**...**) button located to the right of the Columns drop-down list
- select the Round list (it becomes highlighted in blue) and press the **Add** icon
- in the d= edit field enter the value 50
- press the **OK** button
- on the drop-down list for columns in the **Default Sections** dialog box select the section C50 (as shown below) and press the **OK** button.




To set a default material of the column as the concrete, follow the steps below:

- select the menu command Edit / Default Values / Materials or press the Default Materials icon
- on the list for selection of the default material for columns choose Concrete (see the drawing below) and press the **OK** button.



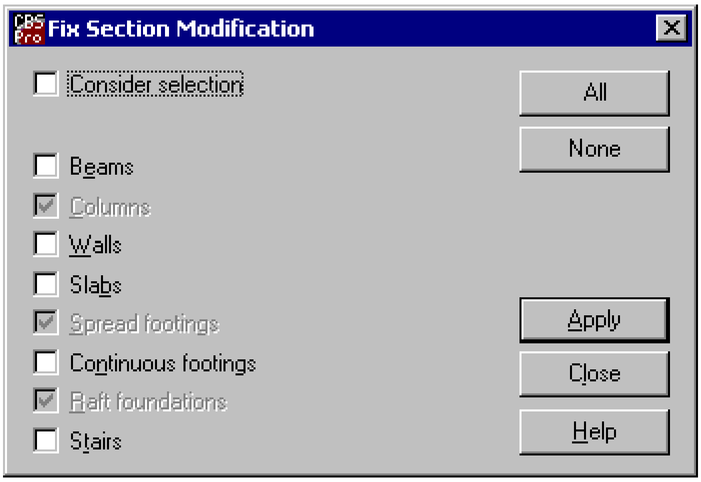

To change a default name of walls to 'Level(story no.)_Wall(wall no.)', follow the steps listed below:

- select the menu command Edit / Default Values / Names
- on the Objects tab in the Walls edit fields enter: Level %p_Wall %e (see the drawing below)


- press the **OK** button.

# 6.24.Fix Section Modification

The *Fix section modification* option is used for global definition of blocking (or unblocking) of sections of individual objects included in a structure. It is accessible by selecting the *Structure / Fix Section Modification* command from the menu.




In the upper part of the dialog box there is the *Consider selection* option; if this option is switched on, then all modifications (blocking/unblocking of sections) will refer only to the selected objects.
Below is the list of all objects which can be defined in a structure (beams, columns, walls, slabs, spread footings, continuous footings, raft foundations, stairs). If the option next to a structure element is switched on (the √ symbol appears), the section is blocked (this is a standard setting); if the structure element is switched off, it means its dimensions may be modified in course of the design of RC elements.

*NOTE:* *The objects, that are not included in a defined structure model, are inaccessible in the above dialog box.*

The right-hand part of the dialog box holds the following buttons (apart from the standard ones: **Apply, Close** and **Help**):

- **All** – pressing this button switches on all the objects available in a structure model in the ***Fix Section Modification*** dialog box

- **None** - pressing this button switches off all the objects available in the structure model in the ***Fix Section Modification*** dialog box.

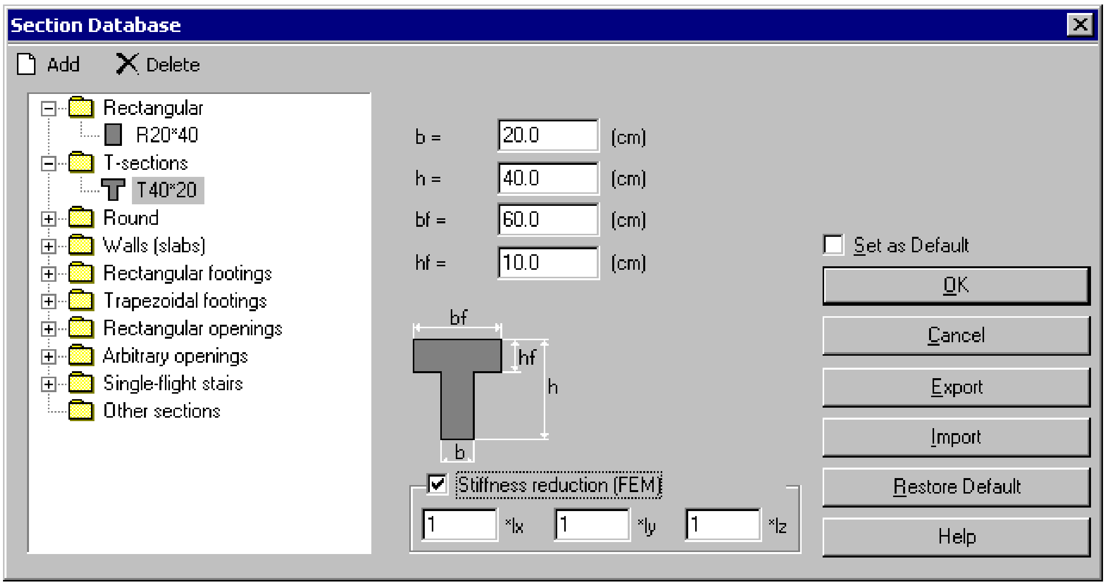
*NOTE:* *When using the* **Fix Section Modification** *option, take note of the status of the selection mode: select – current story or select – whole structure.*

# 6.25.Section Database

The program provides the possibility to define databases of available materials and sections. The option enabling section definition is accessible from:

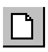
- the menu, by selecting the *Tools / Section database* command

- the toolbar, by pressing the  *Section database* icon.




The program allows definition of the following section types:

- rectangular – section height and width are specified
- T-section - section height and width as well as flange height and width are specified
- round – section diameter is specified
- wall (slab) – wall/slab thickness is specified; if a model is generated in the ***ROBOT Millennium*** program, the load resulting from a layered material that may be assigned to the wall, is automatically modeled there
- rectangular footings – length, width and height of a spread footing are specified
- trapezoidal footings - length, width and height of a spread footing as well as length, width and height of a pier are specified
- rectangular openings – opening height and width are specified; moreover, an opening type should be chosen (doors, windows); if a window is selected, the **Parameters** button is accessible in the dialog box; pressing this button opens the ***Window Parameters*** dialog box
- arbitrary openings - are used to define openings in walls as well as to define doors and windows. To define a new opening, follow the steps below:

  - select a floor cut defined in a slab or the entire slab
  - open the ***Section database*** dialog box and select the *Arbitrary openings* option in the tree
  - press the  *Add* icon; the opening will be added to the list.

  To make the section identification easier, its shape with dimensions of the rectangular contour is displayed. Moreover, it is necessary to choose the opening type (opening, door, window); when the window is chosen, the ***Parameters*** button is available in the dialog box, which when pressed, opens the ***Window Parameters*** dialog box
  Area and volume of openings / windows / doors created using the *Arbitrary openings* option may be presented in the summary table
- single-flight stairs – number of steps, step height and width and plate thickness are specified; first 3 parameters do not have to be defined (when the option at each of them is switched off) – then the program calculates the value of these parameters automatically while defining a certain element: e.g. if all the first 3 parameters are not determined, then

the program calculates the following: height of the entire flight of stairs based on the positions of the adjoining floor slabs, number of steps based on the relation: 2h + s = 63cm (where h – step height, s – step width) and next, values of the remaining parameters missing will be calculated

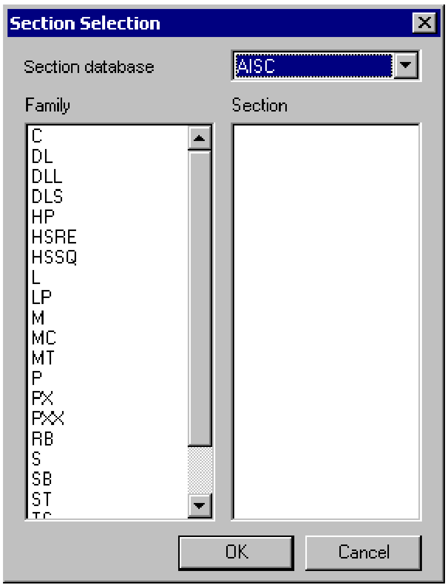
- other sections – during the program installation the user may install optionally steel section databases (these are the databases used in the ***ROBOT*** program); after selecting this option the following information appears in the right part of the dialog box: name of the folder which should contain steel section databases and the list of all available databases; the user may indicate which databases should be applied (limiting the number of databases may speed up the program operation in some situations, e.g. import of an IFC file).

*NOTE:* *There is a possibility to define the reduction of section stiffness due to cracking for rectangular, T-shaped, circular sections and wall (slab) sections. a value of this stiffness will be used in the exact calculations.*

To define a new section in the database, the user should:

- select (set the mouse cursor) a section type e.g. T-section
- press the *Add* icon
- define section dimensions (a section name will be proposed automatically based on the defined dimension values).

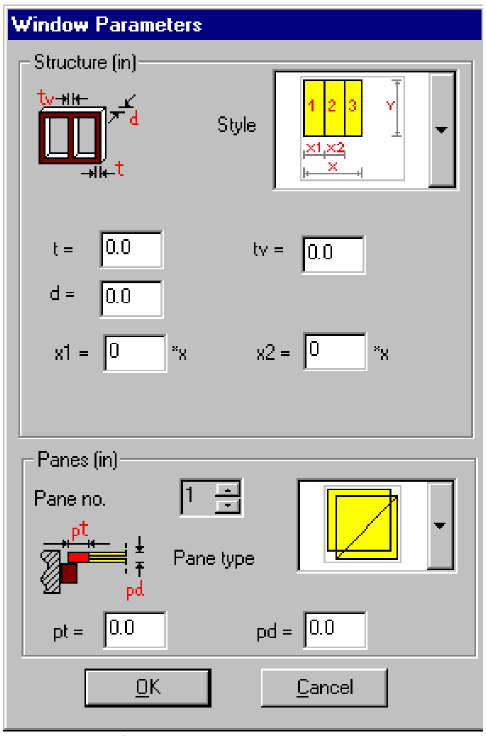
After selecting at least one section database and pressing the **Add** button, the additional dialog box ***Section Selection*** (see the drawing below) is displayed on the screen. In the dialog box below, the user should first choose a section family from the list (e.g. C) and next, indicate sections belonging to this family. The family and the list of selected sections are being added on pressing the **OK** button.




After switching on the *Set as Default* option, selected sections will be saved as a default set.
There are two buttons available in the ***Section database*** dialog box: **Import** and **Export**. Their pressing enables reading and saving, respectively, the section database in the program internal format. It allows e.g. transferring section databases from a program installed on other computer or copying a section database defined for a different language (by standard, there are separate section databases defined for different working languages).
Pressing of the **Restore Default** button restores default sections proposed in the ***CBS Pro*** program.

## 6.26.Window Parameters

The ***Window Parameters*** dialog box presented in the drawing below opens on pressing the **Parameters** button in the ***Section Database*** dialog box. The options in this dialog box allow defining division of a window into panes, determining thicknesses of sections of the frame and the sash as well as a glass pane type (casement, pivoted).



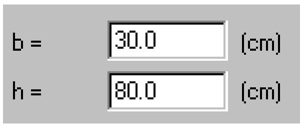

The following parameters may be defined in the above dialog box:
The *Structure* field
- selection of the window division into panes (the *Style* list)
- determining the values t, d, tv, th (thickness of the window frame section)
- determining the values x1, y1,... (relative coordinates of the division into panes)

The *Panes* field
- selection of a number of the currently-defined pane (the *Pane no.* list)
- selection of the pane type - casement, pivoted, casement/pivoted (the *Pane type* list)
- determining of the values pt, pd (width and thickness of the window sash (pane) section).

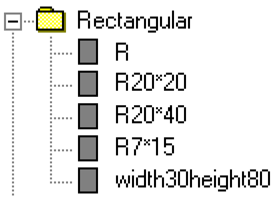
## 6.27.How to Add a New Section to the Section Database

To add a new rectangular section of dimensions 30*80 cm to the section database, follow the steps listed below:

- select the menu command *Tools / Section Database* or press the *Section Database*  icon on the toolbar
- select the *Rectangular* list (it will be highlighted in blue)
- press the **Add** icon
- in the *b=* edit field enter 30, in the *h=* edit field enter 80

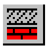


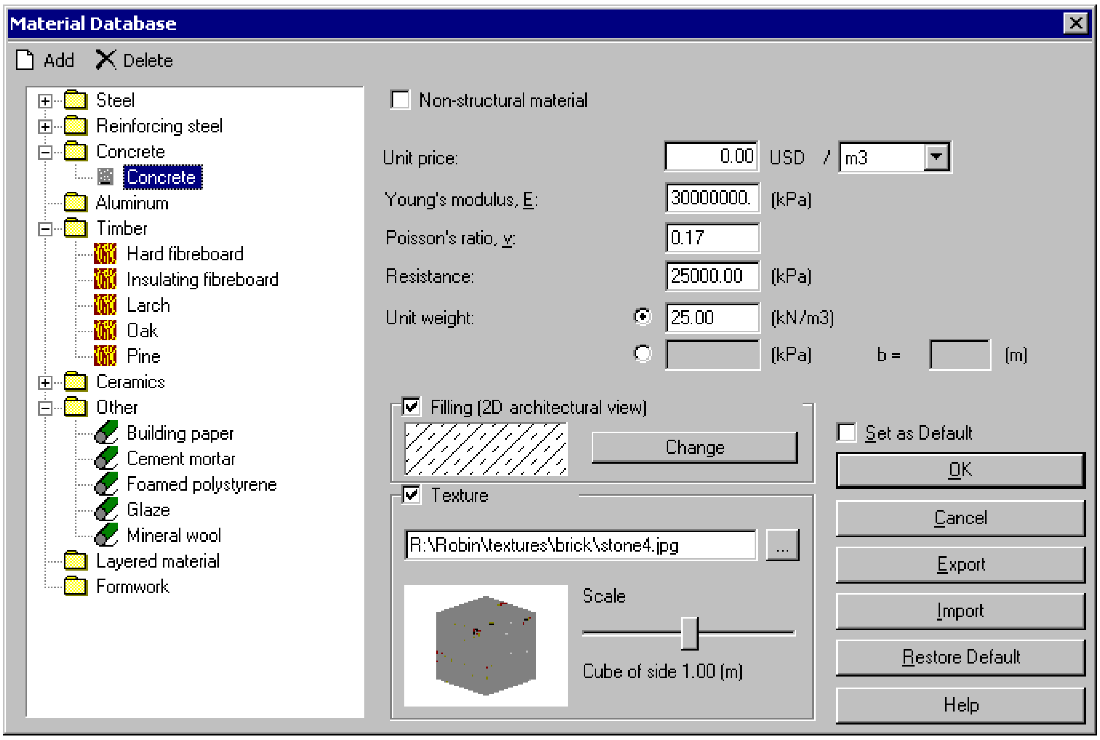

- to modify the section name - highlight the section R30*80 (it will be highlighted in blue) and click once with the left mouse button
- type: width30height80
- press the **Enter** button.




## 6.28.Material Database

The program provides the possibility to define databases of available materials and sections. The option enabling material definition is accessible from:

- the menu, by selecting the *Tools / Material database* command
- the toolbar, by pressing the  *Material database* icon.




The following material types are accessible in the program: steel, reinforcing steel, concrete, aluminum, timber, ceramics, etc. There is a possibility to define a layered material and unit prices for formworks of structure elements (beams, columns, slabs, walls, etc.); definition of the layered material involves determining the arrangement of materials which includes one structural material and any number of non-structural materials. If during definition of a layered material there will be 2 (or more) structural layers, then when accepting the definition, the user will have to choose one layer as a structural one. While defining the layered material the user

chooses materials from the list of materials previously defined in the material database and thickness of layers.
A structural layer of layered material is used in the strength analysis, whereas non-structural layers constitute loads of structural layers.

NOTE: *In the* **CBS Pro** *program (from version 2.2 on) layered materials are presented in drawings as a structural layer.*

*Presentation of walls made of a layered material in the structural and architectural views*
In ***CBS Pro*** layered materials are represented by a structural layer in the structural view, while in the architectural 2D and 3D view the program presents a wall of the actual thickness appropriately positioned (with respect to the axis of the structural layer).
If a layered material in a wall includes a structural layer, then the center of this layer is the axis of the wall in the structural view; if a layered material in a wall does not include a structural layer, then the center of the wall is the axis of the wall in the structural view.
If an offset with respect to the wall edge is defined, it is measured to the actual edge (an not to the structural layer of the wall).

To define a new material, the user should:

- select (set the mouse cursor) a material type e.g. concrete
- press the *Add* icon
- define values of material parameters.

NOTE: *The material database allows defining a unit weight in two ways:*
*- definition in the units: weight / volume*
*- definition in the units: weight / area*
*The latter possibility is available for objects like slab or wall. When using such definition of the material unit weight, it is necessary to determine an element thickness, since the program will not ascribe the section to it. This way of defining the unit weight is available for materials that have not been used in a structure, yet.*
*For an object made of the material with thus-defined unit weight the program does not display the section in the dialog boxes* **Object properties** *and* **Summary table***.*

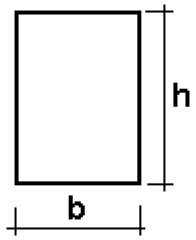
A material being defined may be structural or non-structural (if the *Non-structural material* option is switched on). If it is a non-structural material, then two values are defined for it: unit weight and unit price. A value of unit weight is the quantity needed to calculate the load value when a structure is modeled in the ***ROBOT Millennium*** program. If it is a structural material, then additional parameters are defined (apart from unit weight and unit price): Young's modulus E, Poisson's ratio $\nu$ and resistance (these quantities are used in calculations in the ***ROBOT Millennium*** program).

There is a possibility to choose a color and a filling pattern for each material; switching on the *Filling (2D architectural view)* option allows selection of a filling type which will be presented in the 2D architectural view.
For each material (except layered materials) the user may choose a texture; a texture may be presented on objects when 3D view with shading is activated. The *Texture* option is provided in the bottom part of the dialog box; if it is switched on, it enables selection of a *.bmp or *.jpg extension file. Apart from that, the bitmap dimension should be adjusted (using the scale with the slider) to the actual dimensions. If texture is not ascribed to a material (the *Texture* option is switched off), then in a 3D view elements made of such a material are shown in a color determined on the *Colors* tab in the Preferences dialog box.

*NOTE: While defining a texture, it should be noted that it is laid on textured elements analogously as wall tiles. To obtain a relatively uniform surface, creation of a graphic file with point symmetry retained, is required.*

A cost of a formwork is specified for individual elements of a structure (NOTE: stairs are treated as slabs). A formwork area is determined automatically by the program; results of the formwork estimation are presented along with the cost in the summary table on the *Objects – Details* tab after the *Cost – RC structure* option has been selected.
Formwork areas for individual elements of a structure are calculated as follows:

- beams and continuous footings: the product of the sum of 2 heights h and the width b of a cross-section by the element length L (see the drawing below) = (2 h + b) * L




- columns: the product of a perimeter of a column and its length

- spread footings:

  - the product of a perimeter of a spread footing and its height (rectangular spread footings)
  - the sum of vertical and oblique faces (trapezoidal spread footings)

- slabs: the slab area (in the case there are floor cuts/openings within a slab, the area of these cuts is subtracted)

- walls: the product of a height and a double length of a wall

- stairs: the sum of an area of a slab (bottom and lateral slabs) and the product of a width, a height and a number of stairs.

After switching on the *Set as Default* option, selected materials will be saved as a default set.
There are two buttons available in the ***Material database*** dialog box: **Import** and **Export**. Their pressing enables reading and saving, respectively, the material database in the program internal format. It allows e.g. transferring material databases from a program installed on other computer or copying a material database defined for a different language (by standard, there are separate material databases defined for different working languages).
Pressing of the **Restore Default** button restores default materials proposed in the ***CBS Pro*** program.

## 6.29. How to Add a New Material to the Material Database

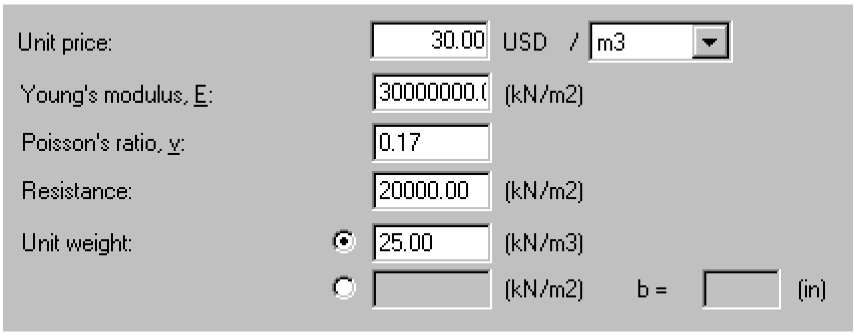
To add a new material (e.g. concrete Co30) to the material database, follow the steps below:

- select the menu command *Tools / Material Database* or press the *Material Database*  icon on the toolbar

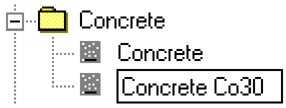
- select the *Concrete* list (it will be highlighted in blue)

- press the **Add** icon

- enter data as in the drawing below




- to modify the material name - highlight the material Co (it will be highlighted in blue) and click once with the left mouse button
- type: Concrete Co30
- press the **Enter** button.




## 6.30.3D Object Database

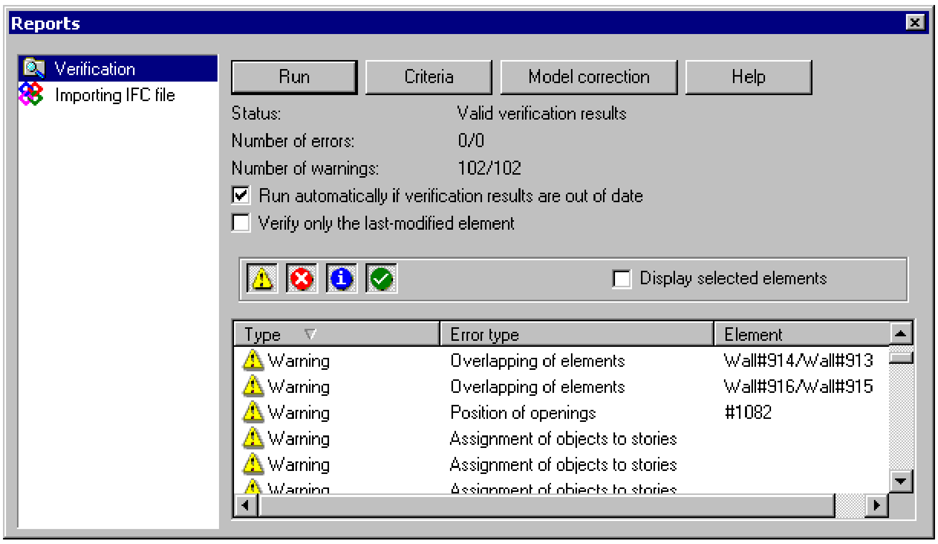
The option is not active in the program until an IFC file including 3D objects is loaded; objects are added to the *Others* group.

## 6.31.Reports

The option enables the user to verify whether the properties of elements included in a defined structure are correct, to verify import of DXF and IFC format files as well as to verify the design performed in the RC modules of the ***ROBOT*** program (RC beams, RC columns, foundations). The option is accessible:

- from the menu by selecting the *Tools / Reports* command
- after pressing the F7 key.

*NOTE:* *The dialog box below is activated automatically after opening *.dxf or *.ifc format files as well as after designing RC elements in the* **ROBOT** *program and returning to the* **CBS Pro** *program.*



*NOTE:* *After selecting the* Tools / Reports *option the dialog box always displays the recently-generated report.*

The left part of the above dialog box holds the list of possible reports:
- *Verification*
- *Static analysis* (for simplified calculations)
- *Calculations* (for advanced calculations)
- *Design of RC elements*
- *Importing IFC file*.

The right part of the above dialog box includes a list of errors/warnings for selected reports, composed of:

- code (icon) in the *Type* column; the following codes may be displayed:

  - error (e.g. there is no calculated amount of the steel needed)
  - warning (e.g. the steel has been calculated, but there are warnings from the RC module of the ***ROBOT*** program)
  - modification of geometry
  - information
  - confirmation of correctness (OK)

  The messages above may be filtered in the dialog box: pressing the icons shown above (switching on / off the option) results in presenting / hiding warnings of a given type on the list of all warnings; moreover, there is the *Display selected elements* option - if it is activated, then messages concerning a selected object / objects are displayed (e.g. it makes it easier for user to find out how many warnings about incorrect positions of nodes refers to selected objects)

- description in the Description column (for Design of RC elements and Importing IFC file) or error type for Structure Verification

  - error (display of the error cause)
  - warning (display of the warning list)
  - modification of geometry (e.g. display of modified sections for spans)
  - OK

- number of a structure element (for *Design of RC elements* and *Structure Verification*).

*NOTE:* *After setting the cursor on a selected element in the right part of the* **Reports** *dialog box, a hint showing additional information (e.g. file path, display of modified sections, etc.). appears on screen.*

*NOTE:* *There is a possibility of sorting the items in the right part of the dialog box by clicking with the left mouse button on a column heading (e.g. on the heading of the* Type *column).*

For the *Design of RC elements* report:

- selection of an element(s) in the dialog box results in selection of the corresponding element in the structure view

- if a selected element does not belong to the story currently presented on screen, then the story view is changed automatically

- if several elements belonging to different stories are selected, the program selects the closest story.

STRUCTURE VERIFICATION
The right part of the dialog box, after selecting the *Verification* option, presents results of structure verification. The following buttons are provided in the top part of the dialog box:

- **Start** – pressing this button starts verification of a defined structure
- **Criteria** - pressing this button opens the ***Verification Criteria*** dialog box where parameters of verification can be determined; the user may run verification assuming different levels of detail
- **Model correction** - pressing this button allows adjusting the height of ends of columns, walls and partition walls to axes of beams and slabs on individual stories; as a result, an axial model of a structure is created so that it is possible to perform calculations; the operation of snapping nodes is carried out for values of parameters defined in the Model Correction dialog box

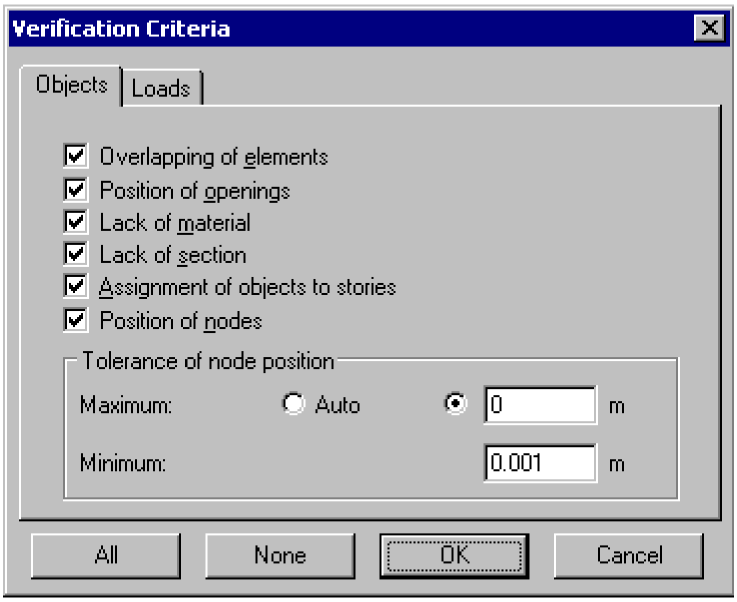
  NOTE: in the current program version it may happen that not all problems related to object positions will be eliminated after performing the *Model correction* operation; this operation reduces the number of existing inconsistencies, however, once it is completed, it is necessary to run *Verification* to assess if a model is correct.
- **Help** - pressing this button opens the help for the dialog box.

Below, the fields showing the current status of structure verification are located.

- *Status* – shows the current verification status; the status *valid verification results* means that since last verification the structure has not been modified, while *verification results out of date* indicates that since last verification the structure has undergone modifications.
- *Number of errors* – number of messages about errors in a structure; the user should remember that one incorrectly defined element is very often responsible for many error messages
- *Number of warnings* – number of warning messages; warnings do not necessarily lead to calculation errors, because the user may deliberately allow for situations that are reported as warnings (the verification takes note only of their occurrence).

If the option *Run automatically if verification results are out of date* is switched on, then automatic verification of a whole structure is run if the status changes to out of date.
Below is the *Verify only the last-modified element* option; it allows removing errors that occur during structure modeling on an ongoing basis while working with a structure model. If the *Verify only the last-modified element* option is switched on, then after modifying an element (or after adding a new element) the program runs the structure verification concerning only that element and all the elements connected with it. Errors and warnings that result from the performed verification of a structure model will be displayed as messages on the screen and additionally, they will appear in the ***Reports*** dialog box. Switching on the *Verify only the last-modified element* option is recommended while working with a structure model. NOTE: If the *Verify only the last-modified element* option is switched on, then the *Run automatically if verification results are out of date* option is not available.

*NOTE:* *For large structures verification (particularly, if the option of object verification named* Position of nodes *is switched on) may last very long.*




The above dialog box consists of two tabs: *Objects* and *Loads*.
Geometrical verification of objects (options provided on the *Objects* tab) enables searching elements that:

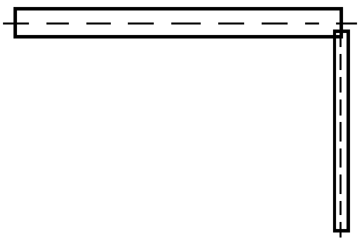
- overlap (the *Overlapping of elements* option is switched on) – the option detects automatically elements of the same type (beam, column, wall, etc.) or their parts that occupy the same space

- are not ascribed material (the *Lack of material* option is switched on) – the option detects elements to which no material has been assigned

- are not ascribed section (the *Lack of section* option is switched on) – the option detects elements to which no section has been assigned

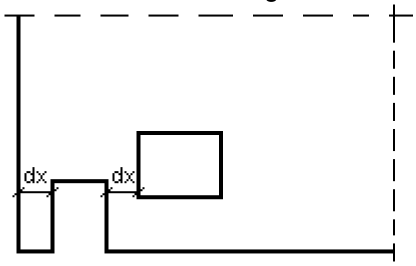
- are positioned out of the story to which they are assigned (the *Assignment of objects to stories* option is switched on) – the option detects elements that after being defined on a given story have been moved above or below the story dimensions

- are in contact and at the same time have no common nodes (the *Position of nodes* option – switched on) – the option enables detection of these elements that have a common geometrical part and simultaneously for which there is no common point on the lines describing their axes (see the drawing below presenting such a situation for a beam-column connection); it means that these elements are actually integrated with each other, but the calculation model treats them as separate elements.



The option *Tolerance of node position* enables defining the size of an area in the vicinity of the node (element's end) and if another element is positioned within this area, then the program treats them as touching elements. It is possible to define the maximum as well as the minimum tolerance value. For the maximum tolerance value, switching on the *Auto* option means that the tolerance dimensions are assumed as equal to geometrical dimensions of elements; such an assumption implies an actual contact of elements in a structure.

Switching on the *Position of openings* option allows search for openings positioned out of elements to which they are assigned; this option detects openings whose geometry is not contained within an object in which they have been defined and openings which overlap with other openings. Additionally, the program verifies positions of door openings in walls; if a coordinate of the door bottom equals zero for individual stories (see the drawing below), then verification of positions of doors will be concerned with their distance from other objects (edges of walls or windows) in order to eliminate too small distances dx that might result in incorrect generation of an FE mesh - see the drawing below.



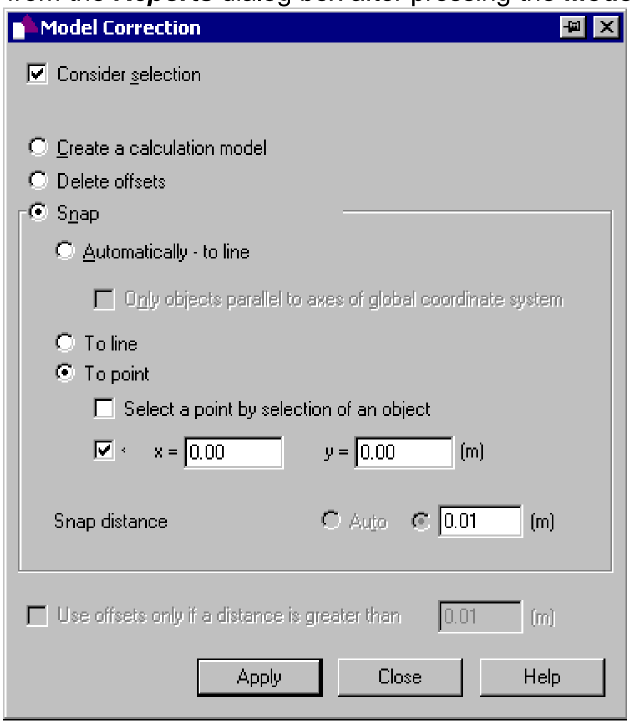

Verification of defined structure loads (options located on the *Loads* tab) enables searching for the following loads:

- loads whose geometry does not correspond to any of the objects; it means that part of the load or entire load is not acting on the structure
- loads that are not taken into account in any load pattern; it causes this load not to be considered during generation of code combinations.

Moreover, the program verifies the sum of loads applied to a structure and the sum of calculated reactions. If these sums are not congruous, the message *Imbalance of reactions and loads* appears (for details see the ***Summary table*** dialog box on the *Reactions* tab).

## 6.32. Structure Model Correction

The option is used to determine parameters of the operation of correcting coordinates of structure nodes; the option may be particularly helpful when importing a model from architectural programs. The option is available from the menu by selecting the *Tools / Model Correct* command or from the ***Reports*** dialog box after pressing the **Model correction** button.



An architectural model usually includes elements whose axes lie on different levels; it means that they do not lie on the level of the story floor (inaccuracies of the structure definition between individual stories), e.g. if all beams are faced (aligned) to the top surface of the slab, then both their axes and ends of columns, walls and partition walls are positioned on a level different from the level of the slab axis.
Thus elements of an architectural model may only touch each other, they may not intersect; for this type of a model, continuity of a model (calculation model) would not be achieved in the ***CBS Pro*** program. All inconsistencies of this type are signaled with the *'Position of nodes'* warnings in the Reports dialog box.
If the *Consider selection* option is switched on, then chosen operations will be performed only on selected objects.
The **Model correction** option results in adjusting the height of ends of columns, walls and partition walls to axes of beams and slabs on individual stories; in consequence, an axial model of a structure is created so that it is possible to perform calculations.
The the **Model Correction** dialog box contains the following options:

- *Create a calculation model* - after selecting this option, the program will automatically determines main levels of floors in a model and main lines on which structure elements are positioned, and next, individual model elements are snapped to main levels and lines; NOTE: the option is intended for structure models imported by means of an IFC file or structure models created in the ***RCAD Formwork Drawings*** program

- *Delete offsets* - selecting this option ascribes coordinates that an object has in the architectural view and deletes offsets

- Snap:

  *- automatically - to line*
  after selecting this option, the program will automatically determines main lines on which structure elements are positioned, and next, snaps to them individual model elements whose nodes are positioned within a user-defined snap distance from main lines determined automatically

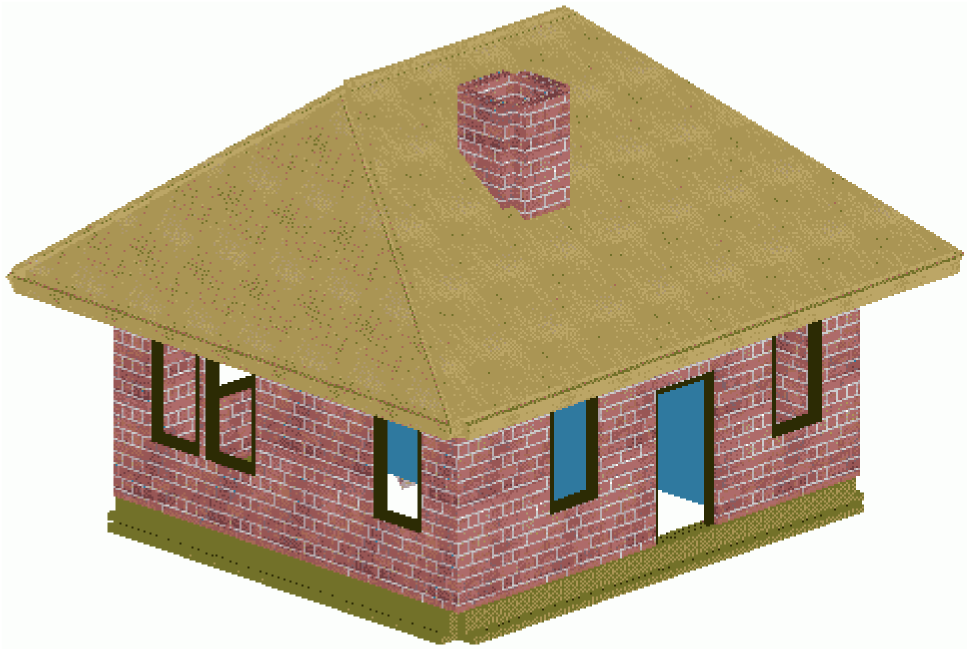
  *- to line*
  after selecting this option, nodes positioned in a distance not greater than the snap distance, will be snapped to it; the snap distance may be defined as one used by default in the program or defined by the user

  *- to point*
  after selecting this option, nodes positioned in a distance not greater than the defined snap distance from a selected point, will be snapped to it; the snap distance is defined by the user

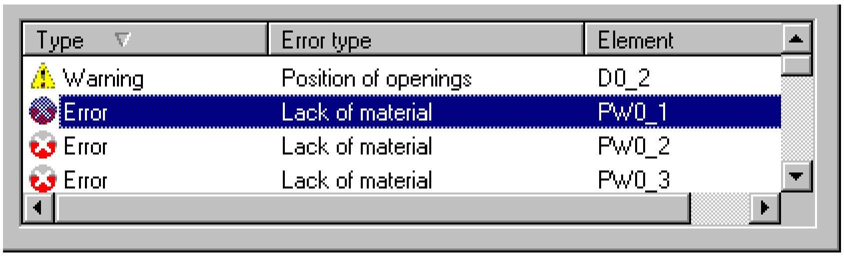
- *Use offsets* - after switching this option on (available for the *Snap to line* option), an object is snapped to the line with assigning an offset (in the computational model the object is snapped, while in the architectural view its current position will be kept - the offset will be set).

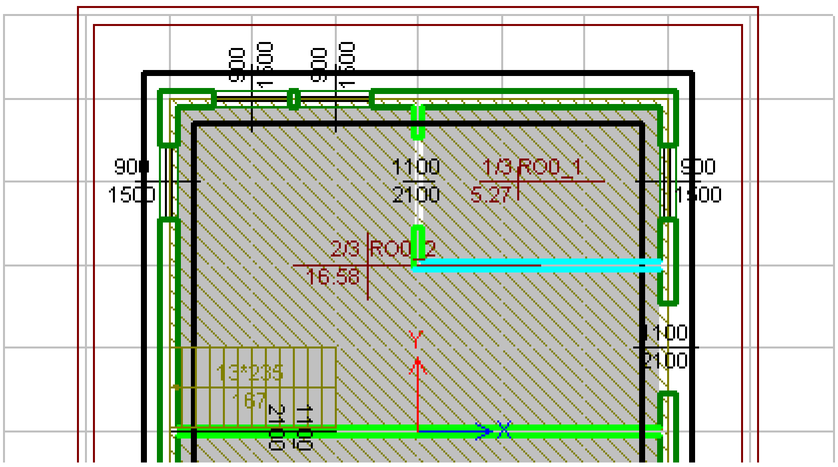
## 6.33.How to Verify a Structure

To perform verification of the structure shown in the drawing below, follow the steps listed below:



- select the command *Tools / Reports* or press the **F7** key
- press the **Criteria** button
- on the *Objects* tab press the **All** button, and next, the **OK** button
- press the **Run** button
- click with the left mouse button on the error message (an element to which a given error refers, will be highlighted automatically).

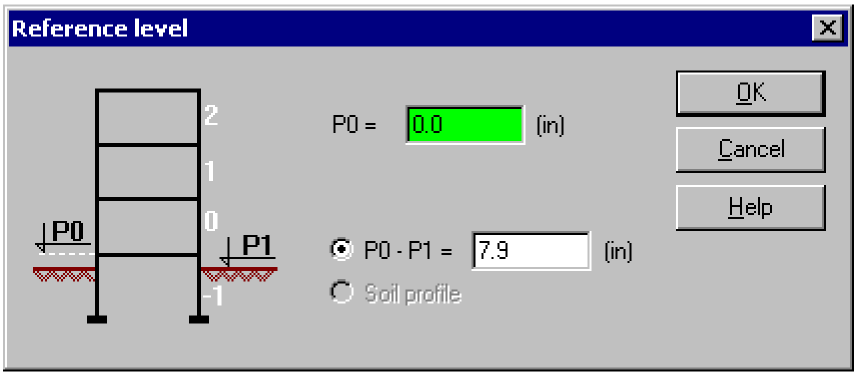


# *7. DEFINITION OF LOADS - DEFAULT LOADS*

## 7.1. Reference Level

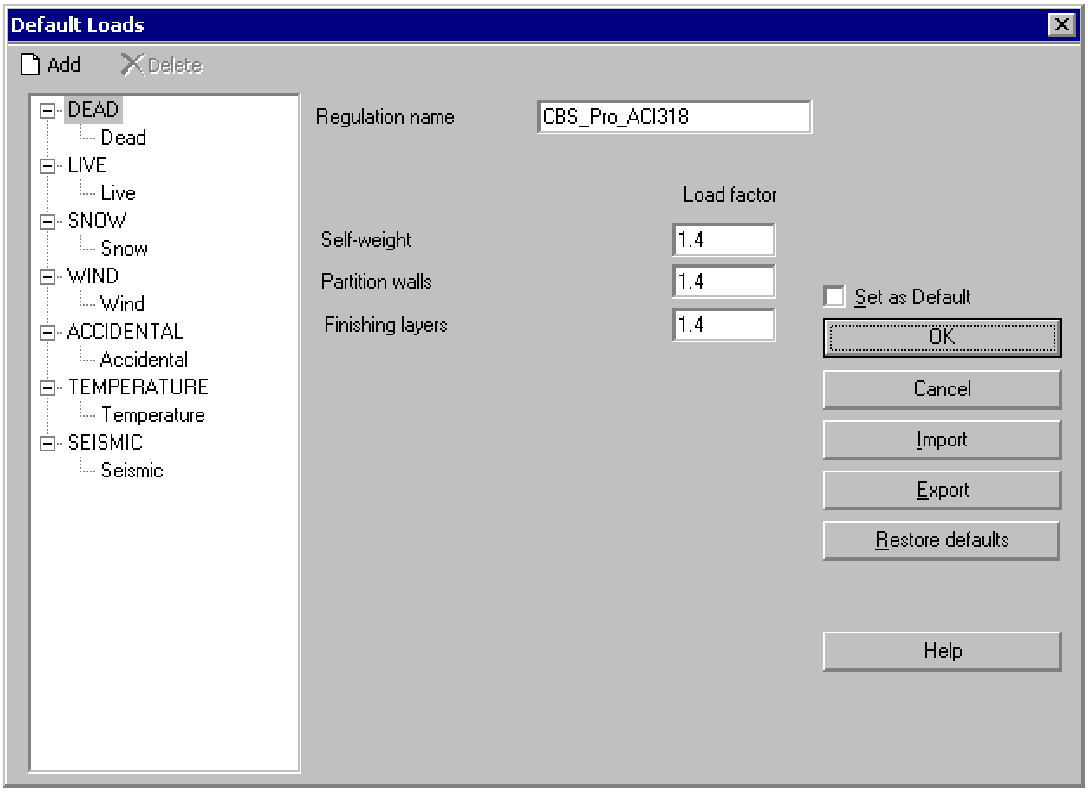
The reference level dialog box is used to determine the height of the walking surface of the ground floor with respect to the structure model and the difference of the height between of the walking surface level and the soil level; it is of significance when defining a height of the soil layer loading the foundation, as well as when generating a wind load and performing the simplified seismic calculations. The option is accessible by selecting the *Structure / Reference Level* command from the menu.




In the case of the wind load, the part of a structure below the soil level is ignored in calculation of the wind pressure onto the building walls.
For the simplified seismic analysis the soil level is significant when masses of individual stories are calculated; the masses of the stories (and their parts) located under the soil level are disregarded, whereas the masses of the parts of the stories situated above the soil level are included in the mass of the first story that is located entirely above the soil level.
The above dialog box contains two edit fields that enable definition of:
- P0: height of the walking surface of the ground floor
- P0-P1: difference of the height between the walking surface of the ground floor and the soil level.

## 7.2. Default Loads

The option enables assigning default values of load records connected with different load groups. The option is available:
- from the menu by selecting the option *Edit / Default Values / Loads*
- after pressing the icon .




In the above dialog box the user may determine a default value of concentrated forces as well as linear and planar loads for groups generated manually. For semi-automatic groups there is a possibility to define parameters being the basis for generation of the loads. For groups generated automatically a value of load factor (partial safety factor) may be determined.
Options in the dialog box allow adding or deleting a load group. Each group has to be assigned to one of load natures. There are seven load natures distinguished in the program: *dead*, *live*, *snow load*, *wind load*, *accidental load*, *temperature load* and *seismic load*, to which groups can be assigned. The basis for generation of a default set of load groups is a file with regulations of code combinations of the ***ROBOT*** program (*.rgl). In the ***CBS Pro*** program the user may add or delete any group.

NOTE: In the ***CBS Pro*** program two load natures (the wind load and the seismic load) have been extended by adding a possibility to generate loads according to the available snow/wind or seismic codes.

After switching on the *Set as default* option, the selected loads are saved as a default set.
The following buttons are located in the bottom part of the dialog box:

- **Import** – enables saving load groups and their factors to a regulation file (*.rgl)
- **Export** – enables opening any regulation file (*.rgl)
- **Restore Defaults** – enables restoring default settings proposed in the ***CBS Pro*** program.

*NOTE:* *In the dialog box the user may delete only such a group to which no load records are ascribed; records defined automatically cannot be deleted; new regulations may be read only then when no load records have been generated.*

*NOTE:* *Load records and load groups are generated according to the rules of load definition in the* **CBS Pro** *program.*

*NOTE:* *While exporting a structure model from the* **CBS Pro** *program to the* **ROBOT** *program, a regulation file CBS_Pro_code_name.rgl is generated in the USER / CFG folder of the* **ROBOT** *program.*

# 7.3. Default Loads - Wind

## 7.3.1. Default Loads - Wind

The *Default Loads / Wind* option available in the ***Default Loads*** dialog box is used to define a wind load according to the rules specified in a selected snow/wind code.
There are three possibilities to generate the wind load:

- if the options: *Direction 1* and *Direction 2* are switched off, then the wind load will not be generated
- if the options: *Direction 1* or *Direction 2* are switched on, then the wind load will be generated on a selected direction (e.g. direction 1)
- if the both options: *Direction 1* and *Direction 2* are switched on, then the wind load will be generated on both directions.

*NOTE:* *As a result of calculations performed for the determined directions of wind action (1 and 2), two load cases with the opposite senses will be generated for each direction, i.e. Wind 1+, Wind 1-, Wind 2+ and Wind 2-.*

The dialog box contains the following options:

- regulation name – a name of the file with code combination regulations
- snow/wind code – a list of available wind load codes
- Direction 1 and 2:
- Angle – a value of the angle between the direction of the wind action and the X axis
- parameters depending on the selected wind load code.

The current version of the ***CBS Pro*** program enables generation of wind loads for the following snow/wind codes:
- American code ASCE-7-02
- British code BS6399 (part 2)
- French code NV65 (modification 04/2000)
- Polish code PN-77/B-02011

Wind loads will be generated automatically in the following range (depending on the calculation method):

- *simplified calculations:* distribution of loads onto walls and columns of buildings (loads onto the roof are disregarded)
- *advanced calculations:* generation of linear loads onto panels and bars positioned on the level of the story floors; generated forces applied to stories equal the forces obtained from simplified calculations.

On the floor level of each story pressure and suction forces are added up and depending on a selected method:

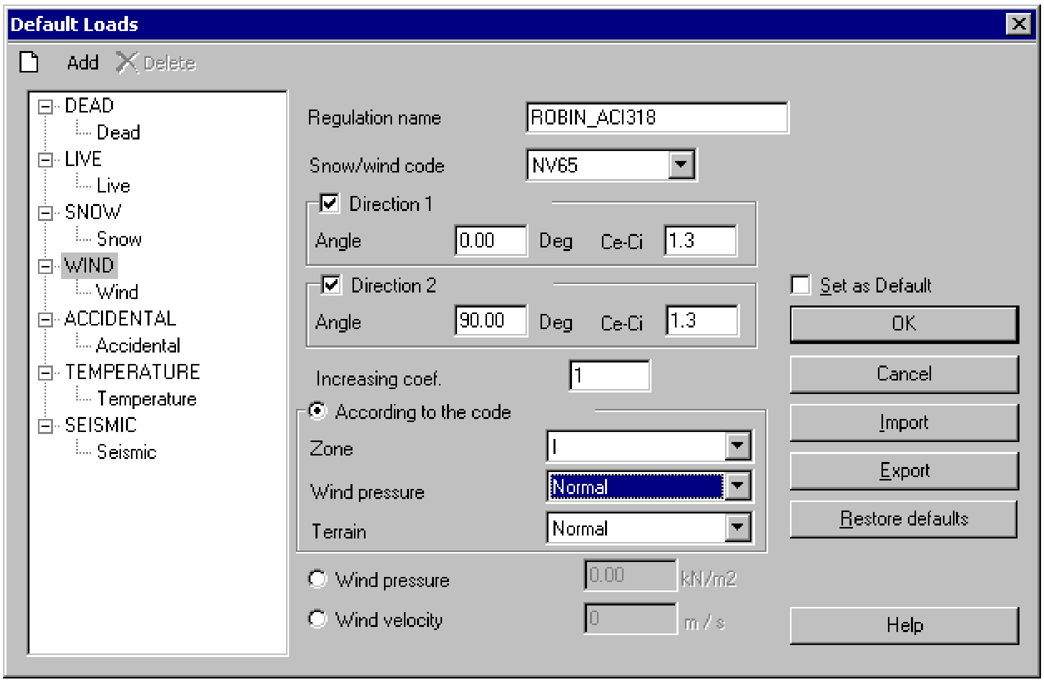
- they are distributed onto individual structure elements (for simplified calculations)

- they are applied as linear loads to panels (floor slabs) and beams that lie on the story level (for advanced calculations).

Once calculations are performed, data needed for defining a wind load and values of forces on individual stories are added to components of a calculation note.

### 7.3.2. Default Loads - Wind (the French Code NV 65)

Once the French wind load code is selected, the *Default Loads / Wind* option is used to define a wind load according to the rules of the French code NV65 (modification 04/2000).




Wind loads according to the French code NV65 are calculated based on the simplified method for typical objects with a height h and a rectangular base (with sides a and b), fulfilling the following requirements (NV65 point 2.91):

a) h<30m

b) h/a>0.25 or
   h/a<2.5 with the additional condition: b/a<0.4 if h/b>2.5

c) f<h/2 for roofs with 2 flat surfaces
   f<2/3*h for vaulted roofs, where f is the roof height

d) the roof cover is:
   - a flat roof
   - a single roof with the height f, with one or two roof surfaces sloping at more than 40°
   - a vault with the plane tangent to the beginning of the lines creating the vault (slope more than 40° and less than 22°)

e) vertical walls have to:
   - be supported directly on the soil
   - be flat without offsets
   - show 5% permeability (NV65 R-III-1,241) at the most, or 35% permeability at the least, for one of the walls

f) the structure site should be a clearly horizontal terrain with a large area (NV65 R-III-1,241).

*NOTE:* *If a building does not meet these requirements, it is possible to determine a value of the wind pressure or velocity or to define the wind load manually.*

The following data has to be determined when defining the wind load:

- *general data*:
  - active wind directions
  - angles between the wind directions and the X axis
  - values of the $C_e$-$C_i$ coefficients for individual directions
- *detailed data for use in the calculation according to the simplified method from the NV65 code*:
  - wind zone
  - wind pressure (normal, extreme)
  - terrain type (protected, normal, exposed)
  - value of the increase coefficient
- *when manual definition of the wind pressure is chosen:*
  - value of the wind pressure or value of the wind velocity.

**Calculation of the wind pressure value**

By standard, the value of the wind pressure is determined as constant along the whole height
$q_0 = (460 + 7h)k_r k_s$ (NV - point 2,991),
where:
h – building height
$k_r$ – regional coefficient
$k_s$ – localization coefficient
$q_0$ – wind pressure expressed in N/m*m.
The pressure value calculated from the above formula is modified according to the formula below:
$q_1 = q_0 a \delta (C_e - C_i)$ (NV - point 2,922-3),
where:
a – increase coefficient
$C_e$-$C_i$ - pressure coefficient
$\delta$ - reduction factor according to point 2,922 (the factor $\delta$ can be calculated from fig. R-III-9 - page 129).

If the wind pressure value is specified, the $q_0$ value is already known and modified according to the formula: $q_1 = q_0 (C_e - C_i)$.
If the wind velocity value is given, the following formula applies:

$$q_0 = \frac{V^2}{1{,}63}$$ (NV - 1,21),

where V is the wind velocity expressed in (m/s), and $q_0$ is the wind pressure expressed in N/m*m.
The thus-calculated wind pressure value is modified according to the formula: $q_1 = q_0 (C_e - C_i)$.

*Calculation of the value of the wind pressure (force) acting on a story*

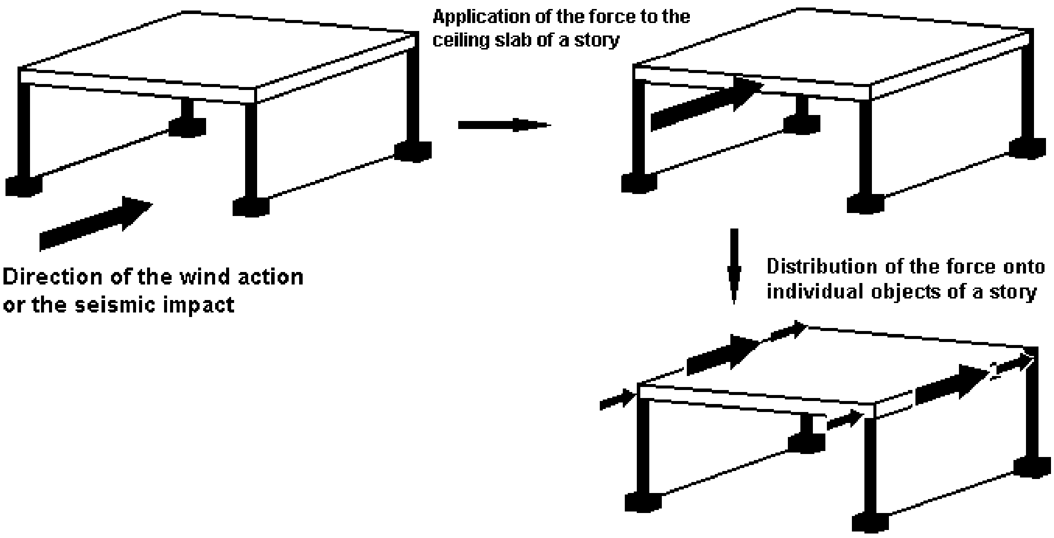
The wind pressure force acting on individual stories is calculated from the following formula:
$F_i = q_1 b_i h_i$
where:
$b_i$ – building width on the story *i*
$h_i$ – height of story *i*
The calculated pressure force is applied at the level of the floor above the story *i*.

*Distribution of the pressure force onto individual elements*

The total action of the wind pressure on a story is applied to the ceiling slab of this story, and then is distributed onto individual objects of the story (distribution of this action is illustrated schematically in the drawing below).

- columns and walls parallel to the direction of the wind action, or slanting walls (in this case, the equivalent stiffness of walls for the appropriate direction of the force action is calculated) for a frame or a mixed structure.

- walls parallel to the direction of the wind action, or slanting walls (in this case, the equivalent stiffness of walls for the appropriate direction of the force action is calculated) for a membrane or a mixed structure.

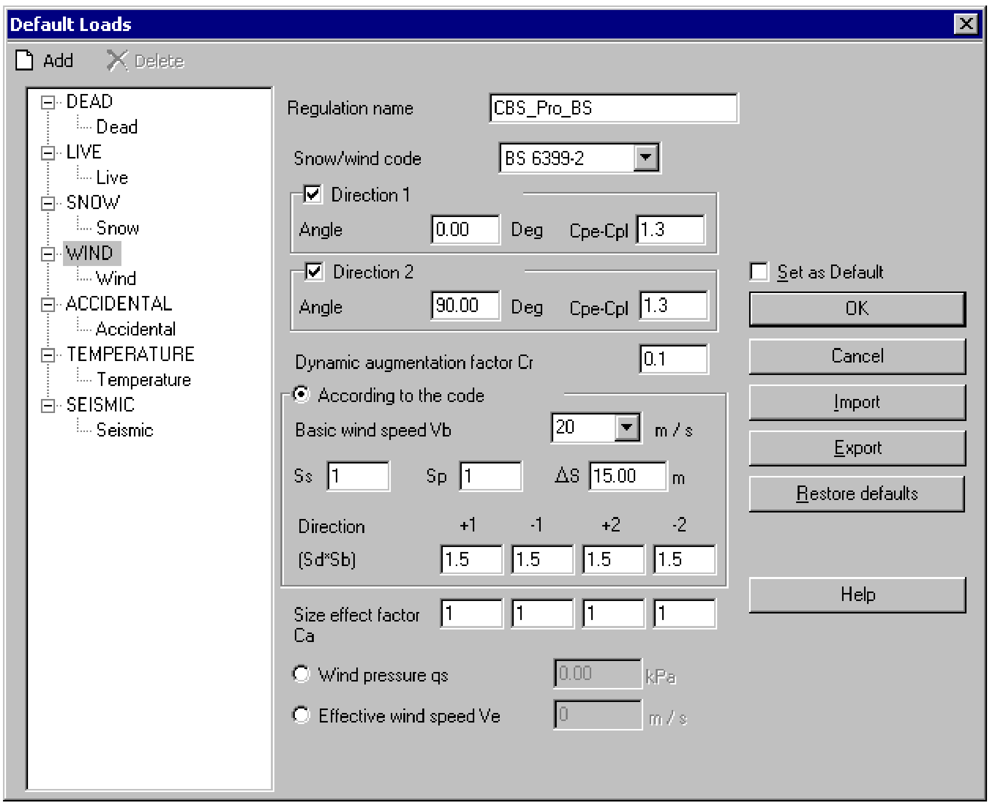



NOTE: *Individual structure objects (walls, columns) may be treated as elements that do not carry horizontal forces (resulting from the wind pressure, seismic loads).*

Forces acting on the objects of the story *i* are recalculated to obtain the equivalent forces acting on the objects of the story *i-1* together with a bending moment (the result of the action of these forces on the arm equal to the story height).

### 7.3.3. Default Loads - Wind (British Code BS)

After selecting the British wind load code, the *Default Loads / Wind* option is used to define a wind load according to the guidelines of the British code BS6399 (part 2).



Wind loads are generated according to the British code BS 6399; loads are distributed onto walls and columns of buildings (disregarding the load applied to the roof).

The following data has to be determined when defining a wind load:
- *general data*:
  active wind directions
  angles between wind directions and the X axis
  values of $C_{pe}$-$C_{pi}$ coefficients for individual directions (pressure onto the windward wall + suction into the leeward wall)
- *detailed data in the case of calculations according to the simplified method from the code BS6399*:
  coefficient Cr (dynamic augmentation coeff.)
  velocity Vb
  modification coefficient (Sd*Sb) - 4 values for directions: +1, -1, +2, -2
  modification coefficient Ca - 4 values for directions: +1, -1, +2, -2
- *in the case the manual definition of the wind pressure is chosen:*
  value of the wind velocity Ve or the wind pressure qk.

**Calculation of the wind pressure value**
Value of the wind pressure

$$V_S = V_b \cdot S_a \cdot S_d \cdot S_b \cdot S_S \cdot S_p \qquad (8\text{-}9)$$

where:
SS = 1.0 - seasonal coefficient
Sp = 1.0 - probability coefficient.
Topography is not considered during calculations: the following formula is applied:
Sa = 1 + 0,001 * ΔS,
where ΔS is an altitude above the sea level.

Calculation of coefficient values for individual directions:
the following simplification is adopted: the wind velocity is uniform for the entire height of a building

The value qs
- in the case a value of the wind velocity is defined manually, the code formula is applied $q_s = 0{,}613 \cdot V_e^2$;
- in the case the wind pressure is defined manually, the value qs is adopted.

The value Cr is defined by the user

Value of the wind pressure

$$p = q_s \cdot (C_{pe} - C_{pi}) \cdot C_a \cdot (1 + C_r)$$

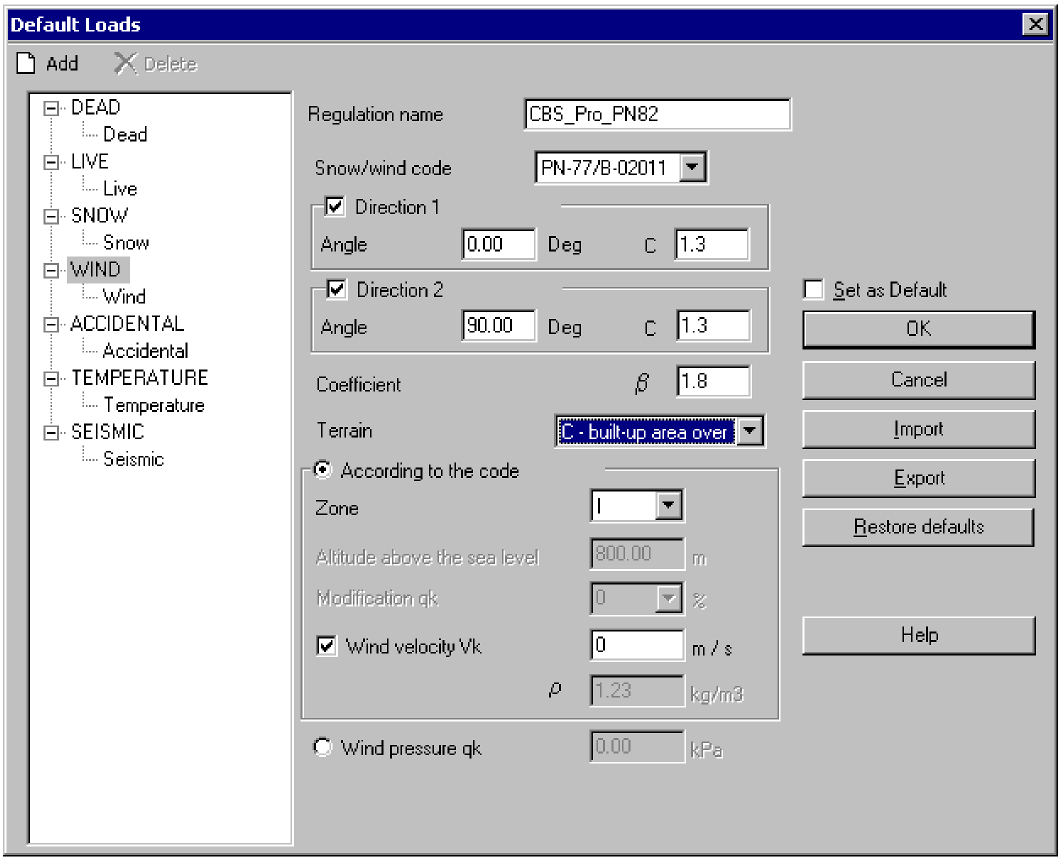
For the directions -1, -2 the program calculates a value opposite to 1 and 2, but modified by the coefficients for the directions -1, -2:

$$p_{-1} = p_1 \cdot \frac{(C_a \cdot S_d \cdot S_b)_{-1}}{(C_a \cdot S_d \cdot S_b)_1}$$

$$p_{-2} = p_2 \cdot \frac{(C_a \cdot S_d \cdot S_b)_{-2}}{(C_a \cdot S_d \cdot S_b)_2}$$

### 7.3.4. Default Loads - Wind (Polish Code PN)

After selecting the Polish wind load code, the *Default Loads / Wind* option is used to define a wind load according to the guidelines of the Polish code PN-77/B-02011.




Wind loads are generated according to the Polish code PN-77/B-02011; loads are distributed onto walls and columns of buildings (disregarding the load applied to the roof).

The following data has to be determined when defining a wind load:
- *general data*:
  active wind directions
  angles between wind directions and the X axis
  values of C coefficients for individual directions (pressure onto the windward wall + suction into the leeward wall)
- *detailed data in the case of calculations according to the code PN-77/ B-02011*:
  coefficient β (actions of wind gusts)
  terrain type (A - opened, B - built-up area up to 10 m, C - built-up area over 10 m)
  wind zone {I; II; IIa; IIb; III}
  - if zone III is selected – altitude above the sea level
- *in the case the manual definition of the wind pressure is chosen:*
  value of the wind velocity Vk (and the value ρ for zone III) or the wind pressure qk.

**Calculation of the wind pressure value**

it is calculated from the formula below for given code parameters

$p_k = q_k \cdot C_e \cdot C \cdot \beta$ (PN - point 2.2 formula 1)

where:

qk - a value of the wind pressure

Ce - exposure coefficient (PN - point 4.1 - table 4)

C - aerodynamic coefficient

β - coefficient related to actions of wind gusts

The value qk is a value of the wind pressure; it depends on a selected zone according to table 3 (H is an altitude above the sea level available only for zone III)
- in the case a value of the wind velocity is defined manually, the code formula is applied (PN – 3.4, code 3): $q_k = \dfrac{\rho \cdot V_k^{\,2}}{2}$

where:

Vk - wind velocity

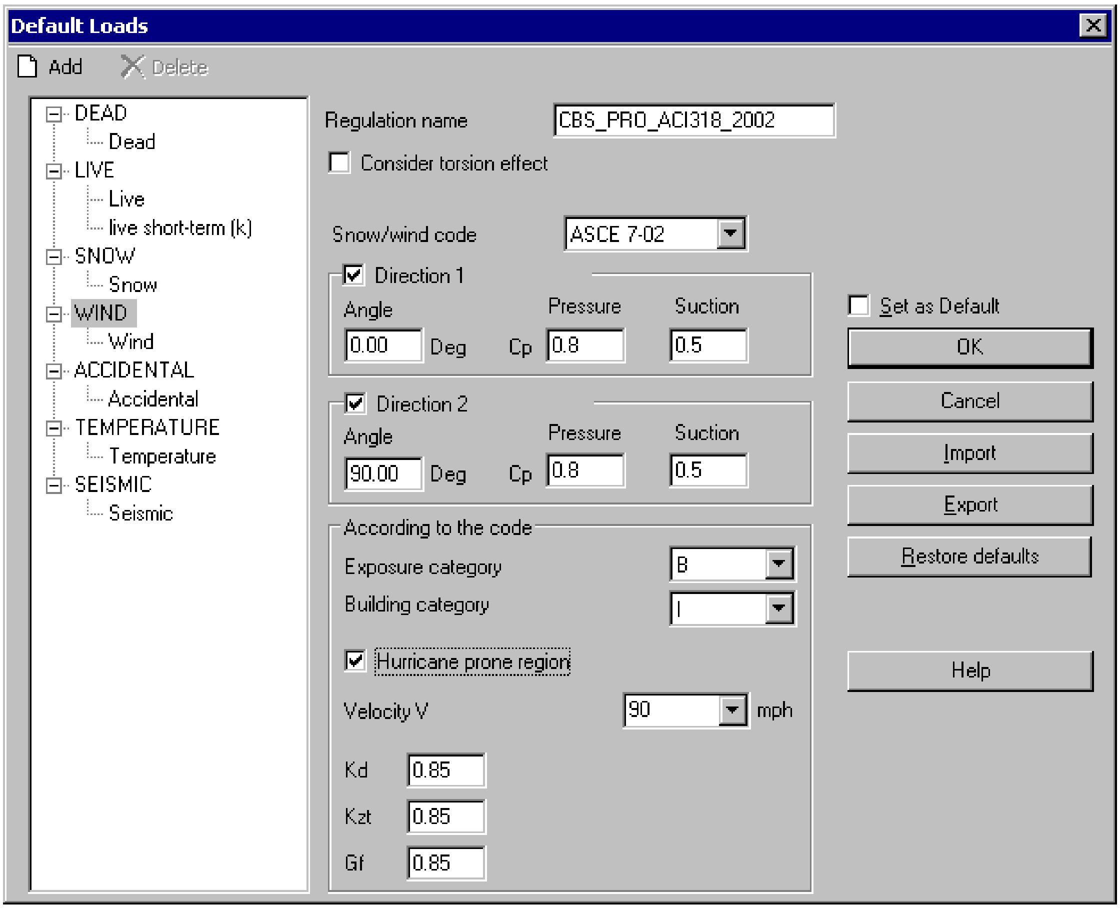
ρ - coefficient equal to 1.23 for zones I and II, and defined by the user for zone III
- in the case the wind pressure is defined manually, the value qk is adopted.

Values of the parameters C and β are defined by the user.

The value Ce
1. for buildings for which H/L > 2 the program assumes a variable Ce value calculated for individual story levels and intermediate levels, depending on a terrain type (according to table 4 of the code PN-77/B-02011)
2. for the remaining buildings – a constant value according to table 4 depending on a terrain type {A; B; C}; value of the coefficient Ce = Ce (z=H).

### 7.3.5. Default Loads - Wind (American Code ASCE-7-02)

After selecting the American wind load code, the *Default Loads / Wind* option is used to define a wind load according to the guidelines of the American code ASCE-7-02.




Wind loads are generated according to the American code ASCE-7-02.

The following data has to be determined when defining a wind load:
- *general data*:
  active wind directions
  angles between wind directions and the X axis
  values of the Cp coefficients for pressure and suction (for pressure Cp = 0.8, but it can be modified)
- *detailed data for calculations according to ASCE-02*:
  exposure category
  building category
  V - wind velocity in (mph)
  values of coefficients defined by the user:
  Kd – wind direction coefficient
  Kzt - topography coefficient
  Gf - wind gust coefficient

*Value of the wind pressure*

for pressure: $qz = 0.00256 * Kz * Kzt * Kd * V^2 * I$

for suction: $qh = 0.00256 * Kh * Kzt * Kd * V^2 * I$

where:
Kz – exposure coefficient determined by the program for levels of individual stories of the building according to ASCE 7-02
Kh = Kz (for the value 'z' equal to the total height of the building)
Kd - wind direction coefficient
V - base wind velocity
I - structure importance factor

*NOTE:* *As specified in ASCE 7-02, the value of pressure from the windward side (pressure) varies depending on the height (Kz); the value of pressure from the leeward side (suction) Kh is constant along the building height.*

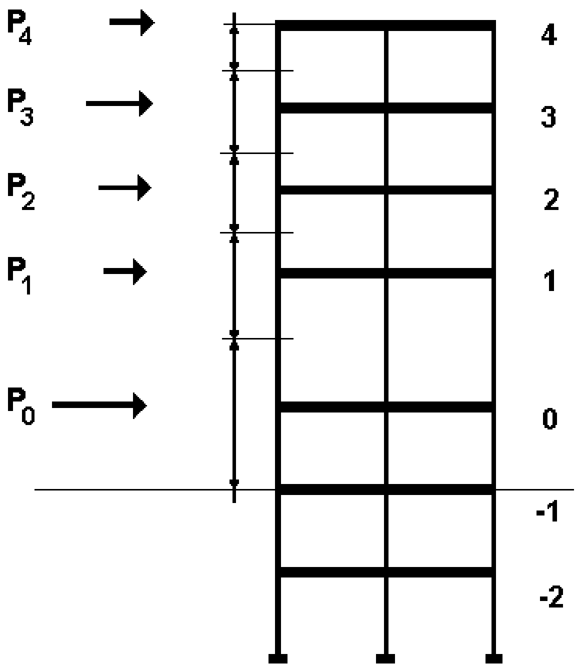
*Values of pressure and suction forces*

windward side: Pz(z) = qz * Gf * Cp

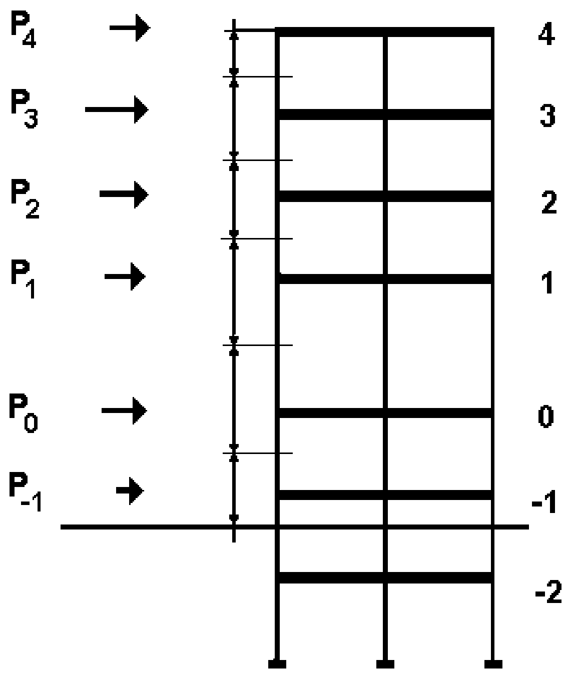
leeward side: Pz = qh * Gf * Cp

### 7.3.6. Application of Wind Loads (Story '0' and the Last Story)

**The basic wind load case of a building:**
Loads are applied to all stories above the reference level; application of loads to the story '0' and the last story (see the drawing below):
- the force on the story '0' is acting on the area of the whole story '0' and half the story '1'
- the force on the last story is acting on half the height of the last story.




If the reference level is positioned below the bottom floor of the level '0' and there are underground stories in the building, then the wind load will be applied to the top floor of the story below the level '0' which is positioned above the reference level (see the drawing below).




# 7.4. Default Loads - Seismic Analysis

## 7.4.1. Default Loads - Seismic Analysis

The *Default Loads / Seismic* option available in the ***Default Loads*** dialog box is used to define the parameters necessary to perform seismic calculations of a structure model according to a selected seismic code. In the present version of the ***CBS Pro*** program it is possible to generate seismic loads for the following seismic codes:

- PS92
- RPA99_03

- RPS2000
- P100-92
- P100-1/2006
- UBC 97
- IBC 2000
- IBC 2006
- Italian seismic code
- spectral analysis (analysis based on a spectrum defined by the user).

*NOTE:* *If the* Seismic code *option is switched off, then seismic/spectral analysis of the structure will not be conducted.*

The programs makes available two methods of seismic analysis of the structure (the *Seismic code* option is switched on; in the analysis seismic cases on the directions X and Y are considered):

- Simplified
- Advanced – the seismic analysis is performed by means of the Finite Element Method.

Seismic or spectral analysis is performed for selected directions considering a value of the coordinate of the excitation vector for individual directions.
The middle part of the dialog box contains options used for defining directions; each of the directions may be switched on (then it will be considered in calculations) or switched off (it will be ignored in seismic / spectral analysis). The edit fields are used to specify values of the coordinate of the excitation vector for individual directions.

| **Simplified method:** | **Advanced method:** |
|---|---|
| 1. allowable directions: X, Y | 1. allowable directions: X, Y, Z |
| 2. values of the vector coordinates (coefficients for a direction) = 1.0 | 2. possibility to change values of each coefficient. |

While performing calculations of the structure with the activated option for generation of code combinations the program generates combinations of seismic directions:
- for the simplified method: Newmark combinations
- for the advanced method: Newmark combinations or quadratic combinations.

At the bottom of the dialog box there are two buttons:

- **Seismic analysis (Spectral analysis)** – opens the dialog box for definition of seismic analysis parameters according to a selected seismic code or for definition of spectral analysis parameters
- **Modal analysis** - opens the dialog box for definition of modal analysis parameters.

In the lower part of the dialog box there is the *Verify necessary wall area* option. It is available for the seismic code P100-1/2006 - see Verification of a necessary wall area (this condition is not given in other codes).

In the upper part of the dialog box there is the *Consider torsion effect* option which is accessible in the case seismic loads generated automatically are activated (for spectral or seismic analysis, according to one of the codes available on the selection list). Once this option is switched on, the following options are available at the bottom of the dialog box:
- normal (normal torsion is considered)
- accidental (apart from normal torsion, accidental torsion is also considered)
- value ea,x: eccentricity in the direction of the X axis, for seismic analysis – on the Y direction
- value ea,y: eccentricity in the direction of the Y axis, for seismic analysis – on the X direction.

### 7.4.2. Default Loads - Seismic Analysis (Simplified Method)

To enable the use of the simplified method in the program, a designed building must satisfy general conditions determined by the individual codes.
Below are presented the conditions for buildings as specified in the available seismic codes as well as the method of calculation of seismic forces:
PS92
RPA99_03
RPS2000
UBC 97
P100-1/2006
Italian seismic code
IBC 2000 / 2006.

For a selected seismic code and structure type the program performs calculation of building displacements using the methods as follows:
- Rayleigh method: for membrane and mixed buildings
- equivalent frame method: for frame buildings.

*NOTE:* *If the simplified method is selected for seismic calculations (selection in the* **Default Loads** *dialog box) and the advanced method for calculations of the entire structure model, then the program will perform advanced calculations in the static range during which seismic forces generated and applied to structure elements as for the simplified method will be used in seismic cases.*
*If the advanced method is selected for seismic calculations (selection in the* **Default Loads** *dialog box) and the simplified method for calculations of the entire structure model, seismic calculations will be performed as for the simplified method.*

### 7.4.3. Simplified Calculation of Structure Displacements due to Horizontal Forces

In the case calculation is performed with the use of the simplified method, for a selected seismic code and structure type the program performs simplified calculation of the building displacements.
For the code PS92 for regular buildings (irrespective of the structure type) displacements are calculated from the formulas described in the pseudostatic method. For the remaining codes (RPA99_03, RPS2000, UBC97, IBC2000, Italian seismic code and for the code PS92 for medium-regular buildings), depending on the building type, displacements are evaluated using the following methods:
- Rayleigh method: for membrane and mixed buildings
- equivalent frame method: for frame buildings.

A. Rayleigh method

a) for the code PS92 - medium-regular buildings:

Step 1: calculation of displacements ui,x ui,y for individual directions X and Y

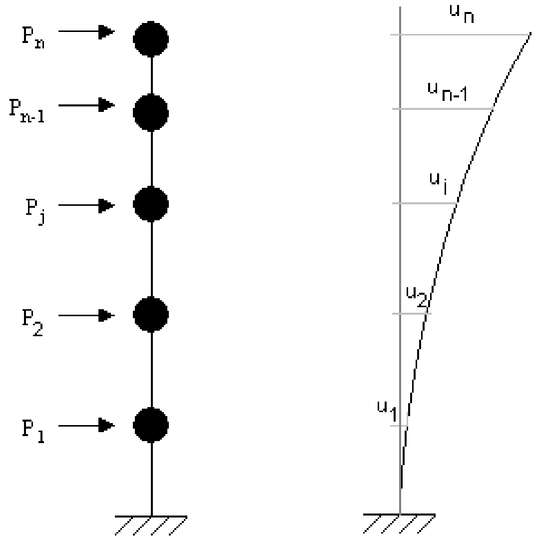
Displacements are calculated with the use of the cantilever method adopting the following assumptions:
- a building is represented by a cantilever bar
- calculations should be performed separately for the directions X and Y
- every story has its stiffness being the sum of stiffnesses of individual structure elements that carry horizontal forces, which equals - according to the formula:

$$E_i J_i = \sum_{k=1}^{m} E_i^k J_i^k$$

where:
m – a number of structural elements on the story i
E – Young modulus of the material which has been used to define the object k

J – moment of inertia about the axis perpendicular to the direction of action of horizontal forces

- horizontal forces Pi = mi*1m/s$^2$ are applied to the ceiling level of each story
- as a result of solving the cantilever, displacements of individual stories are obtained (on the story ceiling level)




Step 2: calculation of displacements from the formulas

$$d_{r,x} = R_x(T_x)\rho_{0x}u_{r,x}\Delta_x\left(\frac{T_x}{2\pi}\right)^2$$

where:
Rx – spectral acceleration
ρ0x – an increase coefficient (ignored modes are considered)
ui,x – a displacement of the story *i* calculated in step 1 for the direction X
Tx – a building vibration period for the direction X
For the direction Y the procedure is analogous.

b) for the remaining codes (RPA99_03, RPS2000, UBC97, IBC2000) step 1 subpoint (a) is carried out assuming as follows:

- horizontal seismic forces Fi calculated by means of the simplified method are applied to the ceiling level of each story.

*NOTE:* *The method described should not be used to evaluate displacements for structures composed of columns and slabs, since displacements calculated for such a structure will be significantly overrated.*

B. Equivalent frame method

Scope of application: frame buildings
Assumptions:

- it is assumed that there are rigid nodes connecting spandrel beams (beams) with the columns, whereas all pinned nodes (releases) defined on beam ends are disregarded
- the program takes account of the method of column –column base connection (fixed or pinned)
- displacements due to horizontal nodal forces have been included in considerations, whereas the influence of changes in column length has been ignored.

Data for calculation:
Qi – a transverse force on the story i (a sum of horizontal forces from the story n to i, the total force on a story and the total stiffness of frames are considered)
hi – a height of the story i
Si – stiffness of columns of the equivalent frame

$$S_i = \frac{1}{2}\sum_k \frac{EJ_{i,k}}{h_i}$$

where:
k – a number of frame columns on the story i=1,...,n
Ji,k - moment of inertia of the column k, on the story i

For the '0' story (connection with the foundation)

$$S_0 = \frac{1}{2}\sum_k \frac{EJ_{0,k}}{h_0} + \frac{1}{2}\sum_l \frac{EJ_{0,l}}{4h_0}$$

k – a number of frame columns with fixed column-column base connection
l - a number of frame columns with pinned column-column base connection

Ri – stiffness of the spandrel beam of the equivalent frame

$$R_i = \sum_m \frac{EJ_{i,k}}{l_m}$$

where:
m – a number of frame spandrel beams on the story i
lm – a length of the spandrel beam m on the story i

Calculations
φi-1 – a rotation angle of the lower node in the frame of the story i
φi - a rotation angle of the upper node in the frame of the story i
ψi – an angle of the frame deviation from the vertical direction on the story i
δi – a horizontal displacement on the story i (displacement with respect to the story i-1)
ui – a total displacement of the story i

$$\varphi_0 = \frac{Q_0 h_0 + Q_1 h_1}{24R_0 + 4S_0}$$

$$\varphi_i = \frac{Q_i h_i + Q_{i+1} h_{i+1}}{24R_i}$$

$$\varphi_n = \frac{Q_n h_n}{24R_n}$$

$$\psi_0 = \frac{\varphi_0}{2} + \frac{Q_0 h_0}{24S_0} \qquad \psi_i = \frac{\varphi_i + \varphi_{i-1}}{2} + \frac{Q_i h_i}{24S_i}$$

After the calculations, relative story displacements are obtained:

$$\delta_i = \psi_i h_i$$

and the next step includes evaluation of absolute displacements of the story i with respect to the building base which are presented in a calculation note:

$$u_i = \sum_{j=0}^{i} \delta_j$$

*NOTE:* *In the case of selecting the frame structure type and creating a structure model without defined beams (spandrel beams of frames), displacements will not be calculated and the table will show displacement values equal to zero (since stiffness of spandrel beams equals zero).*

### 7.4.4. Default Loads - Simplified Method (PS92)

1. The scope of application: buildings that fulfill the general conditions as specified in PS92/6.6.1:
   a) regular buildings - the pseudostatic method is applied
   b) medium-regular buildings - Raileygh's method is applied
   The user performs the classification (criteria of regularity on the plane and in elevation according to PS 92/ 6.6.1.2.1.1; 6.6.1.3.1.1)

2. Design height of a building

the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Pseudostatic method - regular buildings

Calculations should be performed separately for each of the directions X and Y

a) basic vibration periods (for the directions X and Y) – depending on the selected structure type:

frame

$$T_x = 0{,}1\frac{H}{\sqrt{L_x}} \qquad T_y = 0{,}1\frac{H}{\sqrt{L_y}}$$

membrane and mixed structure

$$T_x = 0{,}08\frac{H}{\sqrt{L_x}}\sqrt{\frac{H}{L_x+H}} \qquad T_y = 0{,}08\frac{H}{\sqrt{L_y}}\sqrt{\frac{H}{L_y+H}}$$

b) Equivalent static forces fr and displacements of individual stories dr for the direction X

$$f_{r,x} = \rho_{0x} m_r Z_r^{\alpha}\frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}}\cdot R_x(T_x) \qquad d_{r,x} = R_x(T)\rho_{0x} Z_r^{\alpha}\frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}}\left(\frac{T_x}{2\pi}\right)^2$$

where:

user-defined values:

q – RC behavior factor ($q_x$ for the X direction, $q_y$ for the Y direction), modified by the program in compliance with the code in the following way:

$$q' = \frac{2{,}5\rho}{1-\frac{T}{T_B}\left(1-\frac{2{,}5\rho}{q}\right)}$$

- for T < $T_B$
- in the remaining cases: q' = q (for regular buildings)

$\tau$ - topography coefficient

values calculated by the program:

$\rho$0x – increase coefficient (accounting for the ignored modes)

$$\rho_{0x} = 1+0{,}05\left(\frac{T_x}{T_C}\right)^{3/2} \geq 1{,}05 \quad \text{for frames}$$

$$\rho_{0x} = 1+0{,}10\left(\frac{T_x}{T_C}\right)^{3/2} \geq 1{,}10 \quad \text{for membranes and mixed structures}$$

TC – the maximum period value for the horizontal spectrum segment

mr – mass of the considered story *r* – here are adopted all dead and live loads applied to the floor slab (the top of a given story), multiplied by participation factors (determined in the dialog box), as well as the self-weight of horizontal elements and half the weight of the vertical elements of a given story and of the next story (*r+1*).

In the case of the story *0,* the whole weight of vertical elements of the story *0* is included.

If the soil level is higher than the level of the base of the story *0*, then the weight and the loads of the stories 0 and 1 are included in the story *2.*

zr – the quotient of the story height and the design height of the building Zr = hr / H, where H – the building height counted from the base of the story *0* to the level of the story *n*

$\alpha$ - coefficient dependent on the bracing of the building (frames $\alpha$ = 1.0, walls $\alpha$ = 1.5) spectral acceleration

$R_x(T_x) = a_N \tau \rho R_D(T_x)$

aN – nominal acceleration as specified by the code

$$\rho = \left[\frac{5}{\zeta}\right]^{0,4}$$

where ζ is a damping coefficient expressed in (%)- ($\zeta_x$ for the X direction and $\zeta_y$ for the Y direction)

RD(Tx) – the ordinate of the normalized design spectrum

for the Y direction the procedure looks analogously as for the X direction

$$f_{r,y} = \rho_{0y} m_r Z_r^{\alpha} \frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}} \cdot R_y(T_y) \qquad d_{r,y} = R_y(T)\rho_{0y} Z_r^{\alpha} \frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}} \left(\frac{T_y}{2\pi}\right)^2$$

4. Rayleigh's method – medium-regular buildings

a) Basic vibration periods

$$T_x = \frac{2\pi}{\sqrt{\Delta_x}} \qquad T_y = \frac{2\pi}{\sqrt{\Delta_y}}$$

where:

$$\Delta_x = \frac{\sum m_i u_{i,x}}{\sum m_i u_{i,x}{}^2} \qquad \Delta_y = \frac{\sum m_i u_{i,y}}{\sum m_i u_{i,y}{}^2}$$

mi – mass of the story *i*

ui,x and ui,y – displacement of the story *i*.

- for membranes and mixed structures it is calculated from the solution of the cantilever that represents a building, loaded with the forces Pi=mi*1[m/$s^2$] in the direction of X or Y axes
- for frames it is calculated from the solution of a building model using the equivalent frame method (loading of the building as in the point above)

b) Equivalent static forces fr for the direction X:

$$f_{r,x} = \frac{R_x(T_x)}{q} m_r \rho_{0x} u_{r,x} \Delta_x$$

where:

user-defined values:

q - RC behavior factor, modified by the program in compliance with the code in the following way:

$$q' = \frac{2,5\rho}{1 - \frac{T}{T_B}\left(1 - \frac{2,5\rho}{q}\right)}$$

- for T < $T_B$
- in the remaining cases: q' = 0.85 q (for medium-regular buildings )

values calculated by the program:

ρ0x - increase coefficient (accounting for the ignored modes)

$$\rho_{0x} = 1 + 0,03\left(\frac{T_x}{T_C}\right)^{4/3} \quad \text{for frames}$$

$$\rho_{0x} = 1 + 0,05\left(\frac{T_x}{T_C}\right)^{4/3} \quad \text{for membranes and mixed structures}$$

for the Y direction the calculation proceeds analogously as for the X direction.

## 7.4.5. Default Loads - Simplified Method (RPA99_03)

1. The scope of application: buildings that fulfill the general conditions as specified in RPA99 point 4.1.2

2. Design height of a building
the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Simplified method
Calculations should be performed separately for each of the directions X and Y

a) basic vibration periods (for the directions X and Y) – depending on the selected structure type:
frame

$T = C_T \cdot h_N^{3/4}$,

where:
T – period for the X and Y directions
CT - for the X and Y directions
CT = 0.075 – RC frames
hN – height in (m) measured from the base to the last story

membrane and mixed structure

$T_x = \frac{0{,}09 * h_N}{\sqrt{D_x}}$ $T_y = \frac{0{,}09 * h_N}{\sqrt{D_y}}$, where: $D_x$, $D_y$ – width of the building in the directions X and Y

b) the total force resulting from the seismic impact:
$V_x = A * D * Q / R_x * W$
$V_y = A * D * Q / R_y * W$
where:
user-defined values:
Q – quality coefficient
$R_x$, $R_y$ – behavior factors for the X and Y directions
values calculated by the program:
A - nominal acceleration
D – increase coefficient calculated from damping (RPA'99 formulas 4.2, 4.3), where $\xi$- damping ($\xi_x$ for the X direction, $\xi_y$ for the Y direction)
W – total weight of the structure
$W = \Sigma W_i$, $W_i = W_{Gi} + \beta W_{Qi}$
dead loads and part of live loads, RPA'99 formula 4.5, where $\beta$ - load factor

c) Distribution of the force V ($V_x$ for the X direction, $V_y$ for the Y direction) onto individual stories for Tx > 0.7 (s)
- at the roof level, the following force is acting additionally:

$F_{Tx} = 0{,}07 * T_x V$

analogously, the formula holds for the Y direction, where V is the total seismic force, but $F_T$ is not greater than 0.25 * V (RPA 4.2.5)
- on individual stories – the forces:

$$F_i = \frac{(V - F_t) * W_i * h_i}{\sum_{j=1}^{n} W_j * h_j}$$

where:
Fi – horizontal force on the story *i*
hi – level of the slab to which the force $F_i$ is applied
hj – level of individual stories

wi, wj – weights corresponding to the stories *i, j*

d) displacements of the story *i* for individual directions X and Y are evaluated based on the calculations of the cantilever loaded with forces Fi at the story levels - for membrane and mixed buildings, whereas for frame buildings – with the use of the equivalent frame method
the displacements obtained will be multiplied by the factor $R_x$ or $R_y$ according to the formula 4-19 of the RPA99 code.

### 7.4.6. Default Loads - Simplified Method (RPS2000)

1. The scope of application: buildings that fulfill the general conditions as specified in RPS2000 points 6.2.1.2 and 4.3.1

2. Design height of a building
the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Simplified method
Calculations should be performed separately for each of the directions X and Y

a) basic vibration periods (for the directions X and Y) – depending on the selected structure type:
frame
Tx = Ty = 0.085 N, where N – number of stories in the building
membrane and mixed structure

$$T_x = \frac{0{,}09 * h_N}{\sqrt{L_x}} \qquad T_y = \frac{0{,}09 * h_N}{\sqrt{L_y}}$$

, where: Lx,y - widths of the building in the directions X or Y

b) the total force resulting from the seismic impact:
$V_x = A * S * D * I * W / K_x$
$V_y = A * S * D * I * W / K_y$
where:
user-defined values:
I – structure class coefficient
K – behavior factor ($K_x$ for the X direction, $K_y$ for the Y direction)
values calculated by the program:
A - nominal acceleration
S – site coefficient
D - increase coefficient calculated from the vibration period (RPS2000 table 5.5)
W - total weight of the structure
$W = G + \psi Q$
dead loads and part of live loads, RPS2000 formula 6.2, where $\psi$ - load factor

c) Distribution of the force V ($V_x$ for the X direction, $V_y$ for the Y direction) onto individual stories for Tx > 0.7 (s)
- at the roof level, the following force is acting additionally:

$$F_{Tx} = 0{,}07 * T_x V$$

analogously, the formula holds for the Y direction, where V ($V_x$ for the X direction, $V_y$ for the Y direction) is the total seismic force
- on individual stories – the forces:

$$F_i = \frac{(V - F_t) * W_i * h_i}{\sum_{j=1}^{n} W_j * h_j}$$

where:
Fi - horizontal force on the story *i*

hi - level of the slab to which the force $F_i$ is applied
hj - level of individual stories
wi, wj - weights corresponding to the stories *i, j*

d) displacements of the story *i* for individual directions X and Y are determined based on the calculations of the cantilever loaded with forces Fi at the story levels - for membrane and mixed buildings, whereas for frame buildings – with the use of the equivalent frame method.

### 7.4.7. Default Loads – Simplified Method (UBC 97)

1. The scope of application of the simplified method of the static transverse force
all regular and non-regular structures for the zones:
- seismic zone 1
- seismic zone 2: category 4 and 5
regular structures with the maximum height 240 feet (73.152 m), that have a system of transverse stiffeners according to table 16-N, except for the one listed in 1629.8.4-designation-4
non-regular – max. 5 stories or 65 feet of height (19.812 m)
structures composed of a flexible upper part and a rigid lower part
- both structure parts considered separately may be classified as regular
- average story stiffness in the lower part is at least 10 times greater than the average story stiffness in the upper part
the vibration period of the whole structure is 1.1 times greater than the vibration period of the upper part considered separately, with a fixed connection to the base

2. Design height of a building
the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Description of the simplified method of the static transverse force
Calculations should be performed separately for each of the directions X and Y

a) Total horizontal (seismic) force

$$V = \frac{C_V I}{R \cdot T} \cdot W$$

conditions for the force V:

$$0{,}11 \cdot C_a \cdot I \cdot W \le V \le \frac{2{,}5 \cdot C_a I}{R} \cdot W$$

and additionally for the zone 4

$$V \ge \frac{0{,}8 \cdot Z \cdot N_V I}{R} \cdot W$$

where:
V - $V_x$ for the X direction, $V_y$ for the Y direction
Ca, Cv, Na, Nv – factors conditioned by a zone and a soil profile
R – behavior factor ($R_x$ for the X direction, $R_y$ for the Y direction)
calculations for the soil SF are not performed by the program

b) structure vibration period

$$T = C_t \cdot (h_n)^{3/4}$$

where Ct – coefficient determined by the user ($C_{tx}$ for the X direction, $C_{ty}$ for the Y direction):
0.035 (0.0853) for structures made from steel frames
0.030 (0.0731) for structures made from RC frames
0.020 (0.0488) for other structures
a vibration period for the directions X and Y may be defined manually

c) Division of the horizontal seismic force into individual stories
If T > 0.7 second, then FT = 0.07 * T * V ≤ 0.25 * V
FT – an additional force on the highest story level

$$F_x = \frac{(V - F_t) \cdot w_x \cdot h_x}{\sum_{i=1}^{n} w_i \cdot h_i}$$

where Fx – forces on the levels of individual stories

d) Calculation of displacements of individual stories.
Displacements of the story i for individual directions X and Y are calculated based on the solution of the cantilever loaded with Fi forces on the story levels for membrane and mixed buildings, whereas for frame buildings – with the use of the equivalent frame method.
As a result, the user obtains values of static displacements ΔS which should be modified according to the formula (30-17) - see 1630.9.2

### 7.4.8. Default Loads - Simplified Method (IBC 2000/2006)

1. Scope of application
all regular structures with height up to 240 feet (73.152 m)

2. Design height of a building
the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Method of the equivalent transverse force - chapter 1617.4
Calculations should be performed separately for each of the directions X and Y
Calculations are not performed for the soil F, and also for the soil E if Ss ≥ 1.25 or S1 ≥ 0.5

a) Maximum seismic acceleration factors

$$S_{MS} = F_a S_S \qquad S_{M1} = F_v S_1$$

where Fa, Fv – location factors according to tables 1615.1.2(1-2)

b) Projected seismic acceleration factors

$$S_{DS} = \frac{2}{3} S_{MS} \qquad S_{D1} = \frac{2}{3} S_{M1}$$

c) Total horizontal (seismic) force
V = Cs * W
where:
V - $V_x$ for the X direction, $V_y$ for the Y direction
W – total vertical (dead + live) load on a whole structure
R – behavior factor ($R_x$ for the X direction, $R_y$ for the Y direction)
CS – seismic response factor

$$C_S = \frac{S_{DS}}{\left(\frac{R}{I_E}\right)}$$

conditions for CS for IBC 2006:
the parameter value should not exceed:

$$C_s = \frac{S_{D1}}{T\left(\frac{R}{I}\right)} \quad \text{for } T \leq T_L$$

$$C_s = \frac{S_{D1} T_L}{T^2\left(\frac{R}{I}\right)} \quad \text{for } T > T_L$$

yet it should not be less than CS = 00.1, while for buildings for which S1 ≤ 0.6 * g:

$$C_s = \frac{0.5 S_1}{\left(\frac{R}{I}\right)}$$

where T is a basic structure vibration period

d) Total vibration period of a structure
for IBC 2000

$$T = C_t \cdot (h_n)^{3/4}$$

where Ct - factor defined by the user ($C_{tx}$ for the X direction, $C_{ty}$ for the Y direction):
0.035 (0.0853) for structures made from steel frames
0.030 (0.0731) for structures made from RC frames and eccentricity-braced steel frames
0.020 (0.0488) for other structures
a vibration period for the directions X and Y may be defined manually

for IBC 2006:

$$T = C_t \cdot (h_n)^x$$

where:
Ct - coefficient conditioned by a structure type (table 12.8-2 ASCE 7-05)
x - coefficient adopted from table 12.8-2 (ASCE 7-05) - set by the program automatically for defined Ct and structure types:
Steel frames Ct = 0.028 (0.0724) x=0.8
RC frames Ct = 0.016 (0.0466) x=0.9
Eccentricity-braced steel frames Ct = 0.03 (0.0731) x=0.75
Remaining buildings Ct=0.02 (0.0488) x=0.75

e) Division of the horizontal force into stories

$$F_x = C_{vx} \cdot V$$

where:
Fx – a horizontal force applied to the story x
Cvx – coefficient of force division into stories

$$C_{vx} = \frac{w_x \cdot h_x^k}{\sum_{i=1}^{n} w_i \cdot h_i^k}$$

where: k – period-dependent power exponent:

| | |
|---|---|
| T ≤ 0.5 s | k = 1.0 |
| 0.5s < T ≤ 2.5 s | is a number interpolated linearly, from the interval <1, 2> |
| 2.5 s ≤ T | k = 2.0 |

hi, hx – height measured from the building base to the levels i, x
V – a total horizontal force due to seismic load
wi, wx – a part of the total vertical load W (dead + live) on a whole building applied to the stories i, x

f) Calculation of displacements of individual stories.

Displacements of the story i for individual directions X and Y are calculated based on the solution of the cantilever loaded with Fi forces on the story levels - for membrane and mixed buildings, whereas for frame buildings – with the use of the equivalent frame method.
As a result, the user obtains values of static displacements Δ which should be modified according to IBC2000 / 1617.4.6.1.

g) Division of a transversal force to elements of a story
In proportion to rigidity

## 7.4.9. Default Loads - Simplified Method (Italian Seismic Code)

**1.** Scope of application

The static - linear analysis may be performed for horizontally-projected, regular structures (point 4.3), considering two separate 2D models, as in point 4.4., on the condition that the first structure vibration period (T1) is not greater than 2.5TC.

2. Design height of a building

the above-ground height of the building is adopted (the program takes no account of the underground stories)

3. Method of the static - linear analysis

calculations should be performed separately for each of the directions (X and Y)

a) structure vibration period

$$T_1 = C_1 \cdot H^{3/4}$$

(the code assumes that H ≤ 40m)
where:
C1 - user-defined coefficient ($C_{1x}$ for the X direction, $C_{1y}$ for the Y direction):
0.085 for steel frame structures
0.075 for RC frame structures
0.050 for other structures
H - total height of the building from the foundation level

b) Total horizontal (seismic) force

- for the elastic spectrum $F_h = S_e(T_1) \cdot W \cdot \lambda$
- for the design spectrum $F_h = S_d(T_1) \cdot W \cdot \lambda$

where:
$F_h$ - $F_{hx}$ for the X direction, $F_{hy}$ for the Y direction
Se (T1), Sd (T1) - coordinate of the elastic (design) response spectrum (see the formulas - points 3.2.3, 3.2.5) for individual directions
W - total structure weight (calculated as for the remaining codes)
λ = 0.85, if the number of stories equals at least 3 and T1<2*TC
1.0 – for others

c) Division of the horizontal seismic force into individual stories

$$F_i = F_h \cdot \frac{\sum_i z_i \cdot W_i}{\sum_j z_j \cdot W_j}$$

where:
Fi - horizontal force acting on the story i
Wi, Wj - values of the weight of the stories i and j
zi, zj - values of the height of the stories i and j (from the foundation level)

d) Calculation of displacements of individual stories

Displacements of the story i for individual directions X and Y are calculated based on the solution of the cantilever loaded with Fi forces on the story levels - for membrane and mixed buildings, whereas for frame buildings – with the use of the equivalent frame method.

e) Division of horizontal forces into individual constructional elements of a story
Horizontal forces are divided in proportion to the stiffness (as for other seismic codes).

### 7.4.10. Default Loads - Simplified Method (P100-1/2006)

1. The scope of application of the simplified method: for buildings that satisfy code requirements

2. Description of the method:

a. evaluation of T1 (basic vibration period)

$$T_1 = C_t H^{3/4}$$

where:
T1 - approximate vibration period for individual directions
Ct - coefficient conditioned by a structure type (defined by the user)

b. evaluation of the elastic spectrum for the defined quantities TB, TC, TD and the spectrum ordinate for T1

$$T \le T_B \qquad \beta(T) = 1 + \frac{(\beta_0 - 1)}{T_B} T \qquad (3.2)$$

$$T_B < T \le T_C \qquad \beta(T) = \beta_0 \qquad (3.3)$$

$$T_C < T \le T_D \qquad \beta(T) = \beta_0 \frac{T_C}{T} \qquad (3.4)$$

$$T > T_D \qquad \beta(T) = \beta_0 \frac{T_C T_D}{T^2} \qquad (3.5)$$

β(T) - normalized elastic spectrum
β0 - maximal value of the dynamic amplification factor for the horizontal seismic acceleration
T - basic structure vibration period

| | Values of periods [sec.] | | |
|---|---|---|---|
| TB | 0.07 | 0.1 | 0.16 |
| TC | 0.7 | 1.0 | 1.6 |
| TD | 3 | 3 | 2 |

c. evaluation of the design spectrum

$$0 < T \le T_B \qquad S_d(T) = a_g \left[ 1 + \frac{\frac{\beta_0}{q} - 1}{T_B} T \right] \qquad (3.17)$$

$$T > T_B \qquad = a_g \frac{\beta(T)}{q} \qquad (3.18)$$

d. base shear force
Fb = γ1 Sd(T1) m λ

where:
$Sd(T1)$ - spectrum ordinate for the basic period T1
m - total structure mass
$\gamma 1$ - structure factor
$\lambda$ - correction factor that equals:
0.85 for T1< Tc
1.0 in the remaining cases

e. division of forces onto individual stories

$$F_i = F_b \frac{m_i z_i}{\sum_{i=1}^{n} m_i z_i}$$

where:
zi - height of the story i with reference to the base of the building
mi - mass of the story i.

### 7.4.11. Default Loads - Seismic Analysis (Advanced Method)

When the advanced method is selected, the following options are available in the ***Default Loads / Seismic*** dialog box:
- combination of vibration modes (CQC, SRSS)
The modal response is calculated from the formula below:

$$R_{\max} = \sqrt{\sum_{i=1}^{n}\sum_{j=1}^{n} e_{ij} R_i R_j}$$

where:
n – number of modes
eij – coupling (correlational) coefficients
Ri, Rj – spectral responses for the modes 'i' and 'j'

The following types of quadratic combinations are accessible in the program:

**SRSS method**
For the SRSS method the correlational coefficients equal:
eij = 1 for i=j,
eij = 0 for $i \neq j$,
therefore:

$$R_{\max} = \sqrt{\sum_{i=1}^{n} R_i^2}$$

**CQC method**
For the CQC method the correlational coefficients are calculated from the formula below:

$$eij = \frac{8\sqrt{\zeta_i \zeta_j}\left(\zeta_i + r\zeta_j\right)r^{1.5}}{\left(1-r^2\right)^2 + 4\zeta_i\zeta_j r\left(1+r^2\right) + 4\left(\zeta_i^2 + \zeta_j^2\right)r^2}$$

where:
$\zeta i$, $\zeta j$ – damping coefficients for the modes 'i' and 'j' (relative values)
$r = Tj/Ti \leq 1$
Tj, Ti – vibration periods for the modes 'i' and 'j'.

The formula above is used in the program for the PS92 code, if the *Damping as for PS92* option is switched on in the dialog box with seismic analysis parameters (for the PS92 code). If this option is switched off, one damping value is applied to all modes and the above formula assumes the following form:

$$eij = \frac{8\zeta^2(1+r)r^{1.5}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2}$$

- combinations of seismic directions:

quadratic

according to the formula (the user defines Rx, Ry, Rz)

$$S = \sqrt{R_X S_X^2 + R_Y S_Y^2 + R_Z S_Z^2}$$

Newmark (according to PS 92 chapter 6.4)

Group 1 $S = \pm S_X \pm \lambda S_Y \pm \mu S_Z$

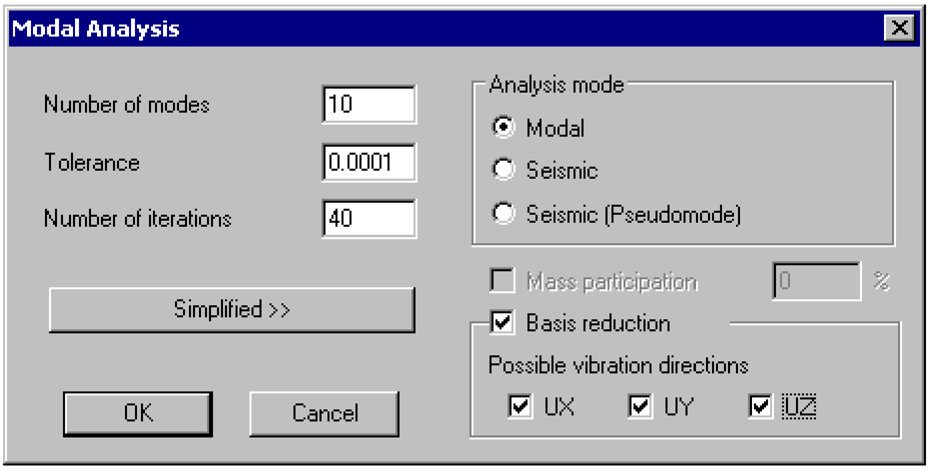
Group 2 $S = \pm \lambda S_X \pm S_Y \pm \mu S_Z$

Group 3 $S = \pm \lambda S_X \pm \mu S_Y \pm S_Z$

where: Sx, Sy, Sz – are the results of the automatically generated cases on the directions X, Y, Z, respectively, calculated according to the combinations SRSS or CQC, whereas $\lambda$ and $\mu$ are decrease coeficients (equal at most to 0.4).
Newmark combinations are displayed in the ***Combinations*** dialog box, on the tab with accidental combinations (ALS).

### 7.4.12. Default Loads - Seismic Analysis (Modal Analysis)

The dialog box below opens on pressing the **Modal analysis** button in the ***Default Loads / Seismic*** dialog box.




The following options are available in the dialog box above:

- *Number of modes* – a number of eigenmodes to be calculated by the program
- *Tolerance* – a value that determines the precision (the criterion of stopping the iterations)
- *Number of iterations* – the maximum allowable number of iterations for each mode (the iteration process for each mode is stopped when the current value of the tolerance is less than the value provided in the *Tolerance* edit field or when the current number of iterations exceeds the specified number of iterations).
-

Pressing the **Advanced >>** button opens an additional part of the dialog box for definition of the following:

1. analysis mode
   - modal

- seismic
- seismic (Pseudomode)

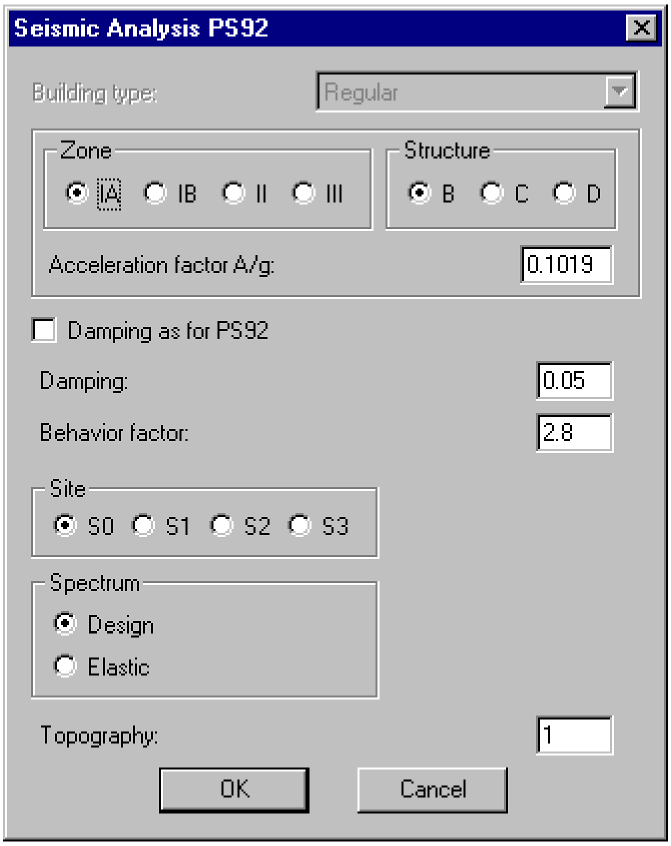
2. Mass participation - the option is active for the seismic and the seismic (Pseudomode) modes; it allows implementing a limit of the percent mass participation
3. Basis reduction - the option is active for the modal mode of the analysis; the list of nodes includes all corner nodes of objects and the vibration directions set (UX, UY, UZ) apply to all nodes.

The consequences of selecting individual analysis modes are a follows;
when the seismic or the seismic (Pseudomode) mode is selected, the following analysis options will be set:

- calculation method - Lanczos method
- 'Sturm check" will be switched off
- 'Disregard density' will be switched off
- mass matrix will be switched to 'Lumped with rotations' instead of 'Lumped without rotations'.

### 7.4.13. Default Loads - Seismic Analysis (PS92)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code PS92 is selected) in the ***Default Loads / Seismic*** dialog box.



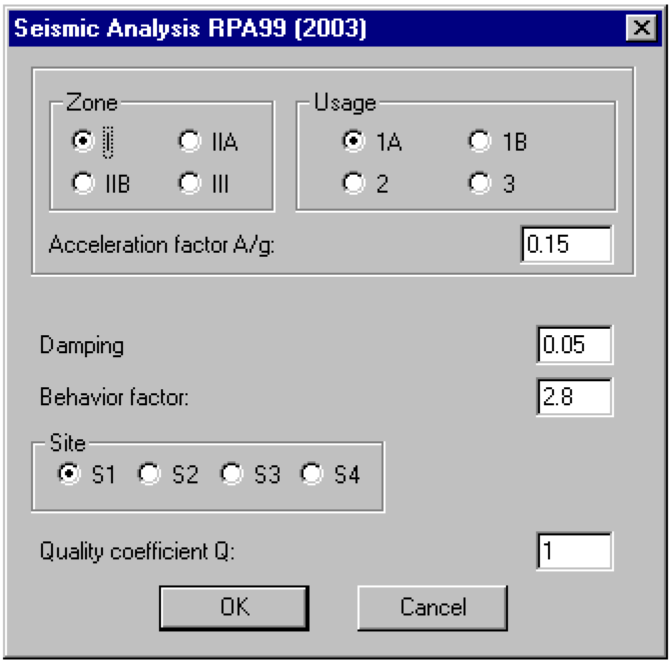

The following options described in the selected seismic code are available in the dialog box above:

- building type (regular, medium-regular)
- seismic zone

- structure
- acceleration factor: the ratio A / g (the seismic acceleration to the acceleration of gravity; the edit field shows the code-specified value of the acceleration factor for selected parameters)
- damping as for PS92 – the option is significant only for the advanced method
- damping – a value of the damping coefficient (for the simplified method: for the X and Y directions)
- site
- spectrum type: design or elastic; if the elastic spectrum is chosen, the behavior factor is disregarded (NOTE: for the simplified method only the design spectrum is accessible)
- topography
- behavior factor for the X and Y directions (it accounts for the manner of the material behavior).

### 7.4.14. Default Loads - Seismic Analysis (RPA99_03)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code RPA99_03 is selected) in the ***Default Loads / Seismic*** dialog box.



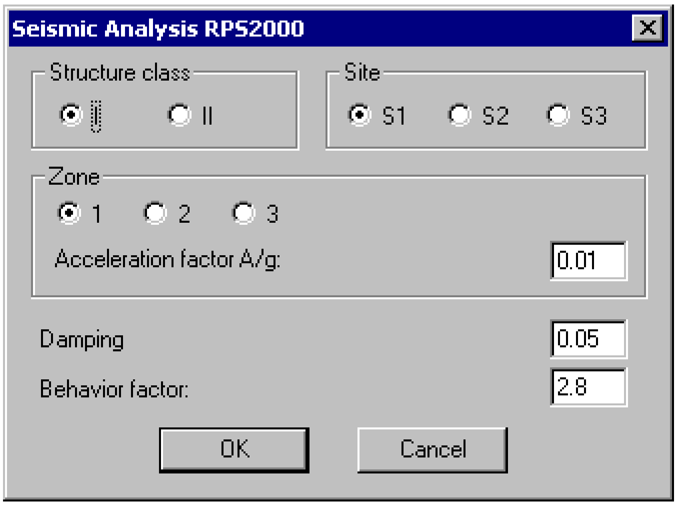

The following options described in the selected seismic code are available in the dialog box above:

- seismic zone
- usage
- acceleration factor: the ratio A / g (the seismic acceleration to the acceleration of gravity; the edit field shows the code-specified value of the acceleration factor for selected parameters)

- damping – a value of the damping coefficient (for the simplified method: for the X and Y directions)
- site
- quality coefficient Q
- behavior factor for the X and Y directions (it accounts for the manner of the material behavior).

### 7.4.15. Default Loads - Seismic Analysis (RPS2000)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code RPS2000 is selected) in the ***Default Loads / Seismic*** dialog box.

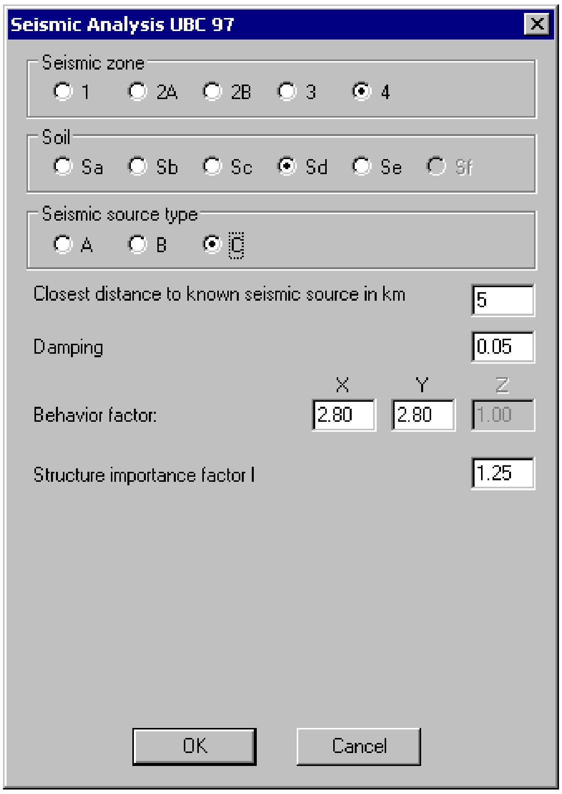



The following options described in the selected seismic code are available in the dialog box above:

- structure class
- site
- seismic zone
- acceleration factor: the ratio A / g (the seismic acceleration to the acceleration of gravity; the edit field shows the code-specified value of the acceleration factor for selected parameters)
- damping – a value of the damping coefficient
- behavior factor for the X and Y directions (it accounts for the manner of the material behavior).

### 7.4.16. Default loads - Seismic Analysis (UBC 97)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code UBC 97 is selected) in the ***Default Loads / Seismic*** dialog box.



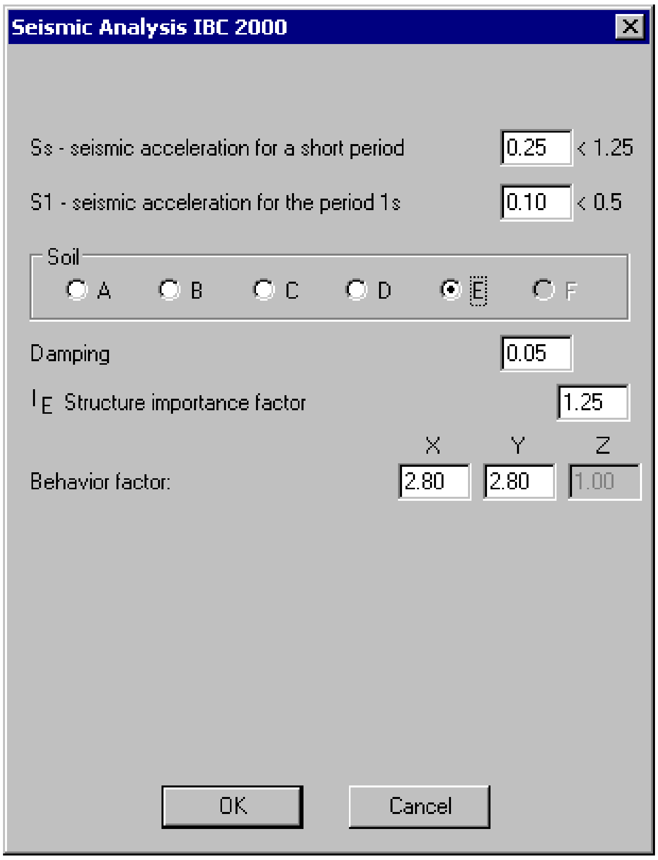

The following options described in the selected seismic code are available in the dialog box above:

- seismic zone
- soil
- seismic source type (active for zone 4)
- the closest distance to the seismic source (active for zone 4)
- importance factor
- damping – a damping coefficient value (considered in advanced calculations)
- behavior factor for the X, Y and Z directions (considers material behavior)
- CT coefficient for the X and Y directions for calculation of the approximated period.

The code UBC97 assumes the following coefficient values:
- 0.035 (0.0853 – in the metric system) – steel frames
- 0.030 (0.0731) – RC frames and eccentricity-braced steel frames
- 0.020 (0.0488) – other structures.

## 7.4.17. Default Loads - Seismic Analysis (IBC 2000)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code IBC 2000 is selected) in the ***Default Loads / Seismic*** dialog box.




In the upper part of the dialog box is the option *Consider the minimum lateral forces for the category A*.
If this option is switched on, it allows performing simplified calculations considering the minimal lateral forces as for the seismic design category A (according to the requirements of the IBC2000 code, point 1616.4.1); after this option is switched on, the seismic parameters usually defined for the IBC2000 code are not accessible. For each of the directions X and Y, the lateral force on the story 'x' equals $F_x = 0.01 * W_x$, where $W_x$ is the weight ascribed to the story 'x'.
Calculations are performed analogously as for the simplified method according to IBC2000 after distribution of forces.

If the *Consider the minimum lateral forces for the category A* option is switched off, the following options described in the selected seismic code are available in the dialog box above:

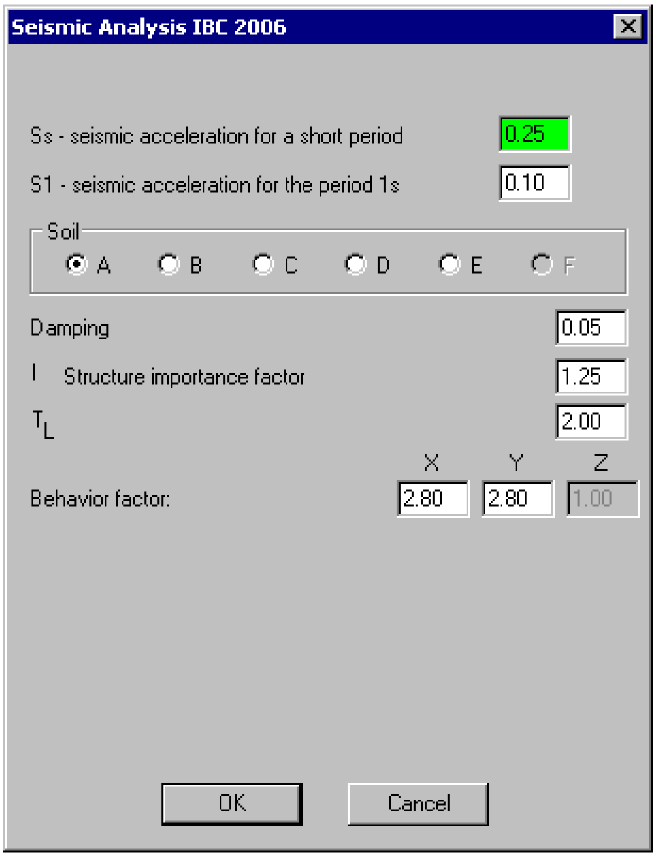
- seismic acceleration for a short period (0.2s) and period 1s
- seismic zone
- soil
- structure importance factor Ie
- damping – a damping coefficient value
- behavior factor for the X, Y and Z directions (considers material behavior)

- CT coefficient for the X and Y directions for calculation of the approximated period.

The code IBC2000 assumes the following coefficient values:
- 0.035 (0.0853 - in the metric system) – steel frames
- 0.030 (0.0731) - RC frames
- 0.020 (0.0488) - other structures.

### 7.4.18. Default Loads - Seismic Analysis (IBC 2006)

Below is the dialog box that opens after pressing the **Seismic analysis** button (the IBC 2006 seismic code is chosen) in the ***Default Loads / Seismic analysis*** dialog box.




The above dialog box holds the following options described in the selected seismic code:

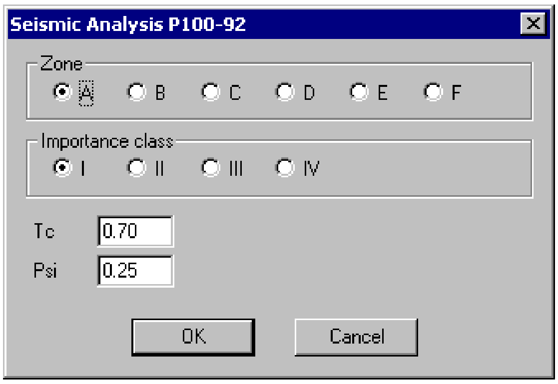
- seismic acceleration for a short period (0.2s) and a period of 1s
- seismic zone
- soil type
- structure importance factor Ie
- damping - value of a damping coefficient
- behavior factor for the directions X, Y and Z (takes into account how a material behaves)
- T - period defined by the user
- TL - value of a long-term period
- $C_T$ coefficient for the directions X and Y for calculations of an approximate period.

IBC 2006 assumes the following coefficient values:

- 0.035 (0.0853 - in the metric system) - steel frames
- 0.030 (0.0731) - RC frames
- 0.020 (0.0488) - other structures.

## 7.4.19. Default Loads - Seismic Analysis (P100-92)

The dialog box below opens on pressing the **Seismic analysis** button (if the seismic code P100-92 is selected) in the ***Default Loads / Seismic*** dialog box.

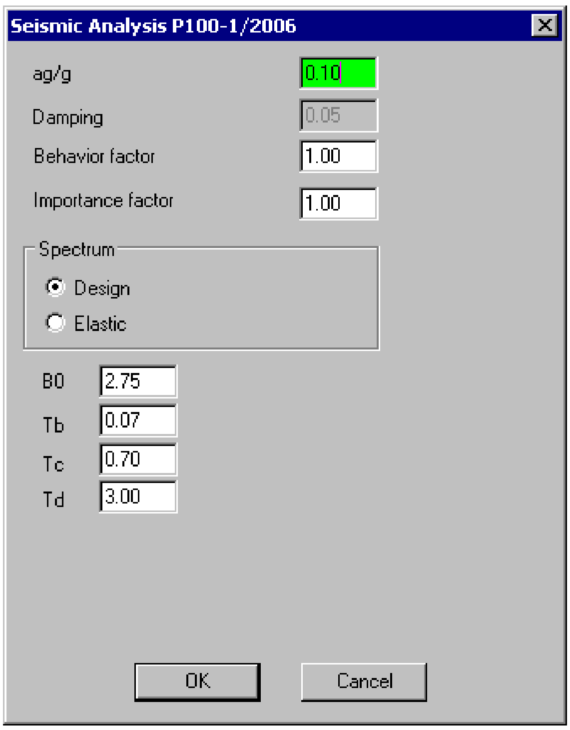



The following options described in the selected seismic code are available in the dialog box above:

- seismic zone
- importance class
- values of the coefficients: Tc and Psi.

## 7.4.20. Default Loads - Seismic Analysis (P100-1/2006)

Below is the dialog box that opens after pressing the **Seismic analysis** button (the P100-1/2006 seismic code is chosen) in the ***Default Loads / Seismic analysis*** dialog box.



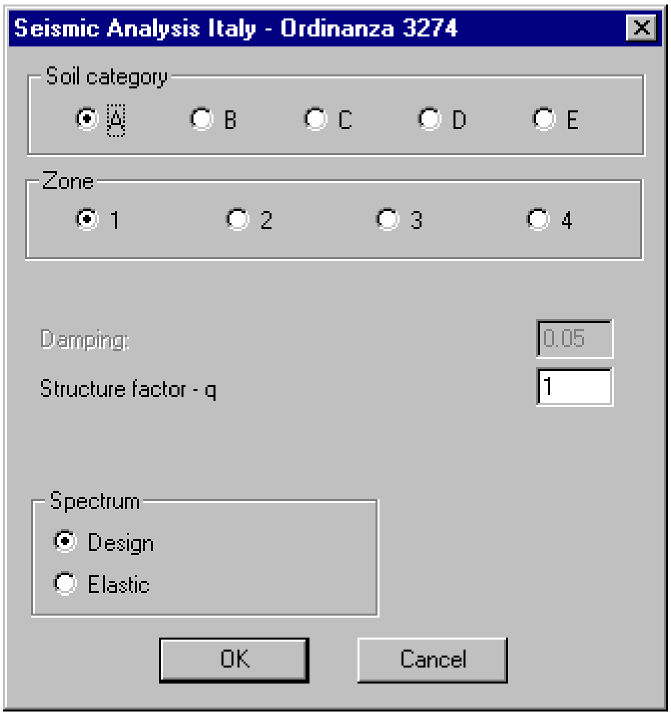
The above dialog box holds the following options described in the selected seismic code:

- ag / g - quotient of spectral to gravitational acceleration for a given position
- damping (the edit field is not available; the value 0.05 has been adopted)
- values of the behavior factor q (the factor available for a design spectrum) and the importance factor $\gamma$I
- spectrum type (design, elastic)
- amplification factor $\beta$0
- values of individual periods characteristic of a spectrum: TB, TC, TD.

Moreover, the Ct coefficient (the coefficient for evaluation of a basic vibration period for the direction X and the direction Y) can be defined for the simplified method.
Ct = 0.085 - steel frames
Ct = 0.075 - RC frames
Ct = 0.05 - remaining structures;
H - building height (measured from the foundation or from the rigid ground).

### 7.4.21. Default Loads – Italian Seismic Code

Below is the dialog box that opens after pressing the **Seismic analysis** button (Italian seismic code is chosen on the seismic code list) in the ***Default Loads / Seismic analysis*** dialog box.




The above dialog box holds the following options described in the selected seismic code:

- soil category
- seismic zone
- damping – value of the damping coefficient (accessible depending on the chosen spectrum type)

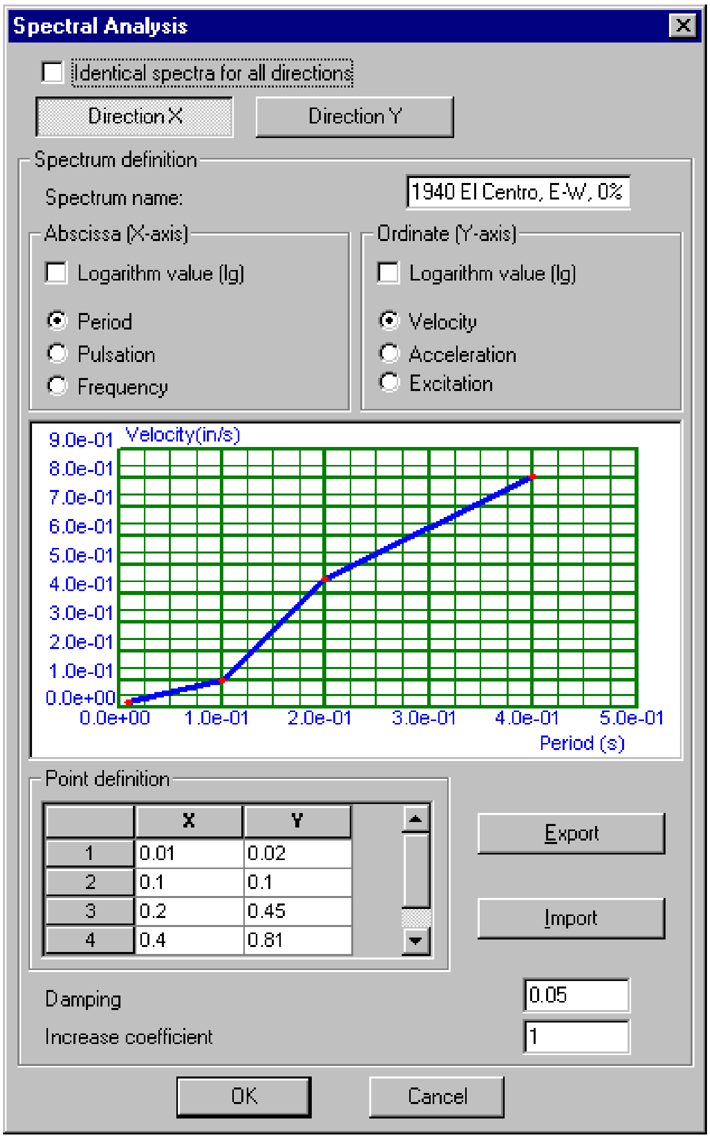
- structure factor q for the directions X and Y (accessible depending on the chosen spectrum type)
- $C_1$ coefficient for the directions X and Y for calculation of an approximate structure vibration period (user-defined value)

  0.085 - steel frames
  0.075 - RC frames
  0.05 - other structures

- spectrum type: design or elastic.

### 7.4.22. Default Loads - Seismic Analysis (Spectral Analysis)

Below is the dialog box that opens after pressing the **Spectral analysis** button (spectral analysis must be chosen on the seismic code list) in the ***Default Loads / Seismic analysis*** dialog box.




The above dialog box is used to define parameters of spectral analysis for a new dynamic case. Depending on a selected calculation method in the program it is possible to generate spectral analysis cases for 2 directions (X, Y, if a structure is calculated with the use of simplified methods) or for 3 directions (if a structure is calculated using advanced methods).

Spectra can be defined for individual directions; they may be defined as a function of velocity, acceleration or excitation, in relation to a period, pulsations or frequency.

In the upper part of the dialog box there are options for selection of a direction of spectral/seismic excitation. The following situations are possible:

- after pressing the **Direction X** button, the direction of spectral excitation will be the direction of the X axis of the global coordinate system
- after pressing the **Direction Y** button, the direction of spectral excitation will be the direction of the Y axis of the global coordinate system
- after pressing the **Direction Z** button, the direction of spectral excitation will be the direction of the Z axis of the global coordinate system (available only for the advanced method)
- after switching on the *Identical spectra for all directions* option, spectral excitation will be defined for both directions (i.e. in the direction of the axes X and Y of the global coordinate system).

To define a spectrum, follow the steps below:

- specify the spectrum name in the *Spectrum name* edit field
- select quantities whose values will be used to define the spectrum; it is possible in the respective fields: *Abscissa (X-axis)* and *Ordinate (Y-axis)*: the abscissa axis is used to define values of a period, frequency or pulsations, whereas the ordinate axis - values of acceleration, velocity or excitation (values shown on both axes may be also presented in the logarithmic scale, i.e. the axes represent logarithms of selected quantities)
- specify successive points of the spectrum (enter values for selected quantities in the *Point definition* table)
- determine a damping value
- determine a value of the increase coefficient Ci (available only for the simplified method).

In the middle part of the dialog box is the field with a coordinate system in which the defined or modified spectrum is presented.

Spectra defined by the user can be saved to files; to do it, press the **Export** button and specify a file name (with the *.spe extension) and the file location on the disk.
A spectrum can be also loaded from a *.spe extension file; to do it, press the **Import** button and indicate an appropriate file.

*NOTE:* *When a spectrum is imported from a file created in the* **ROBOT** *program, an additional dialog box opens on the screen showing a list of spectra defined in the selected file.*

In the program, spectral analysis may be performed using the following methods:

- simplified
- advanced (the program assumes in calculations values determined in the dialog boxes for defining parameters of seismic analysis according to a selected seismic code or in the dialog box for defining parameters of spectral analysis).

### 7.4.23. Default Loads - Spectral Analysis (Simplified Method)

The pseudostatic method for regular buildings from the French seismic code PS 92 has been used for calculation of forces and displacements of stories. Calculations in the program are performed separately for individual directions X and Y.

Individual calculation steps look analogously as for PS-92.

a) basic vibration periods (for the directions X and Y) – depending on a selected structure type:

frame

$$T_x = 0{,}1\frac{H}{\sqrt{L_x}} \qquad T_y = 0{,}1\frac{H}{\sqrt{L_y}}$$

membrane and mixed structure

$$T_x = 0{,}08\frac{H}{\sqrt{L_x}}\sqrt{\frac{H}{L_x+H}} \qquad T_y = 0{,}08\frac{H}{\sqrt{L_y}}\sqrt{\frac{H}{L_y+H}}$$

b) Equivalent static forces fr and displacements of individual stories dr for the direction X

$$f_{r,x} = \rho_{0x} m_r Z_r^{\alpha}\frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}}\cdot R_x(T_x) \qquad d_{r,x} = R_x(T)\rho_{0x} Z_r^{\alpha}\frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}}\left(\frac{T_x}{2\pi}\right)^2$$

where:

values calculated by the program:

ρ0x - increase coefficient (taking account of the ignored modes)

$$\rho_{0x} = 1+0{,}05\left(\frac{T_x}{T_C}\right)^{3/2} \geq 1{,}05 \quad \text{for frames}$$

$$\rho_{0x} = 1+0{,}10\left(\frac{T_x}{T_C}\right)^{3/2} \geq 1{,}10 \quad \text{for membranes and mixed structures}$$

TC - the maximum period value for the maximal value a(T), and in the case a spectrum is defined based on other values, this quantity is adopted in the program after being converted to the relation a(T)

mr - mass of the considered story r – here are adopted all dead and live loads applied to the floor slab (the top of a given story), multiplied by participation factors (determined in the dialog box), as well as the self-weight of horizontal elements and half the weight of the vertical elements of a given story and of the next story (*r+1*).

In the case of the story 0, the whole weight of vertical elements of the story 0 is included.

If the soil level is higher than the level of the base of the story 0, then the weight and the loads of the stories 0 and 1 are included in the story 2.

zr - quotient of the story height and the design height of the building Zr = hr / H, where

H – building height counted from the base of the story *0* to the level of the story n

α - coefficient dependent on the bracing of the building (frames α = 1,0, walls α = 1.5)

spectral acceleration

$$R_x(T_x) = C_i \rho R_D(T_x)$$

instead of the quantities aN and τ from the PS 92 code, the increase coefficient Ci has been introduced

$$\rho = \left[\frac{5}{\zeta}\right]^{0,4}$$

where $\zeta$ is a damping coefficient expressed in (%)
RD(Tx) - ordinate of the normalized design spectrum, defined earlier for the direction X, in the case a spectrum is defined using a relation other than a(T), in the stage of calculations the program converts the spectrum definition and takes account of the value a(Tx)

for the Y direction the procedure looks analogously as for the X direction

$$f_{r,y} = \rho_{0y} m_r Z_r^{\alpha} \frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}} \cdot R_y(T_y) \quad d_{r,y} = R_y(T)\rho_{0y} Z_r^{\alpha} \frac{\sum m_i Z_i^{\alpha}}{\sum m_i Z_i^{2\alpha}} \left(\frac{T_y}{2\pi}\right)^2$$

## 7.4.24. Seismic / Spectral Analysis Considering the Torsion Effect

SIMPLIFIED CALCULATIONS
In the program simplified calculations may be performed in 3 ways:

- without considering the torsion effect
- considering normal torsion (considering of the theoretical eccentricity, i.e. the distance between the mass centroid and the center of torsion for individual stories of a building)
- considering accidental torsion (additionally, the not-intended eccentricity of the mass centroid location is taken into account; code values, and in the case of spectral analysis – user-defined values, are adopted).

**Simplified calculations - procedure**

The following assumptions have been adopted:

- when calculating the mass centroid of a story, the story weight evaluated in such a way as during seismic calculations is considered
- loads are applied to the ceiling slab of a given story
- walls do not carry loads perpendicular with respect to their plane (therefore, their stiffness in the direction perpendicular to the axis equals zero).

(1) Evaluation of the mass centroid G (xG,r, yG,r ) of the story *r*

$$x_{G\,r} = \frac{\sum_i m_{r,i} \cdot x_{r,i} + \sum_j \frac{w_{r,j}}{g} \cdot x_{w\,r\,j} + \sum_k \frac{q_{r,k} \cdot \psi_k}{g} \cdot x_{q\,r\,k}}{\sum_i m_{r,i} + \sum_j \frac{w_{r,j}}{g} + \sum_k \frac{q_{r,k} \cdot \psi_k}{g}}$$

where:
xG,r – coordinate x of the mass centroid G
mr,i - mass of a structure element belonging to the story r; how to determine which elements belong to which stories:
- horizontal elements: belong to current stories
- vertical elements:
the last story: 1 of the mass
the first story (0): mass of elements on the story 0 + 1 of the mass on the story 1
intermediate stories: 1 of the mass of the stories i and i+1
wr,j - dead load acting on a structure within the story r

qr,k - variable load (live, wind, snow) acting on a structure within the story r
ψk – load conversion factor for seismic analysis
g – acceleration of gravity
xr,i – coordinate x of the center of gravity of the element i
xw,r,,j - coordinate x of the center of gravity of the dead load j
xq,r,k - coordinate x of the center of gravity of the variable load (live, snow, accidental) k.
The coordinate below is determined analogously

$$y_{G,r} = \frac{\sum_i m_{r,i} \cdot y_{r,i} + \sum_j \frac{w_{r,j}}{g} \cdot y_{w,r,j} + \sum_k \frac{q_{r,k} \cdot \psi_k}{g} \cdot y_{q,r,k}}{\sum_i m_{r,i} + \sum_j \frac{w_{r,j}}{g} + \sum_k \frac{q_{r,k} \cdot \psi_k}{g}}$$

(2) Evaluation of the center of torsion T of the story *r*

(a) moments of inertia in local systems
For all grouped walls properties of a compound section are calculated
structure walls and columns that carry horizontal loads are taken into consideration
For each wall the following values are calculated:

$J'_{x,r,i} = \frac{b_i \cdot h_i^3}{12}$ - greater moment of inertia in the local system of the wall (in the direction parallel to the wall)

$J'_{y,r,i} = 0$ - smaller moment of inertia in the local system of the wall (in the direction perpendicular to the wall)

$\vartheta_{r,i}$ – angle between the axis parallel to the wall (along the length) and the Y axis

For each column the following values are calculated:

$J'_{x,r,i} = \frac{b_i \cdot h_i^3}{12}$ - greater moment of inertia in the local system of the column

$J'_{y,r,i} = \frac{h_i \cdot b_i^3}{12}$ - smaller moment of inertia in the local system of the column

$\vartheta_{r,i}$ – angle between the Y' axis (of the local system) and the Y axis (of the global system)

(b) moments of inertia in the global system

$$J_{x,r,i} = J'_{x,r,i} \cdot \cos^2\theta_i + J'_{y,r,i} \cdot \sin^2\theta_i$$

$$J_{y,r,i} = J'_{x,r,i} \cdot \sin^2\theta_i + J'_{y,r,i} \cdot \cos^2\theta_i$$

(c) product of inertia in the global system

$$J_{xy,r,i} = (J'_{x,r,i} - J'_{y,r,i}) \cdot \cos\theta_i \cdot \sin\theta_i$$

(d) centers of torsion of each of the elements are their geometrical centers xO,r,i and yO,r,i
except grouped walls, for which centers of torsion are determined according to the open thin-walled member theory

(e) auxiliary values

$$P_{x,r} = \sum J_{y,r,i}$$
$$P_{y,r} = \sum J_{xy,r,i}$$
$$Q_{x,r} = \sum J_{xy,r,i}$$
$$Q_{y,r} = \sum J_{x,r,i}$$
$$M_{P,r} = \sum (J_{xy,r,i} \cdot x_{O,r,i} - J_{y,r,i} \cdot y_{O,r,i})$$
$$M_{Q,r} = \sum (J_{x,r,i} \cdot x_{O,r,i} - J_{xy,r,i} \cdot y_{O,r,i})$$

(f) coordinates of the center of torsion

- if:

$$-Q_{x,r} \cdot P_{y,r} + Q_{y,r} \cdot P_{x,r} \neq 0$$

$$x_{t,r} = \frac{-M_{P,r} \cdot Q_{x,r} + M_{Q,r} \cdot P_{x,r}}{-Q_{x,r} \cdot P_{y,r} + Q_{y,r} \cdot P_{x,r}} \qquad y_{t,r} = \frac{M_{P,r} \cdot Q_{y,r} - M_{Q,r} \cdot P_{y,r}}{Q_{x,r} \cdot P_{y,r} - Q_{y,r} \cdot P_{x,r}}$$

- if:

$$\sum J_{x,r} \neq 0 \; ; \sum J_{y,r} = 0$$

$$x_{t,r} = \frac{\sum E \cdot J_{x,i,r} \cdot x_{0,i,r}}{\sum J_{x,i,r}} \qquad y_{t,r} = \frac{\sum A_{i,r} \cdot y_{0,i,r}}{\sum A_{i,r}}$$

where Ai is the wall (column) area

- if:

$$\sum J_{x,r} = 0 \; ; \sum J_{y,r} \neq 0$$

$$x_{t,r} = \frac{\sum A_{i,r} \cdot x_{0,i,r}}{\sum A_{i,r}} \qquad y_{t,r} = \frac{\sum E \cdot J_{y,i,r} \cdot y_{0,i,r}}{\sum J_{y,i,r}}$$

- if:

$$\sum J_{x,r} > 0 \; ; \sum J_{y,r} > 0$$

$$x_{t,r} = \frac{\sum E \cdot J_{x,i,r} \cdot x_{0,i,r}}{\sum J_{x,i,r}} \qquad y_{t,r} = \frac{\sum E \cdot J_{y,i,r} \cdot y_{0,i,r}}{\sum J_{y,i,r}}$$

- in the remaining cases:

$x_{t,r} = 0$ , $y_{t,r} = 0$

(2a) Evaluation of the theoretical eccentricity e0r (e0rx, e0ry ): distance between G (CM) and T (CR), where G(CM) – mass centroid, and T (CR) - center of torsion, respectively for individual stories .

$$e_{0r,x} = x_{G,r} - x_{T,r} ; \qquad e_{0r,y} = -y_{G,r} + y_{T,r}$$

(3) Value of the torsional moment for individual seismic directions

$$M_{Tr,x} = V_{r,x} \cdot e_{0ry}$$
$$M_{Tr,y} = V_{r,y} \cdot e_{0rx}$$

(4) Evaluation of the accidental eccentricity

- PS92 code (France)

$e_{r,x,acc,1} = 0{,}1 \cdot L_x \cdot z_r$; $\quad e_{r,x,acc,2} = -0{,}1 \cdot L_x \cdot z_r$

$e_{r,y,acc,1} = 0{,}1 \cdot L_y \cdot z_r$; $\quad e_{r,y,acc,2} = -0{,}1 \cdot L_y \cdot z_r$

- RPS2000 code

$$e_{r,x,acc,1} = \max(0{,}5 \cdot e_{0r,x} + 0{,}05 \cdot L_x ; 0{,}05 \cdot L_x)$$
$$e_{r,x,acc,2} = \min(0{,}5 \cdot e_{0r,x} - 0{,}05 \cdot L_x ; -0{,}05 \cdot L_x)$$
$$e_{r,y,acc,1} = \max(0{,}5 \cdot e_{0r,y} + 0{,}05 \cdot L_y ; 0{,}05 \cdot L_y)$$
$$e_{r,y,acc,2} = \min(0{,}5 \cdot e_{0r,y} - 0{,}05 \cdot L_y ; -0{,}05 \cdot L_y)$$

- RPA99_2003 code

$$e_{r,x,acc,1} = \max\{0{,}05 \cdot \max\{L_x, L_y\}; e_{0r,x}\}$$
$$e_{r,x,acc,2} = \min\{-0{,}05 \cdot \max\{L_x, L_y\}; e_{0r,x}\}$$
$$e_{r,y,acc,1} = \max\{0{,}05 \cdot \max\{L_x, L_y\}; e_{0r,y}\}$$
$$e_{r,y,acc,2} = \min\{-0{,}05 \cdot \max\{L_x, L_y\}; e_{0r,y}\}$$

- the remaining codes (UBC97, IBC2000, Italian seismic code)

$$e_{r,x,acc,1} = 0{,}05 \cdot L_x ; e_{r,x,acc,2} = -0{,}05 \cdot L_x$$
$$e_{r,y,acc,1} = 0{,}05 \cdot L_y ; e_{r,y,acc,2} = -0{,}05 \cdot L_y$$

- spectral analysis

$e_{r,x,acc,1} = E\_x \cdot L_x$; $\quad e_{r,x,acc,2} = -E\_x \cdot L_x$

$e_{r,y,acc,1} = E\_y \cdot L_y$; $\quad e_{r,y,acc,2} = -E\_y \cdot L_y$

where the values E_x, E_y are defined by the user

(5) Torsional moment for the direction X

- RPA99_2003 code

$M_{Tr,x,1} = V_{x,r} \cdot e_{r,y,acc,1}$; $\quad M_{Tr,x,2} = V_{x,r} \cdot e_{r,y,acc,2}$

- the remaining codes and spectral analysis

$M_{Tr,x,1} = V_{x,r} \cdot (e_{r,y} + e_{r,y,acc,1})$; $\quad M_{Tr,x,2} = V_{x,r} \cdot (e_{r,y} + e_{r,y,acc,2})$

A value of the moment for the direction Y is determined analogously.

(5a) Adding up moments when passing to a lower story

Values of torsional moments evaluated in the way presented above are the moments resulting from forces acting on individual stories; moments that equal as follows should be assumed for distribution:

$$M_{Tr} = \sum_{i=r,\dots,n} M_{Ti}$$

(5b) Distance of the wall (column) centroid from the center of torsion in the wall (column) local system

- coordinates of the center of torsion in the local system of an object

$$x'_{iC,r} = (x_{T,r} - x_{Oi,r}) \cdot \cos\theta_i - (y_{T,r} - y_{Oi,r}) \cdot \sin\theta_i$$
$$y'_{iC,r} = (y_{T,r} - y_{Oi,r}) \cdot \cos\theta_i + (x_{T,r} - x_{Oi,r}) \cdot \sin\theta_i$$

- distance of the object centroid from the center of torsion of a story

$r_{xi,r} = y'_{iC,r} \qquad r_{yi,r} = -x'_{iC,r}$

(6) Polar moment of inertia of the story *r*

$$J_{pv,r} = \sum_i (r_{yi}^2 \cdot J'_{r,x,i} + r_{xi}^2 \cdot J'_{r,y,i})$$

(7) Radius of torsion

$$r_{r,x} = \sqrt{\frac{J_{pv,r}}{\sum_i J'_{r,x,i}}}, \qquad r_{r,y} = \sqrt{\frac{J_{pv,r}}{\sum_i J'_{r,y,i}}}$$

(8) Distribution of forces due to torsion

- due to normal torsion (normal torsion is selected – 2 seismic cases) – for individual directions, respectively, MTrx or MTry are substituted

  - walls

  $$R_{r,y,i}" = \frac{M_{Tr} \cdot r_{yi} \cdot J'_{r,x,i}}{J_{pv,r}}$$

  - columns

  X component:

  $$R_{r,x,i}" = \frac{M_{Tr} \cdot r_{xi} \cdot J'_{r,y,i}}{J_{pv,r}}$$

  Y component:

  $$R_{r,y,i}" = \frac{M_{Tr} \cdot r_{yi} \cdot J'_{r,x,i}}{J_{pv,r}}$$

- due to accidental torsion (accidental torsion is selected - 4 seismic cases)

  Seismic analysis X-1:

  - walls

  $$R_{r,y,i}" = \frac{M_{Tr,x1} \cdot r_{yi} \cdot J'_{r,x,i}}{J_{pv,r}}$$

  - columns

  X component:

  $$R_{r,x,i}" = \frac{M_{Tr,x1} \cdot r_{xi} \cdot J'_{r,y,i}}{J_{pv,r}}$$

  Y component:

  $$R_{r,y,i}" = \frac{M_{Tr,x1} \cdot r_{yi} \cdot J'_{r,x,i}}{J_{pv,r}}$$

  Seismic analysis X-2: $M_{Tr,x,2}$
  Seismic analysis Y-1: $M_{Tr,y,1}$
  Seismic analysis Y-2: $M_{Tr,y,2}$

(9) Resultant distribution of forces

Depending on the option selected (normal or accidental torsion), the normal or accidental R" forces are added to generated forces. In the case of normal torsion, forces are summed within seismic cases, whereas for calculations taking account of accidental torsion 2 cases per each declared direction are created (considering of eccentricities on both sides of the mass centroid).

NOTE: If calculations are performed according to the RPA99_2003 code, the program ignores the influence of horizontal forces resulting from torsion which would decrease the forces obtained in the first distribution stage.

Moments acting in columns / walls resulting from horizontal forces are also evaluated considering the forces due to torsion Rr,i" (for walls on one direction, and for columns on 2 directions).

ADVANCED CALCULATIONS

In the program advanced calculations may be performed in 2 ways:

- without considering the accidental torsion effect
- considering the accidental torsion effect (additionally, the not-intended eccentricity of the mass centroid location is taken into account; user-defined values are adopted).

When calculations considering accidental torsion are performed, the modal analysis is conducted for the distributed mass matrix and the stiffness matrix taking account of a shift by a defined eccentricity.

*NOTE: Since the real value of the eccentricity is assumed in advanced calculations, the sign is of importance (according to code regulations the eccentricity should be considered on both sides of the mass centroid, and in this case the user decides about it by entering a negative or positive value).*

### 7.4.25. Verification of a Necessary Wall Area

The verification is available for the seismic code P100-1/2006.

Verification of a necessary wall area:

- calculation method: simplified analysis
- comparison of the area of walls that carry horizontal forces on the directions X and Y with the code condition.

To verify the necessary area of walls, it is needed to define a value of the modification factor that equals I * ag / g, where I is a structure importance factor, while ag and g are seismic and gravitational accelerations, respectively.

During verification of a necessary wall area the following condition is checked:

$$\frac{\sum A_p}{A_{pl}} \geq \frac{I \cdot \frac{a_g}{g} \cdot n}{100}$$

where:

Ap - sum of the horizontal area of all walls that carry horizontal seismic forces

Apl - area of a slab positioned on a considered story

I * ag / g - product of a structure importance factor and a quotient of the seismic and gravitational acceleration (the value depends of the structure importance and a seismic zone)

n - number of stories of a building above the considered section.

If the condition above is not satisfied, an appropriate message appears on the screen.

The check of the condition for individual directions X and Y on each story of a building is presented in a calculation note.

# 8. DEFINITION OF LOADS

## 8.1. Rules of Load Definition in the CBS Pro Program

The following two terms have been introduced in the program: load group and load record.

Load group
Each load defined in the ***CBS Pro*** program has to belong to a load group. A group of loads contains load records resulting from the same type of actions and showing the same load factor (partial safety factor). Groups may be generated automatically, semi-automatically and manually. Semi-automatic and automatic generation of load groups involves automatic generation of load records.
Automatic generation concerns groups of loads resulting from non-structural elements (e.g. partition walls, layered walls, etc.). Load records are generated as soon as a model is being generated for the ***ROBOT*** program.
Semi-automatic generation refers to seismic loads. After defining code parameters, the program generates by itself required load cases (due to specific character of the load, load records are not generated) – the option is not available in the current program version.
There is also a possibility of semi-automatic generation of groups of live loads. The option enables assigning loads to earlier-defined rooms intended for the same purpose. In this case, load records are generated once an appropriate option is activated.

Load record
A load record is a single load applied to an object or its part. Several records may be grouped using the *Loads / Live Load Patterns / Group* option. Return to the original loads takes place once the *Loads / Live Load Patterns / Ungroup option* is used.

## 8.2. Types of Loads in the CBS Pro Program

The following load types are available in the ***CBS Pro*** program:

- concentrated load

  A concentrated load occurs in two variants: as a vertical load or as a horizontal load . The vertical load denotes a load compatible with the direction of *z* axis; the horizontal load is a load acting in the plane perpendicular to *z* axis. An angle for the horizontal load is measured from *x* axis in the direction of *y* axis.
  The concentrated load is assigned to quasi-point objects (column and spread footing) and that is interpreted as application of a load to their top part; this type of load may also be applied to any point of the structure.

- linear load

  A linear load occurs in two variants: as a vertical load or as a horizontal load . The vertical load denotes a load compatible with the direction of *z* axis; the horizontal load is a load acting in the plane perpendicular to *z* axis. An angle for the horizontal load is measured from *x* axis in the direction of *y* axis.
  The linear load may act in the direction perpendicular to an object or be a 'projected' load (then the load refers to the object length projected onto the plane normal to the direction of load).
  The linear load is assigned to linear objects (beam, continuous footing, top wall edge); it may also be applied to any point of the structure.

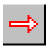
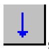
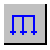
- planar load

  A planar load occurs in two variants: as a vertical load or as a horizontal load . The vertical load denotes a load compatible with the direction of *z* axis; the horizontal load

is a load acting in the plane perpendicular to *z* axis. An angle of the horizontal load is measured from *x* axis in the direction of *y* axis.
The planar load may act in the direction perpendicular to an object or be a 'projected' load (then the load refers to the object length projected onto the plane normal to the direction of load).
The planar load is assigned to planar objects (slab, wall, raft foundation); it may also be applied to any point of the structure.

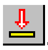
## 8.3. Definition of Loads

To define a load in the ***CBS Pro*** program, the user should follow the steps below:

- choose load variant: vertical load  or horizontal load 
- select load type: concentrated load , linear load  or planar load 
- determine a load value in the edit field that becomes accessible after selection of a load type
- choose a load nature from the drop-down list located to the right of the edit field where the load value is defined
- apply the defined load to an object; the user may assign a load only to this object type which allows for application of a chosen load type.

There is a certain difference in definition of vertical and horizontal loads. If the vertical load is selected, during definition the user determines point (or points) where a force is applied (for a load assigned to object, the user chooses an object). For the horizontal load there are two possibilities:

- geometrical definition – a point (or points) to which a force is applied are defined; after defining the last characteristic load point, the cursor switches to the mode which enables defining an angle – the distance of the mouse cursor to the snap point is unimportant then, only the angle value is significant. NOTE: There are 'preferred directions' set for the cursor, which results in that that if the angle differs negligibly from the direction of axes X or Y, the angle is 'snapped' to the direction of the axes mentioned.
- definition of a load assigned to object – the program waits for the user to indicate an element to which the load is to be applied. Objects are filtered, which means that it is possible to select only this element type to which a given type of load may be applied. Once an object is indicated, the cursor changes its mode to that which enables defining an angle (as described for the geometrical definition).

The program includes the option *Load Assigned to Object* . The option is available both in the 2D view and in the 3D view.

*NOTE:* *When defining a load assigned to an object of the value and subnature identical as a load already existing, the program displays a message informing that such load has been applied to an object to avoid doubling the loads.*
*It is possible to define simultaneously a planar load assigned to an object for more than one slab in the case of window selection in the 2D or 3D view.*

When defining a load the user should pay attention to:

- load position (only for forces not assigned to object) – for a concentrated force, it is one point where the force is applied, for a linear load – these are two values (beginning and end points of a load), for a planar load – these are contour coordinates

- load value – for a concentrated force and planar load, it is one value, whereas for a linear load – two values: value at the load beginning point and value at the load end point.

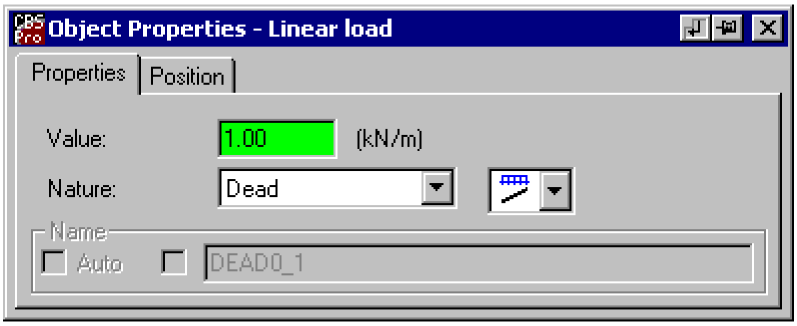
The load direction for individual load natures is adopted in the following way:
- *dead* load: vertical
- *live* load: any
- *snow* load: vertical - projected
- *wind* load: normal to the plane
- *accidental* load: any
- *temperature* load: any
- *seismic* load: any.

While defining a load the user should keep in mind that loads assigned to objects should be of the same type as object:

- for concentrated forces – spread footings, columns
- for linear loads – beams, continuous footings, walls (top edge)
- for planar loads – slabs and raft foundations (vertical and perpendicular to the plane) and walls (horizontally and perpendicularly to the plane).

## 8.4. Modification of Loads

For modification of loads the Properties dialog box is used (similarly as for modification of other objects in the ***CBS Pro*** program).




In the above dialog box the options provided on the *Properties* tab allow modification of the following parameters of a selected load:

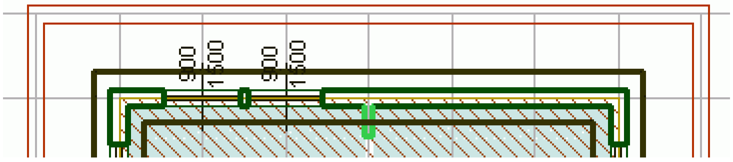
- load value
- load nature (dead, live, wind, snow, etc.)

the manner how the load is applied to object:
- vertical load
- horizontal load
- load perpendicular to object
- load projected onto object.

Options located on the *Position* tab enable modification of the place to which a load is applied.

## 8.5. How to Apply Load to a Structure

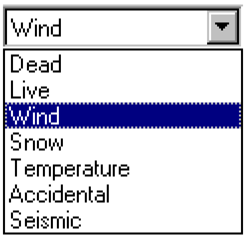
To apply a wind - planar load of value 0.4 kN/m$^2$ to the building wall visible in the drawing below, follow the steps below:



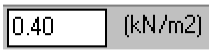

- switch on the *Load assigned to object* option by selecting the menu command *Loads / Assigned to Object* or by pressing the *Load assigned to object* icon on the *Loads* toolbar

- switch on the *Planar Load* option by selecting the menu command *Loads / Planar* or by pressing the *Planar load* icon on the *Loads* toolbar

- change the load nature by selecting the *Wind* option from the drop-down list located in the *Loads* toolbar



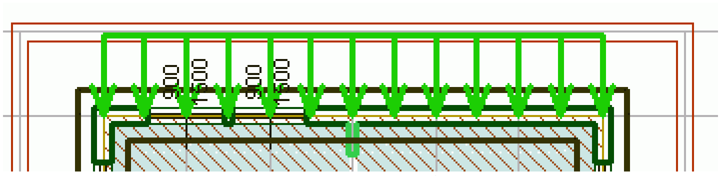

- enter the value of load (0.4 kN/m$^2$) into the edit field on the *Loads* toolbar

- 

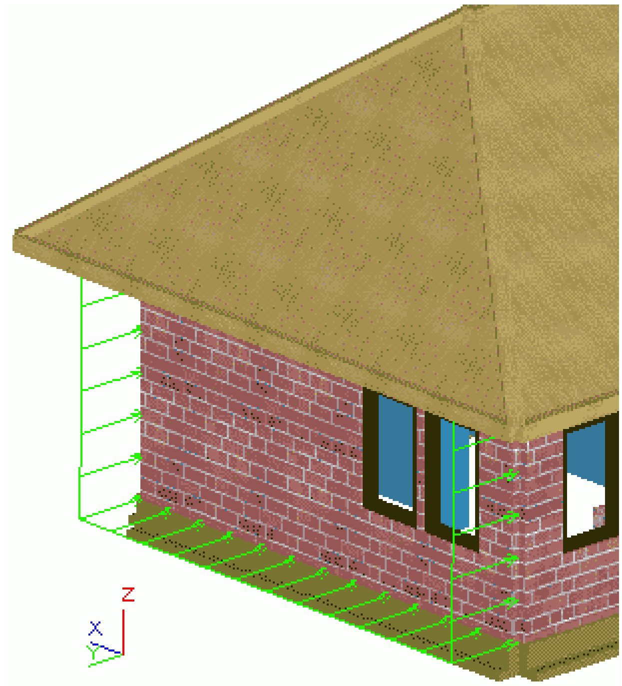

- switch on the *Horizontal Load* option by selecting the menu command *Loads / Horizontal* or by pressing the *Horizontal load* icon on the *Loads* toolbar

- in the 2D viewer select the wall to which the load is to be applied by positioning the mouse cursor over it (the wall becomes highlighted in yellow) and clicking with the left mouse button

- by means of the mouse choose the direction of load application, next, click with the left mouse button; the defined load is presented in the drawing below.



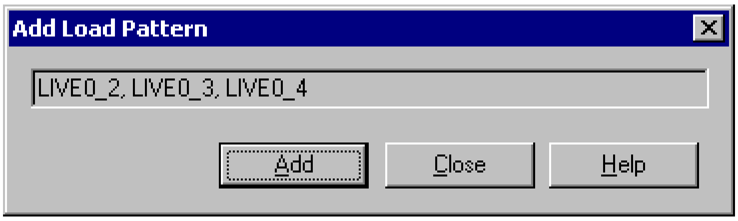
In the following 3D drawing it can be seen that the defined load is a planar one.




## 8.6. Add Load Pattern

The option enables manual definition of any load pattern. It is accessible from:

- the menu after selecting the *Loads / Live Load Patterns / Add* command
- the toolbar after pressing the icon.

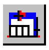


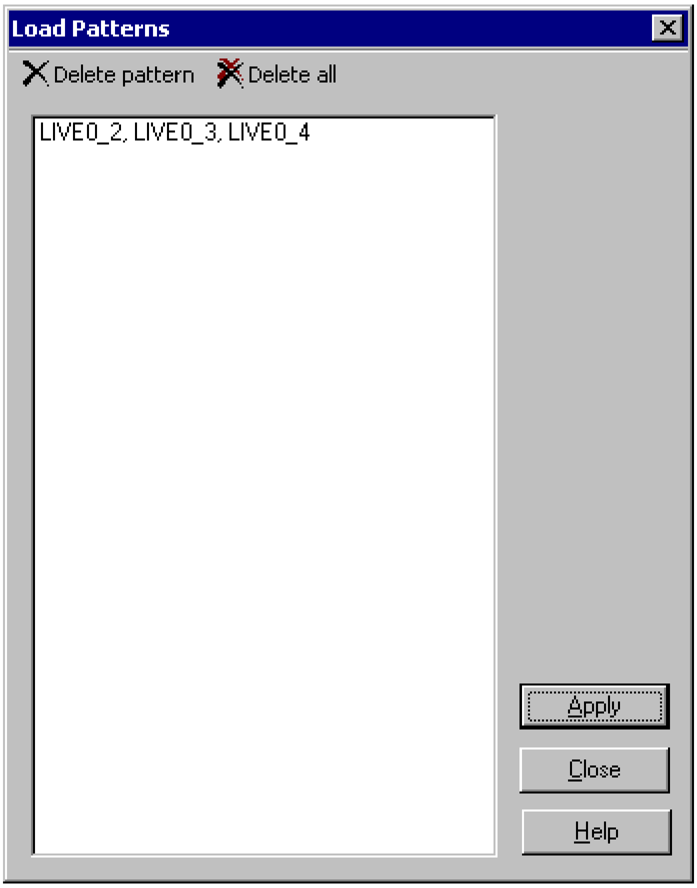

A load pattern is a set of live loads that may occur during operation of the structure. It consists of one or several live load records.
Definition of a load pattern is based on the current selection. In the dialog box above, the gray edit field presents the current selection; it shows names of selected loads. Selecting the following option from the menu (or the toolbar):

- *Select - Whole Structure* allows definition of patterns comprising loads from different stories
- *Select - Current Story* causes selected loads only from the current story to be taken into account.

## 8.7. Load Patterns

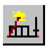
The option enables viewing load patterns defined automatically or manually. It is accessible from:

- the menu after selecting the Loads / Live Load Patterns / Display command
- the toolbar after pressing the  icon.



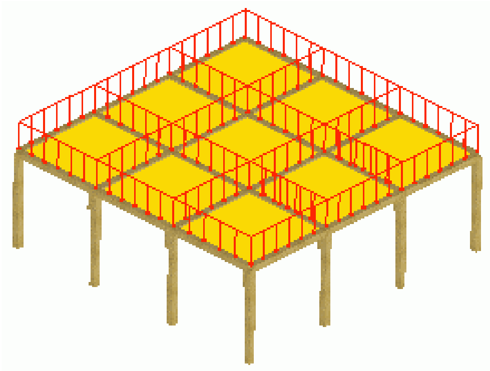

The above dialog box contains the list of all the defined load patterns. Once a load pattern is chosen in the dialog box, the 3D view presents the set of load records included in this pattern. In the dialog box load patterns can be also deleted (the buttons **Delete pattern** and **Delete all** located in the top part of the dialog box).

## 8.8. Automatic Generation of Load Patterns

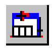
In the ***CBS Pro*** program the automatic generation of live load patterns consists in performing permutations of all live loads on each story; apart from that, to thus-generated permutations the program adds a load case containing all the loads positioned above the given story. Such a method of load generation enables the user to generate a highly probable set of forces being the set of forces designing all the structural elements and at the same time – being the set limiting the number of generated combinations.
To start automatic generation of load patterns the user should select the menu command: *Loads / Live Load Patterns / Generate* or press the  icon.

# 8.9. How to Define Automatically Load Patterns

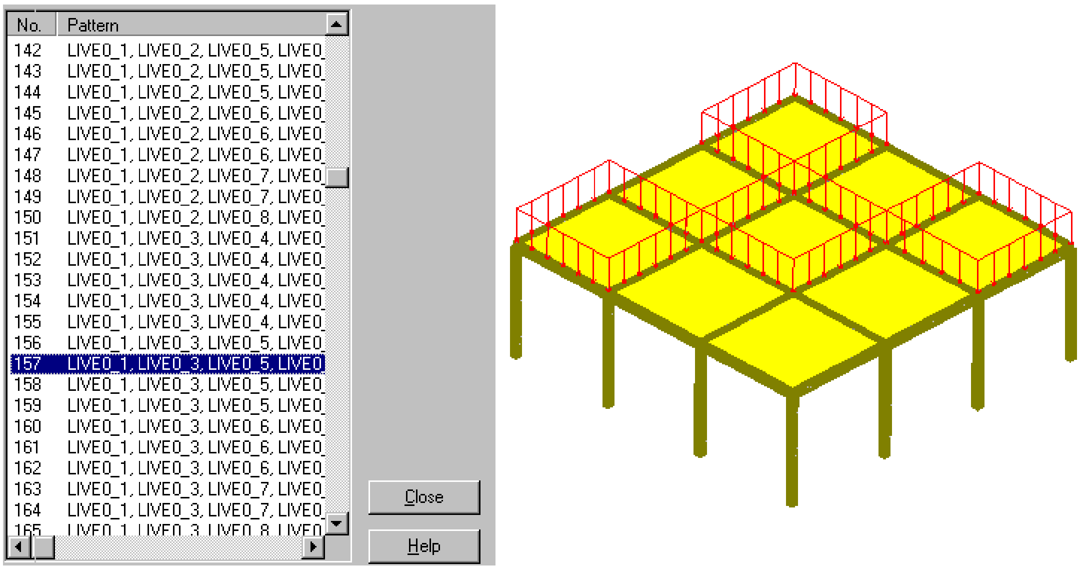
To generate automatically all load patterns for nine independent, live loads applied to the slab, presented in the drawing below (each slab is loaded by a separate planar load), follow the steps below:



- select the menu command *Loads /Live Load Patterns / Generate* or press the *Generate*  icon in the *Loads* toolbar

- select the menu command *Loads /Live Load Patterns / Display* or press the *Display*  icon in the *Loads* toolbar

- in the ***Load Patterns*** dialog box click with the left mouse button on one of the patterns (it becomes highlighted in blue) – a selected load pattern will be shown in the 3D viewer (see the drawing below).



# 8.10.Load Conversion during Generation of a Model in the ROBOT Program

The following rules of conversion of loads defined in the ***CBS Pro*** program apply during generation of a structure calculation model for the ***ROBOT*** program:

- dead loads – all the groups of dead loads defined in the ***CBS Pro*** program belong to one nature of dead loads in the ***ROBOT*** program. Each group of dead loads is converted into a subnature of dead loads with the name compatible to the group's name. Each of these subnatures contains exactly one load case (equivalent to the relevant group of loads from the ***CBS Pro*** program). All the load records from a given group of dead loads from the ***CBS Pro*** program are converted into load records in the corresponding case of the ***ROBOT*** program. Besides, the program generates dead loads resulting from self-weight, partition walls and finishing layers.

- live loads - all the groups of live loads defined in the ***CBS Pro*** program belong to one nature of live loads in the ***ROBOT*** program. Each group of live loads is converted into a subnature of live loads with the name compatible to the group's name. If no additional limitations (load patterns) are introduced, then each load record within a given group is converted into a separate calculation case. It means that the ***ROBOT*** program generates as many load cases as many load records have been defined in the ***CBS Pro*** program.

*NOTE:* *Load generation without using the option* Loads / Live Load Patterns / Add *or* Loads / Live Load Patterns / Group *may only be performed for small structures because the growing number of code combinations may quickly exceed the maximal number of combinations allowed in the* **ROBOT** *program. The limit value of code combinations in the* **ROBOT** *program is reached after defining ten-odd live load records in the* **CBS Pro** *program.*

*Load patterns*

Load patterns are the most natural way of reducing code combination cases. On their basis appropriate combinations are generated in the ***ROBOT*** program. After switching on the *Loads / Live Load Patterns / Add* option all the records from one load group within a story (or within the whole structure) are combined into several independent load patterns defined by the user.

Group loads

Load records combined by means of the *Loads / Live Load Patterns / Group* option are treated as a single record. It means that they are included in code combinations simultaneously.

- snow load – all the groups of the snow load are ascribed to one nature: *Snow* in the ***ROBOT*** program. Each of the groups makes up a separate load case in the ***ROBOT*** program with the name compatible to the group's name determined in the ***CBS Pro*** program. Relations between load cases are set automatically to exclusive. All the load records in this load group of the ***CBS Pro*** program are converted into load records in the corresponding load case of the ***ROBOT*** program.

- wind load - all the groups of the wind load are ascribed to one nature: *Wind* in the ***ROBOT*** program. Each of the groups makes up a separate load case in the ***ROBOT*** program with the name compatible to the group's name determined in the ***CBS Pro*** program. Relations between load cases are set automatically to exclusive. All the load records in this load group of the ***CBS Pro*** program are converted into load records in the corresponding load case of the ***ROBOT*** program.

- accidental load - all the groups of the accidental load are ascribed to one nature: *Accidental* in the ***ROBOT*** program. Each of the groups makes up a separate load case in the ***ROBOT*** program with the name compatible to the group's name determined in the ***CBS Pro*** program. Relations between load cases are set automatically to exclusive. All the load records in this load group of the ***CBS Pro*** program are converted into load records in the corresponding load case of the ***ROBOT*** program.

- temperature load - all the groups of the temperature load are ascribed to one nature: *Temperature* in the ***ROBOT*** program. Each of the groups makes up a separate load case in the ***ROBOT*** program with the name compatible to the group's name determined in the ***CBS Pro*** program. Relations between load cases are set automatically to exclusive. All the load records in this load group of the ***CBS Pro*** program are converted into load records in the corresponding load case of the ***ROBOT*** program.

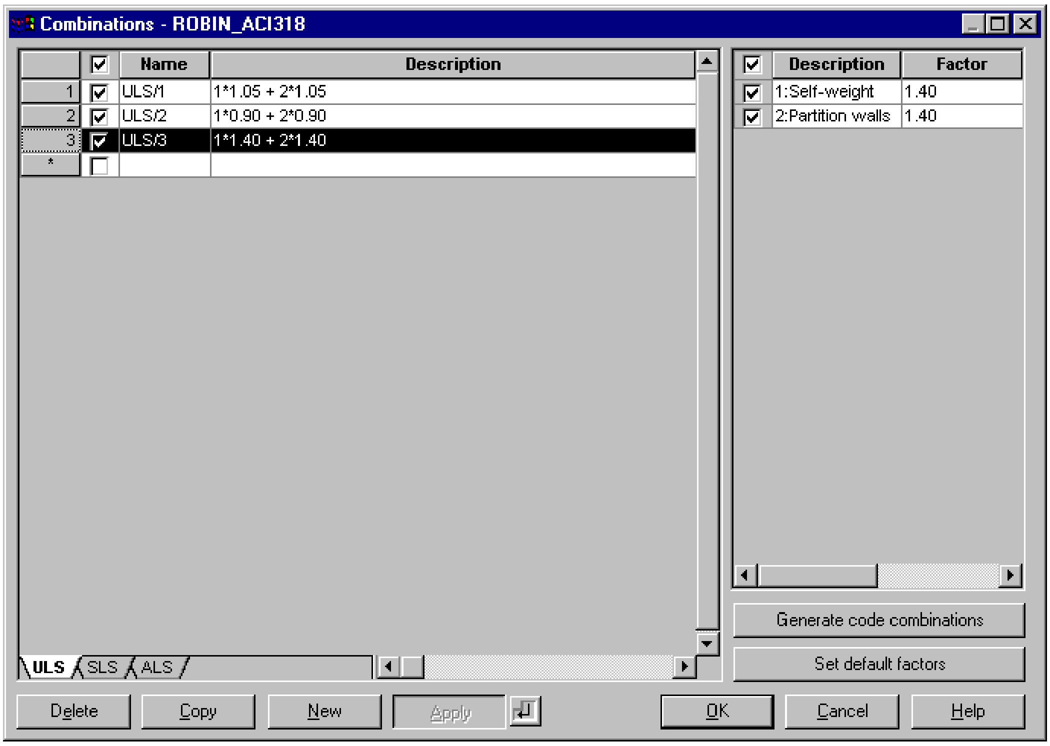
- seismic load – seismic loads are generated automatically according to defined parameters – the option is not available in the current program version. Load groups are reflected in seismic load cases in the ***ROBOT*** program.

## 8.11.Load Combinations

The option allows defining code and manual combinations according to a regulation selected in the ***Default loads*** dialog box. The option is available from:

- the menu after choosing the *Loads / Combinations* command

- the toolbar, after pressing the icon.




The left-hand part of the dialog box includes a table containing a list of manual combinations or generated code combinations (components of code combinations are marked with a color background).
Combinations are grouped on the relevant tabs (ULS, SLS, ALS) presenting combinations for individual limit states depending on a selected regulation of code combinations.
The combination table is composed of the following columns:

- name of a combination / component

- list of cases (with factors for each case) that are included in a given combination.

Each combination or component of code combinations may be switched on / off. If a combination is switched off in the table, it will not be considered while presenting results (after recalculation of a structure).

In the right-hand part of the dialog box there is a table which presents load cases (along with factors for each case) that can be used in definition of a new combination or modification of an existing one. In the column with factors it is possible to enter user-defined values of load factors for individual cases.

The bottom right part of the dialog box holds the buttons as follows:

- **Generate code combinations** – pressing this button results in defining code combinations according to an adopted regulation; if load combinations have already been defined for a structure and this button is pressed, the following question will appear on the screen: 'Do you want to delete existing combinations?'

  After pressing the **YES** button, the existing load combinations will be deleted and code combinations will be generated.
  After pressing the **NO** button, a list of generated code combinations will b added to the list of defined combinations.

- **Set default factors** – after pressing this button default values of factors from selected regulations will be adopted.

At the bottom of the dialog box (under the combination table) are the following the buttons:

- **Delete** - pressing this button deletes a combination or a component of a code combination

- **Copy** - pressing this button copies a combination or a component of a code combination

- **New** - pressing this button inserts an additional blank row in the combination table; this row may be used to define a new combination – it is defined by selecting cases, located in the load case table (the right-hand side of the dialog box), included in the combination and editing load factors

- **Apply** - pressing this button accepts changes (the button is accessible if the icon **Enter** is switched off)

- **Enter** – after activating this icon the changes made are automatically accepted.

*NOTE:* *If the* **Combinations** *dialog box includes load combinations defined earlier and the* ***Generate code combinations*** *button is pressed, then all combinations existing so far will be deleted (even if combinations have been modified by the user) after pressing the* ***YES*** *button (see the description of the* Generate code combinations *option provided above).*

*NOTE:* *In the case code combinations are edited (a change of a load factor, deleting of a component), the program treats the remaining set of components of a combination as manual combinations.*

# 9. STRUCTURE CALCULATIONS

## 9.1. Structure Calculation

The calculation of a structure in the ***CBS Pro*** program can be performed:
- for the whole structure
- for each story separately.

In the latter case the analysis takes account only of the vertical forces applied directly to the story in question.

To start the calculation of the structure, follow the steps below:
*for calculation of the whole structure*

- select the Calculations / Calculations of Whole Structure command from the menu
- press the icon .

*for calculation of the current story*

- select the Calculations / Calculations of Current Story command from the menu
- press the icon .

NOTE: *If calculations are to be performed for a selected story, then two simplified methods of structure analysis are available; the structure analysis using the exact method based on the finite element method (FEM) is impossible.*

If in the ***Calculations*** dialog box (for the whole structure or the current story) the option *Always display this dialog box before calculations* is switched on, then the ***Calculations*** dialog box containing the structure calculation options appears on the screen; to start the structure calculation, press the **Calculate** button.

## 9.2. Calculation Options

The ***Calculation Options*** dialog box is used to determine parameters of the static calculation of a structure, the code calculation of RC elements of the structure as well as the analysis of the load capacity of the foundations (soil parameters).
The option is accessible:

- from the menu after selecting the *Calculations / Calculation Options* command
- after pressing the icon .

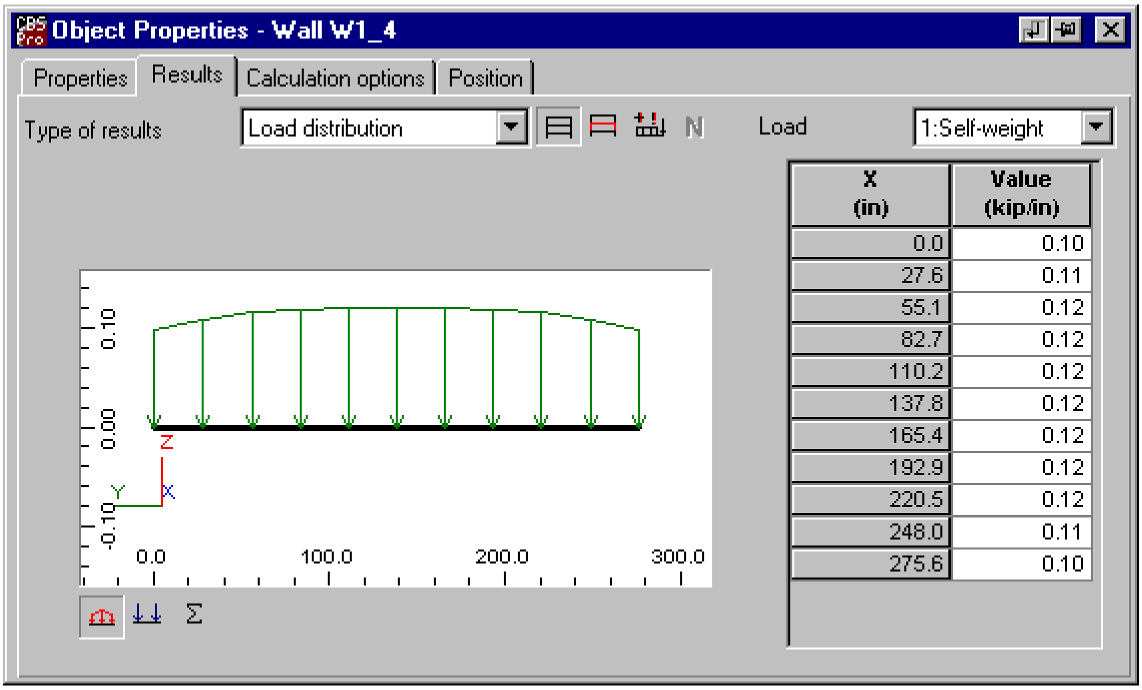
The ***Calculation Options*** dialog box consists of the following three tabs:
- Calculations
- RC element design
- Soil.

In the dialog box it is also possible to save a set of selected options as default for new structure projects. It is enabled by the *Set as Default* option located at the bottom of the dialog box.

# 9.3. Object Properties - Results and Calculation Options

The tabs: *Results* and *Calculation options* in the ***Object Properties*** dialog box are displayed for the following objects:

- *Results* tab – all objects
- *Calculation options* tab – all objects except continuous footings.




*NOTE:* *It is possible to change dimensions of the* **Object Properties** *dialog box on the* Results *tab (for all element types) and on the* Calculation options *tab (for slabs and beams).*

The *Results* tab includes:

- *Type of results* selection list, which comprises the following options: *Load distribution*, *Internal forces*, *Design*
- *Load* selection list, which comprises load subnatures applied in a structure model (they are visible once the calculations are performed)
- object view presenting the results; for members the dialog box presents a static scheme of a member, for spread footings - a view of a spread footing with designation of forces applied to it, for slabs - a slab with supports
- table of internal forces
- at the bottom of the dialog box – controls that will be described further on, depending on the type of calculations and object.

Results may be presented for:

- whole structure
- current story.

*NOTE: If the option for presenting combinations is switched off (the icon is switched off), the load case list includes, apart from simple cases, only extreme combinations (ULS+, ULS-, SLS+, SLS-, ALS+, ALS-). After activating this option (the icon is switched on) the list of cases also comprises all components of code combinations and combinations defined manually which are marked as active in the* **Combinations** *dialog box.*

In the upper part of the dialog box is also the icon, which when pressed opens the ***Add to Note*** dialog box; the option allows adding components of calculation results (both results of simplified and advanced calculations) and design results for structure elements to a calculation note.
The *Load distribution* option is available to calculations performed in the ***CBS Pro*** program; the appearance of the dialog box changes with type of a selected object:

- slabs - the dialog box presents a view of load distribution zones as well as a table with a list of elements supporting the slab and the total of loads; below are accessible the options: Zoom Window and Zoom All

- beams, walls - the dialog box presents a view of loads resulting from slabs - according to a selected option (linear loads, concentrated loads or the total of all loads), together with a static scheme of a member and a table including elements corresponding to the selected options: values of a linear load at 10 points of the object, values of concentrated forces, totals of linear and concentrated loads applied to the element

- columns - the dialog box presents a view of internal forces (axial force, shear forces and moments) resulting from the elements supported by the column and a table with values of these forces; additionally, the static scheme of a column is displayed; the default column scheme is a column with fixed supports on both ends, however, the following column schemes may appear as well:

  - pinned support at the column base if this is a column with a spread footing
  - pinned supports on both ends of a column, if the column is disregarded for carrying of transversal forces.

The *Internal forces* option is available to calculations performed by the ***ROBOT*** program; the appearance of the dialog box changes with type of a selected object:

- slabs, raft foundations, walls - the dialog box presents a map of a selected quantity (internal forces: M'xx, M'yy, M'xy, N'xx, N'yy, N'xy, elastic displacements (without taking account of cracking of RC elements): U'x, U'y, Uz or reinforcement areas: A'x+, A'x-, A'y+, A'y-) and a table with values and coordinates of nodes; the following options are available:

  the activated icon means that values will be presented in a displayed map

  pressing this icon increases the font size in descriptions of values on maps

  pressing this icon reduces the font size in descriptions of values on maps

  pressing this icon selects the graphical selection mode within the presented map

  Zoom-window

  Initial-zoom

  switches on/off an FE mesh

  allows defining a mesh for all maps of internal forces after calculations using the simplified and exact methods and maps of the required reinforcement; it opens the ***User-defined Mesh*** dialog box

  under the table the following options are provided:

  values of a selected quantity at individual mesh nodes; after activating the icon, maps will present values of a selected quantity

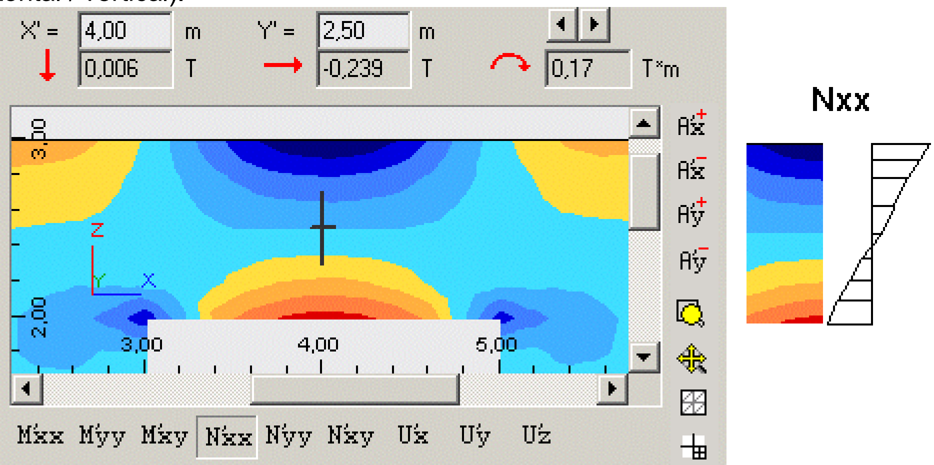
NOTE: Quantities for presentation may be selected by (with the selection mode active):
- activating options (the √ symbol appears) in the table with values for nodes / elements
- indicating a point on the map with the mouse cursor; it results in displaying a value at the closest node / center of a finite element.

min./ max. values

values of a selected quantity at centers of individual finite elements; after activating the icon, maps will present values of a selected quantity

scale and two options for selection of the scale: automatic (non-editable) and user-defined.

In case of the automatic scale, 10 colors are generated (segments of equal length) by default; the scale includes colors from red to blue; if min./max. values differ, then the number of positive colors is different from the number of negative colors (e.g. 7 : 3), in such a case, the intensity of negative (positive) colors stays in proportion to their number (they are selected as if there are 7 of them, while only first 3 of them are displayed)
The user-defined scale is global for different map types and includes only 4 lines with default values corresponding to individual quantities:
M - M'xx, M'yy, M'xy – default intervals 20, 0, -20 [kNm]
N - N'xx, N'yy, N'xy - default intervals 100, 0, -100 [kN]
U - U'x, U'y, Uz - default intervals 0.03, 0, -0.03 [m]
User-defined scales may be modified using the buttons: **Insert, Delete**; these scales are available globally within one project.
For walls it is possible to perform calculations of reduced forces, which enable determining the equivalent set of internal forces: the horizontal force H, the vertical force V and the moment M. These forces may be used in the design of reinforcement of lintels or for the check of the shear capacity of a wall. The calculations of forces are based on the cut that passes through the point X'. To calculate the moment the point on the cut X' with the coordinate Y' is used.
NOTE: The program calculates automatically values of reduced forces in sections defined in the ***Reduced Forces*** dialog box. To move between the characteristic points determined automatically, the buttons in the top right part of the frame are used.
The user may specify arbitrary values of coordinates as well as change a cut type (horizontal / vertical).




Calculations of internal forces consists in integration of the appropriate internal forces of a wall:

$$H = \int_L^U N_{xx} dy'$$

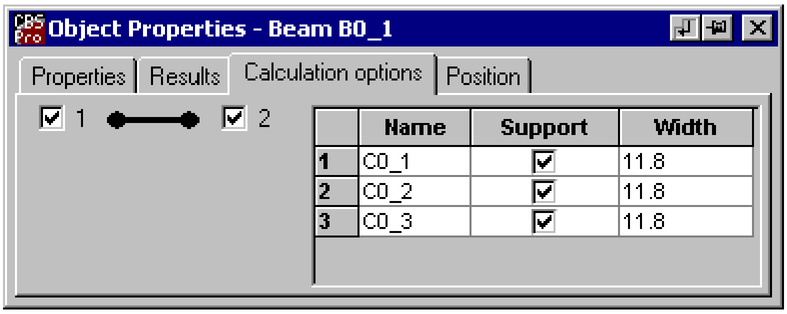
$$V = \int_L^U N_{xy} dy'$$

$$M = \int_L^U N_{xx} \cdot (Y' - y') dy'$$

NOTE: The boundaries of the integration regions are the closest edges of a wall and openings. It means that for a point above an opening the regions located below its lower edge are not considered.

- beams, continuous footings, columns – the dialog box presents diagrams of a selected quantity (Mx, My, Mz, Fx, Fy, Fz) and a table with values and the coordinates of division points corresponding to these values; below are the options: values at division points, min./max. values
  NOTE: For continuous footings the forces FX, FY and FZ (reactions for walls calculated in ***ROBOT***) may be presented – see Default options
- spread footing - the dialog box presents a view of the spread footing with designation of forces applied to it and a table with a list of forces showing values for a selected load subnature.

For individual types of objects it is set by default that diagrams are displayed below the following values:
- for columns and spread footings: axial force Fx
- for beams and continuous footings: bending moment My
- for slabs, raft foundations and walls: Mxx moment.




The *Calculation options* tab comprises the following options:
- in the left part of the dialog box:
  for beams: switching on the options 1, 2 means that a release has been defined on a selected beam end
  for slabs, raft foundations:

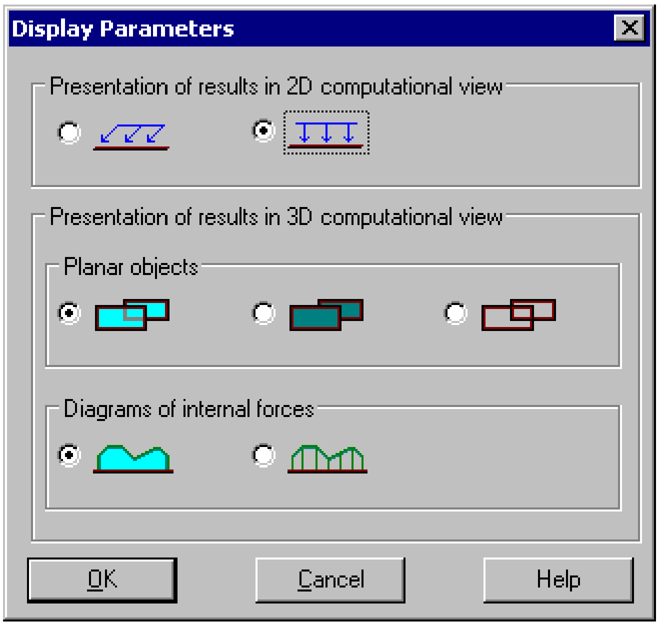
  *Main direction* allows defining a direction of transferring loads from a slab onto supports (in the case of simplified calculations) and determining the local direction of a panel. It may be done as follows:
  - by selecting the option *Define main direction according to an object* and choosing the edge to which the X' axis should be parallel
  - by selecting the option *Define main direction by means of a line* and indicating the direction of the X' axis in the structure view
  - by deleting the main direction and restoring default settings of support selection
- in the right part of the dialog box – a table with the list of objects that may act as a support for the object in question; it presents a selected static scheme of a support (pinned / fixed); there is a possibility to modify the program settings and to switch on/off options or to change a support scheme, and additionally for simplified calculations, it is possible to change the coefficient value in order to modify the structure work model

- switching on / off the option that allows disregarding objects which do not carry horizontal forces - it refers only to walls (in the simplified method) and columns (in all calculation methods)
  In the case of simplified calculations, disregarded objects do not participate in the distribution of horizontal forces in a model.
  In the case of advanced calculations, a static scheme of pinned supports on both ends is assumed for columns; as a result, columns carry only vertical loads.

NOTE: *If a linear support of a slab (beam or wall) is switched off on the support list, then the directions Rx and Uz on this edge will be released in calculations using the exact method.*

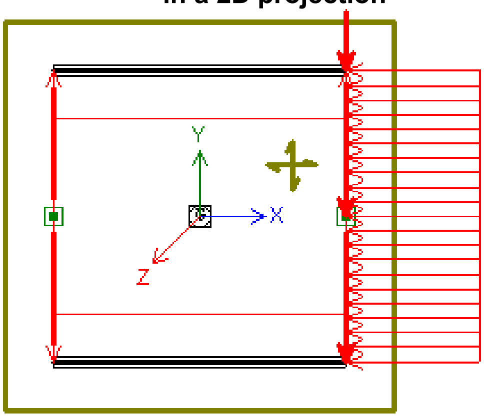
## 9.4. Presentation of Results in a 2D and 3D View

The option allows setting parameters of result presentation in a 2D / 3D view. The option is available from the menu after selecting the *Calculations / Display Parameters* command.

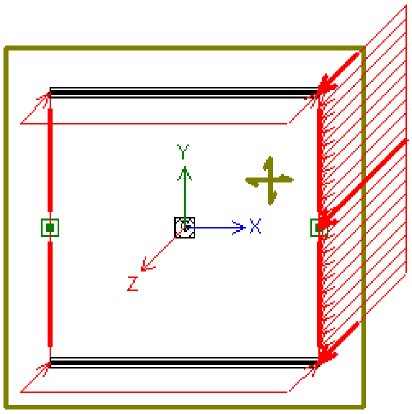



In a 2D view results (loads parallel to the Z axis) may be presented:

**in a 2D projection**



**in an axonometric view**



In the 3D view results may be presented:

- Planar objects

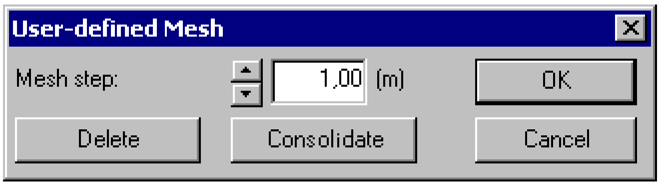
transparent objects
filled objects
blank objects

- Diagrams of internal forces:

filled
hatched.

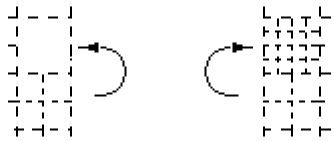
## 9.5. User-defined Mesh

The dialog box opens after pressing the icon in the ***Properties - Results*** dialog box.




The options in the above dialog box allow defining a value of the cell in a mesh (in the *Mesh step* field) that will be generated on a considered panel. Individual mesh fields display the maximum value of required reinforcement.
Once a base mesh is defined, it is possible to refine the mesh locally as well as to consolidate several mesh cells into larger, rectangle-shaped areas (the drawing below illustrates the operations of mesh consolidation and refinement - the lower mesh is the initial one).

*CONSOLIDATION* *REFINEMENT*



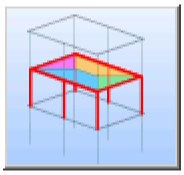
After switching on the user-defined mesh, the automatic scale is presented to the right of the dialog box; there is, however, a possibility to change the automatic scale to the user-defined one and to define the variability intervals for the scale.
The user-defined mesh may be deleted by:
1. pressing the **Delete** button
2. changing the rotation angle of the panel local system.

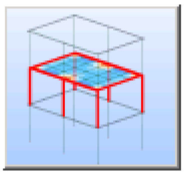
The arrows provided in the above dialog box enable increasing or decreasing a value of the mesh cell. Pressing the **Consolidate** button enables consolidation of the mesh cells into larger, rectangular areas.
A view of required reinforcement with a user-defined mesh presented in the ***Properties - Results*** dialog box may be added to a calculation note.

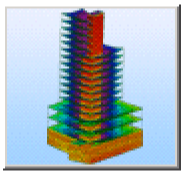
*NOTE:* *The defined mesh may be modified as follows:*
*- select a mesh cell (or several cells)*
*- in the dialog box specify a step value (size of the mesh cell).*
*The dialog box provides a value of the mesh cell defined so far; if there are different values of the mesh cell, it displays the least of them.*

## 9.6. Calculation - Calculation Options

The calculation of a structure in the ***CBS Pro*** program can be performed using one of the three available methods of analysis of the loads applied to the structure.

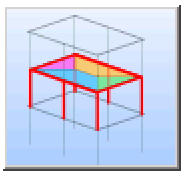
-  simplified method, that employs the trapezoidal and triangular method for distribution of vertical loads and the simplified distribution of horizontal forces based on the model of the frame subjected to shear (the trapezoidal and triangular method)

-  simplified method, that employs the analysis of vertical forces based on the FEM calculations for the singled-out slabs and transfer of the reaction to supporting elements as well as simplified distribution of horizontal forces based on the model of the frame subjected to shear

-  exact method based on the FEM analysis of the whole structure (NOTE: this method is not available for calculation of a single story).

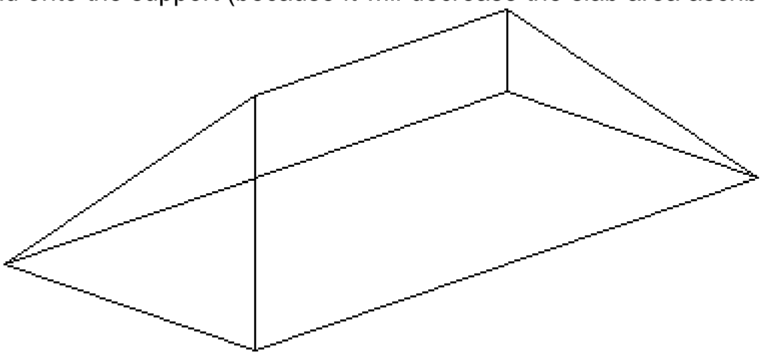
In the upper part of the dialog box there is the *Generation of code combinations* option. If this option is switched on, it means that during calculations of a structure the program will generate code combinations according to a code selected in the ***Preferences*** dialog box.
If load combinations have already been defined for the structure, and the next step is generation of code combinations, then the following question appears on the screen: 'Do you want to delete existing combinations?' After pressing the **YES** button the existing load combinations will be deleted and the program will generate code combinations; after pressing the **NO** button a list of generated code combinations will be added to the list of defined combinations.
The *Tolerance of reaction balance* edit field allows defining an admissible value of the error of the load and reaction balance for a structure; the tolerance is specified in percent. If the tolerance value for a designed structure is exceeded, then the message about the imbalance of reactions and loads appears on the screen. NOTE: in the current program version, the reaction and load balance is checked only in the case of calculations with the use of simplified methods.
After choosing one of the simplified methods, the upper part of the dialog box includes two selection lists (*Structure type: Direction X* and *Direction Y*) which allow a separate definition of the system carrying horizontal forces for individual directions (**frame** - horizontal forces are carried only by columns, **membrane** - horizontal forces are carried only by walls, **mixed structure** - horizontal forces are carried by columns and walls) – see the description of Default Loads (wind and seismic loads).
Additionally, the *Structure type* field holds the ***Openings*** button; when pressed, it opens the ***Openings*** dialog box.
If one of the simplified methods is chosen the *Default support conditions* option is also available in the dialog box. If the option is switched on, the following operations will be performed:
- the program will automatically recognize the support system of individual structure elements
- all slabs will be assigned 2 load-carrying directions
- values of all reduction factors (distribution of a load onto a support) will equal 1.0.
When running calculations of the structure, all manual settings defined earlier by the user will be deleted and replaced by the parameters mentioned above.

# 9.7. Calculation - Calculation Options (Trapezoidal and Triangular Method)

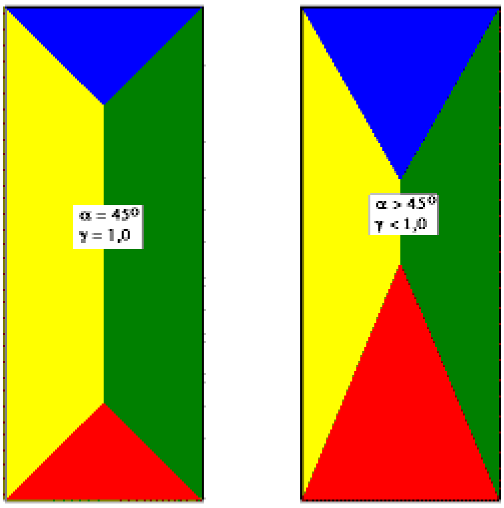


The method is applied after pressing the icon that represents the trapezoidal and triangular method of load distribution in the ***Calculation options*** dialog box.

The first analysis step consists in dividing a slab into regions from which the loads are distributed onto individual supports. The division is based on the analogy of the sloping surfaces of the roof. By default, the sloping surfaces are inclined at the $45^{O}$ angle with respect to the slab plane (see the drawing below). This angle may be increased by modification of a value of the reduction factor available in the ***Properties / Calculation options*** dialog box. The lesser factor will increase the pitch of the surface, and in consequence, will result in a lesser value of the load onto the support (because it will decrease the slab area ascribed to it).

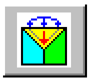


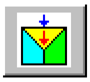
In simple cases, using this method the program generates triangular and trapezoidal regions, from which the loads are transferred onto the supports of the slab.



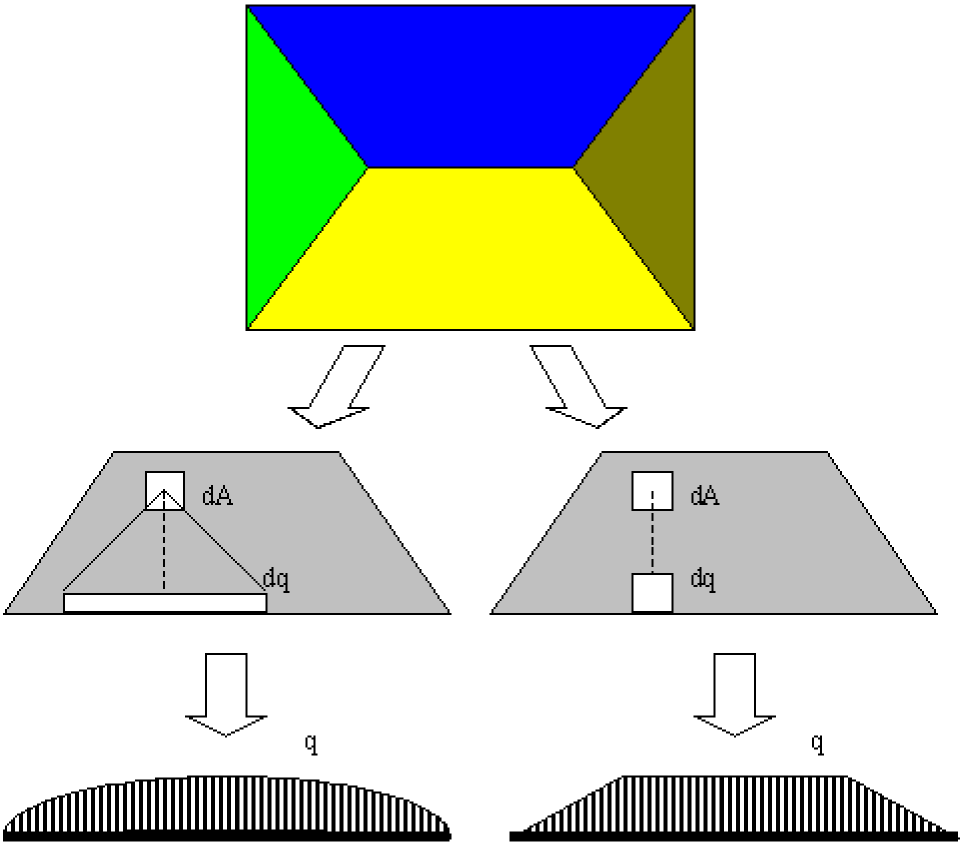

The division of the slab can be seen once maps of the triangular and trapezoidal regions are displayed in the ***Properties*** dialog box on the *Results* tab. They are available, as well, in the computational view of the structure, where it is possible to display them for all slabs or for any selection of them.

The second step of the algorithm is to calculate the equivalent of the region of the loaded slab in terms of the forces applied to the supporting elements. The elements may be loaded with trapezoidal loads on the elements' segments and concentrated forces.
The following two icons available in the ***Calculation options*** dialog box are used to steer with this part of the algorithm:

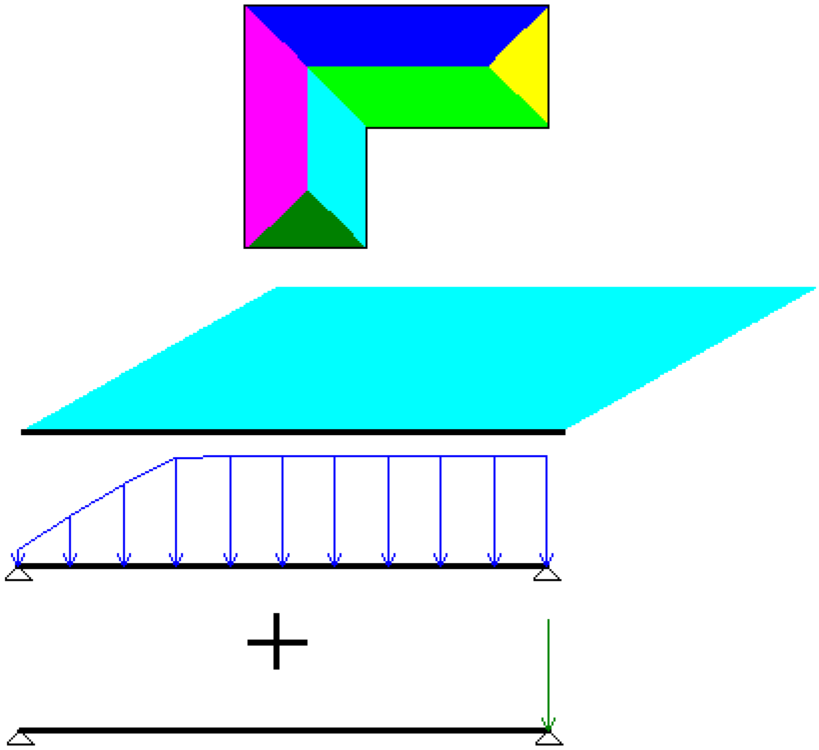
 – transforms the load region into the force applied to the supporting element, assuming that the extent of the region acted on by the force is limited with the 45° angle.

 – converts the region to the force in such a way that the height of the region at a given point is directly projected and the value of the projected height is assumed to be the value of the force applied to the given point of a supporting element.

It is presented schematically in the drawing below:




If part of the projected region extends off the support, then this part is transformed into a concentrated force applied to the element end – see the drawing below.



Effects of the action of horizontal forces are calculated from the analysis of the frame with pliable columns (walls) and infinitely rigid spandrel beams (the slab-beams system). A detailed description of the method is included in the description of the algorithm for calculation of the moments in columns and walls caused by horizontal forces.

Simplified calculations allow now taking account of a wall supported by supports at points (columns) or by walls in a calculation model. At present, such a wall is analyzed as deep-beam.
Deep-beams carry loads from slabs positioned above them and underslung to them. Similarly as for other objects, diagrams of load distribution are presented in the 2D and 3D computational view:

- in the 3D computational view results are presented at the top of the wall (from the upper slab) and at the bottom of a wall (from the lower slab)
- in the 2D view and in the ***Object properties*** dialog box, a diagram of load distribution for a deep-beam includes a sum of loads applied to the upper and bottom levels of a wall

In the ***Object properties*** dialog box, the *Results* tab presents a static scheme of a deep-beam, while the *Calculation options* tab presents support conditions as for beams.
It is possible to load a deep-beam to the ***ROBOT*** RC module **Deep-beams**; loads are applied to the upper and/or lower level of a deep-beam.

*NOTE:* *In the case of structure calculations by means of the triangular and trapezoidal method, the self-weight and loads of stairs will be taken into account if supports are defined for flights of stairs.*

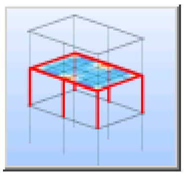
The final result of this analysis are:
- regions of the load distribution
- forces applied to elements.

*NOTE:* *In the current program version loads applied to walls are transferred to lower stories taking account of the angle of load distribution that equals 45 degrees (irrespective of a wall material selected).*

Moreover, this simplified method allows calculation of the provided reinforcement for RC slabs; calculations of the provided reinforcement area of slabs are performed by ***ROBOT*** RC

modules. Before starting calculations of the provided reinforcement, the ***Meshing Options*** dialog box appears and the mesh cell should be defined there.
A slab together with beams on which it is supported and supports in the places where columns and walls are defined is sent to the ***ROBOT*** RC module. There advanced static calculations and calculations of required and provided reinforcement are performed.
Types of slab reinforcement are ascribed analogously as in the case of advanced calculations of a whole structure including calculations of required reinforcement of panels.

## 9.8. Calculation - Calculation Options (Simplified Method)



The method is applied after pressing the icon that represents the simplified calculation method in the ***Calculation options*** dialog box.

The calculation in this method is performed story by story. Slabs together with the supporting elements (walls, columns, beams) and the vertical loads applied to these slabs are singled out from a story. The following step is solution of the singled out slabs by means of the FEM method, assuming fixed supports at the points of the supporting elements. The reactions obtained from the FEM solution are distributed onto the supporting elements as loads.
This method allows steering the size of a mesh of finite elements generated on slabs using the *Meshing* option provided in the ***Calculation options*** dialog box. Coarse meshing results in generating a small number of elements (approx. 2-8 finite elements for a slab) thus enabling quick analysis of large structures. The *Element size* option is used to determine a preferred size of a finite element.

Effects of the action of horizontal forces are calculated from the analysis of the frame with pliable columns (walls) and infinitely rigid spandrel beams (the slab-beams system). A detailed description of the method is included in the description of the algorithm for calculation of the moments in columns and walls caused by horizontal forces.

Simplified calculations allow now taking account of a wall supported by supports at points (columns) or by walls in a calculation model. At present, such a wall is analyzed as deep-beam.
Deep-beams carry loads from slabs positioned above them and underslung to them. Similarly as for other objects, diagrams of load distribution are presented in the 2D and 3D computational view:

- in the 3D computational view results are presented at the top of the wall (from the upper slab) and at the bottom of a wall (from the lower slab)

- in the 2D view and in the ***Object properties*** dialog box, a diagram of load distribution for a deep-beam includes a sum of loads applied to the upper and bottom levels of a wall

In the ***Object properties*** dialog box, the *Results* tab presents a static scheme of a deep-beam, while the *Calculation options* tab presents support conditions as for beams.
It is possible to load a deep-beam to the ***ROBOT*** RC module **Deep-beams**; loads are applied to the upper and/or lower level of a deep-beam.

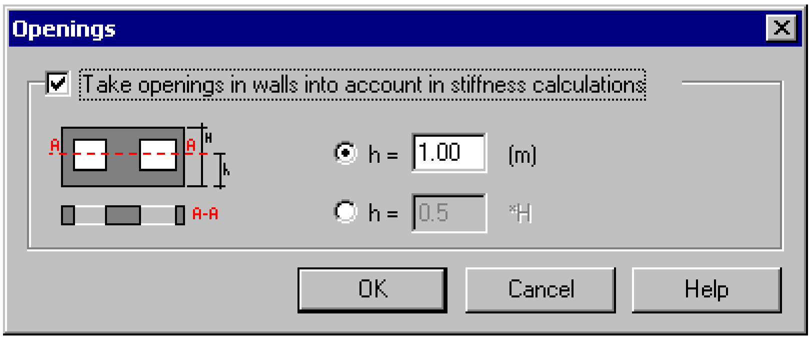
*NOTE:* *In the case of structure calculations by means of the simplified method, the self-weight and loads of stairs will be taken into account if supports are defined for flights of stairs.*

The final result of this analysis are:
- maps and values of moments for slabs
- forces applied to elements.

Moreover, this simplified method allows calculation of the provided reinforcement for RC slabs; calculations of the provided reinforcement area of slabs are performed by ***ROBOT*** RC modules. For calculations, this method uses an element mesh.
The local system of raft foundations is directed opposite to the axis Z of the global system when the provided reinforcement is calculated; this is done to ensure that punching is taken into account. Thus reinforcement layers are presented in the opposite way than in the case of regular slabs.
A slab together with beams on which it is supported and supports in the places where columns and walls are defined, is transferred to the ***ROBOT*** RC module. There advanced static calculations and calculations of required and provided reinforcement are performed.
Types of slab reinforcement are ascribed analogously as in the case of advanced calculations of a whole structure including calculations of required reinforcement of panels.

## 9.9. Openings

The options in the dialog box below allow the user to take account of the reduction of rigidity of walls including openings (windows, doors) in the case of simplified calculations. The dialog box opens on pressing the ***Openings*** button in the Calculation Options dialog box.

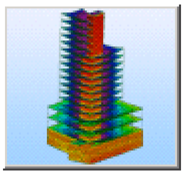



If there are openings in a wall and in the above dialog box the option which allows considering openings in stiffness calculations is switched on, then the program checks if openings intersect the level defined in the options. If they do intersect, then the equivalent wall stiffness is calculated according to the formula below:

$$J_{eq} = \frac{b \cdot H^3}{12} - \sum_i \left( \frac{b \cdot h_i^{\,3}}{12} - b \cdot h_i \cdot l_i^{\,2} \right)$$

where:
b - wall width
H - wall length
h_i - length of the i opening (segment on the level defined in the dialog box)
l_i - distance between the center of the i opening (segment) and the center of the wall.
If a wall belongs to a group, openings are taken into account in calculations of the center of gravity and the center of torsion. A wall with openings is treated as a solid wall of reduced thickness (the reduction of thickness is directly proportional to the reduction of rigidity).

# 9.10. Calculation - Calculation Options (Exact Method)



The method is applied after pressing the icon that represents the exact calculation method in the ***Calculation options*** dialog box. This option is not accessible for the analysis of a single story.
In this case the FEM analysis comprises both vertical and horizontal forces.

This method consists in generation of a full FEM model considering all the elements and loads. There is a possibility to steer with the mesh size by means of the *Meshing* option in the ***Calculation options*** dialog box. The coarse meshing results in generation of a fairly small number of the elements (approx. 2-8 finite elements per planar element), which enables quick analysis of large structures. The *Element size* option allows determining a preferred dimension of the finite element.
If the *Required reinforcement of slabs and walls* option is switched on, then in course of structure static and dynamic analyses the program will calculate the required reinforcement area for all concrete slabs and walls (panels made of materials different than those from the concrete group in the material database will not be included in calculation of the required reinforcement). Switching on the *Consider selection* option results in calculation of the required reinforcement of selected panels (slabs). Pressing the **Code parameters** button opens the ***Code parameters*** dialog box; the options provided in this dialog box depend on a selected RC design code.

In the lower part is the *Use load report of plates* option. The option is used to perform advanced calculations after distributing loads applied to plates onto elements supporting them (walls, columns, beams); once supports and directions of load transfer are selected, simplified calculations are performed (then distribution of the loads applied to plates onto elements supporting them is obtained), for thus-assumed system exact calculations are carried out.

*NOTE:* *The* Use load report of plates *option is available only after performing structure calculations by means of one of the simplified methods; if a structure has not been calculated using one of the simplified methods, the option is not available; the name of the option depends of the selected simplified method:*
*for the trapezoidal and triangular method:* Use load report of plates according to the trapezoidal and triangular method
*for the simplified method (finite element mesh):* Use load report of plates according to the simplified FE method.

All vertical loads (including the self-weight of structure elements) are distributed; NOTE: plates are treated as weightless and fulfill the role of stiffening membranes in the structure, on the levels of individual stories.
Note should be taken that if a wall supported only on a plate (and not on a beam) is used in the structure, there may appear discrepancies between results of internal forces obtained in simplified and advanced calculations for individual elements (it refers mainly to columns and walls, since in that case a part of loads applied to the wall is distributed on the plate because there is no beam to carry it).

*NOTE:* *The self-weight and loads of stairs are taken into account during calculations of a structure by means of the advanced method; stairs are treated as a slab, while all stair loads as slab loads.*

*NOTE:* *If a linear support of a slab (beam or wall) is switched off on the support list, then the directions Rx and Uz on this edge will be released in calculations using the exact method.*

If the option for automatic generation of wind is switched on (in the ***Default Loads*** dialog box), then for it to be taken into account in the advanced calculations of a structure, the *Generate wind loads* option needs to be activated in the ***Calculation Options*** dialog box (for the advanced method).

In the lower part the user may also decide which program will be used during structure calculations by means of the exact method: ***ROBOT Kernel*** or ***ROBOT Millennium***.
Differences in the use of these programs during calculations are concerned with:

1. method of performing calculations
   ***ROBOT Kernel***: if calculations are performed by means of the ***ROBOT Kernel*** program, then the ***ROBOT Millennium*** calculation kernel is used, however, without the possibility of graphical support of obtained results (it is impossible to check the results in ***ROBOT Millennium***)
   ***ROBOT Millennium***: if calculations are performed by means of the ***ROBOT Millennium*** program, then after calculations are complete, it is possible to:
   - check the results obtained
   - mark the objects for which warnings were displayed in ***ROBOT Millennium***
   - save them to an .rtd file
   - make modifications and carry out the analysis in ***ROBOT Millennium***

2. scope of results
   after calculations performed using the ***ROBOT Kernel*** program, additional calculation results are achieved in ***CBS Pro***:
   values of reduced forces in panels for seismic cases, Newmark combinations and envelopes of accidental combinations

3. duration of calculations and getting results
   duration of calculations is identical, while getting calculation results from ***ROBOT Millennium*** takes longer than for calculations performed using ***ROBOT Kernel***.

The final result of this analysis are:
- maps and values of moments, forces and displacements for slabs and walls

*NOTE:* *The displacements presented are elastic ones (without taking account of cracking of RC elements). Values of deflections of cracked slabs as well as values of the crack width are not loaded into* **CBS Pro**, *but verified during design of the required reinforcement of panels in the* **ROBOT** *program.*

- diagrams and values of moments, forces for bar elements
- displacements of a structure

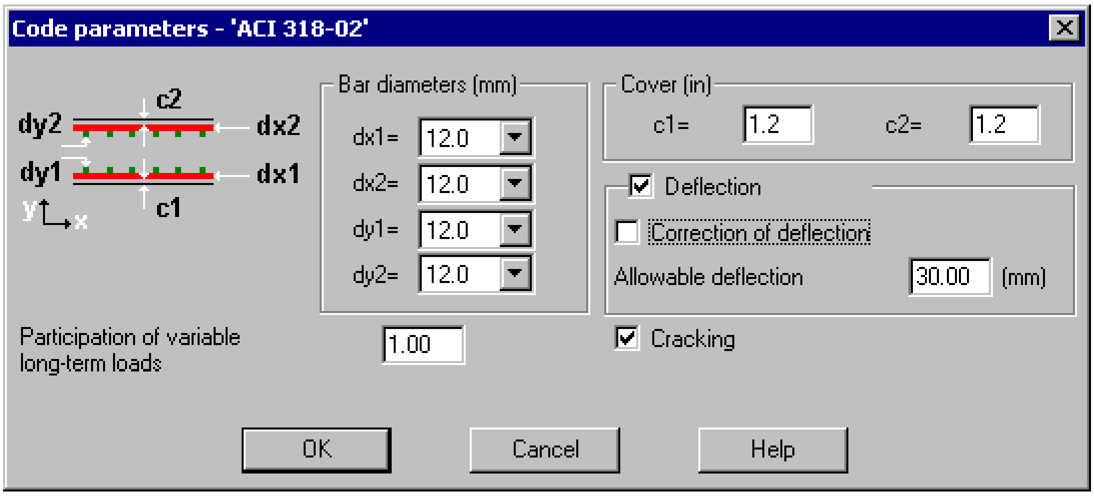
*NOTE:* *The displacements presented are elastic ones (without taking account of cracking of RC elements).*

- support reactions
- required reinforcement areas of concrete slabs and walls.

Moreover, this method allows calculation of the provided reinforcement for RC slabs; calculations of the provided reinforcement area of slabs are performed by ***ROBOT*** RC modules.
The local system of raft foundations is directed opposite to the axis Z of the global system when the provided reinforcement is calculated; this is done to ensure that punching is taken into account. Thus reinforcement layers are presented in the opposite way than in the case of regular slabs.

# 9.11. Code Parameters - Required Reinforcement of Slabs and Walls

The dialog box that opens on pressing the **Code parameters** button in the ***Default Options*** dialog box (for the exact method). The options located in this dialog box enable definition of code parameters necessary to perform required reinforcement calculations according to the conditions described in a selected RC code (see Preferences); apart from that, they are set while performing calculations by means of the ***ROBOT*** program through definition of a new reinforcement template and assigning it to all concrete panels. Thanks to this option it is possible to carry out reinforcement calculations for other parameters than the standard ones set in the ***ROBOT*** program.




The above dialog allows determining basic parameters connected with calculation of the required reinforcement area of structure slabs and walls. There is a possibility to select bar diameters for top and bottom reinforcements and to define a reinforcing bar cover.
If the *Deflection* option is switched on, the program will calculate a value of deflection of an RC slab considering cracking, and switching on the *Correction of deflection* option allows automatic correction of deflections by increasing the reinforcement area (reducing deflection below the admissible value).
After switching on the *Cracking* option the cracking width is calculated, and switching on the *Correction of cracking* option makes possible automatic correction of the cracking width by increasing the reinforcement area (reducing cracking below the admissible value).

*NOTE:* *Values of deflections of cracked slabs as well as values of the crack width are not loaded into* **CBS Pro**, *but are verified during design of the required reinforcement of panels in the* **ROBOT** *program. If calculated values of deflections exceed allowable values, then a message for an appropriate object is displayed.*

*NOTE:* *If calculations by means of the exact method have been performed, then it is possible to calculate the required reinforcement of slabs and walls without the necessity to carry our static calculations again.*

# 9.12. Assumptions Adopted in the Structure Calculation and the Algorithm for Calculation of Moments

**ASSUMPTIONS**

- Walls carry horizontal forces only in their plane
- Columns carry loads in both directions
- Non-structural elements or elements made from non-structural materials do not carry loads
- Spandrel beams of the frame (floors of the stories) are infinitely rigid
- Moments on vertical elements are distributed in proportion to their stiffness.

NOTE: If a structure model includes non-structural elements other than partition walls, then after starting calculations, the following message appears: 'Non-structural elements (*list*) will be disregarded in calculations'. The following rules apply to non-structural elements in the program: - spread footings / continuous footings are generated under unsupported columns / non-structural walls
- the weight of disregarded non-structural elements is not included in the structure self-weight
- non-structural elements are disregarded during design of structure elements.

**ALGORITHM FOR CALCULATION OF MOMENTS**

1. Calculation of the moments of inertia I'x, I'y, I'xy for every cross-section of vertical elements (columns + walls)
2. Transformation of the moments of inertia to the global system, in which the following forces are acting:

$$\begin{vmatrix} I'x & I'xy \\ I'xy & I'y \end{vmatrix} \longrightarrow \begin{vmatrix} Ix & Ixy \\ Ixy & Iy \end{vmatrix}$$

3. Summing up the forces on each story from *1* to *k* and calculation of the total moment acting on the story *i*

$$M_i = h_i \cdot \sum_{n=0}^{i} F_{k-n}$$

4. Summing up the stiffness moments B of all nodes (ends) of the considered columns and walls *w* from the story *i*

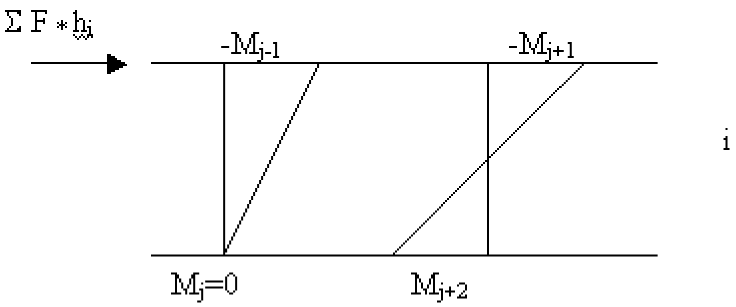
*Li* – height of a column or a wall
*hi* – story height
*Ii* - moment of inertia in the direction of the current calculation
*Ei* - Young's modulus
XXX – fixed support
OOO – pinned support

$$B_j = \begin{cases} \begin{array}{ll} \dfrac{6 \cdot E_j \cdot I_j}{L_j^{\ 2}} & \Leftarrow \begin{array}{l} XXX \\ XXX \end{array} \\ \dfrac{6 \cdot E_j \cdot I_j}{L_j^{\ 2}} & \\ \dfrac{3 \cdot E_j \cdot I_j}{L_j^{\ 2}} & \Leftarrow \begin{array}{l} XXX \\ OOO \end{array} \\ 0{,}0 & \\ 0{,}0 & \Leftarrow \begin{array}{l} OOO \\ OOO \end{array} \\ 0{,}0 & \end{array} \end{cases}$$

$$Bk_i = \sum_{j=1}^{w} B_j$$

5. Distribution of the moment *Mi* onto all nodes *j*, in proportion to their stiffness

$$M_j = \frac{M_i}{B_i} \cdot B_j$$

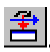


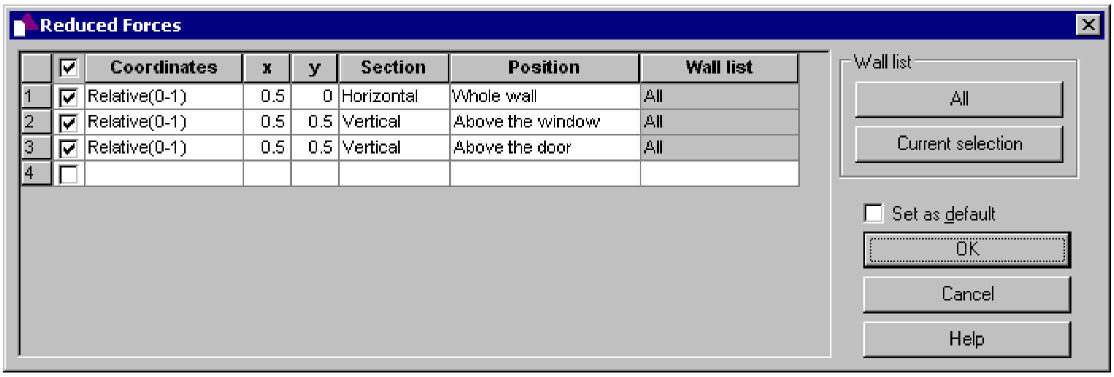

# 9.13.Reduced Forces

The ***Reduced Forced*** dialog box is used for defining cuts through walls (horizontal, vertical) and points at which the program will calculate the reduced forces V, H, M.

*NOTE:* *These forces may be presented in the* **Object Properties** *dialog box or in the computational 3D view only after calculations by means of the exact method are performed.*

The option is available:

- from the menu after choosing the *Calculations / Reduced forces* command
- after pressing the icon .




Each of defined cut types may be ascribed to all walls or only to selected walls. A list of wall cuts and points at which reduced forces are calculated constitutes a template; it may be saved as a default one, and next used in new projects.

Presentation of reduced forces in the ***Object Properties*** dialog box is possible after the *Reduced forces* option is activated on the *Results* tab. A successive point is selected by means of arrows. It is also possible to evaluate values of reduced forces for any point in a wall; then coordinates of the point should be typed into the edit fields.
Presentation of reduced forces in the 3D view is possible after selecting a reduced value. It displays values of forces at all predefined points.

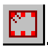
The above dialog box contains a table whose columns include:

- first column is used to switch on/off a selected cut type
- coordinates: selection of a type of position definition (Relative / Absolute)
- x, y: coordinates of the cut point
- Section: available cut directions (horizontal / vertical)
- Position: available zones of definition of cuts (whole wall / above the window / above the door)
- Wall list: all / current selection; this column shows walls selected before the dialog box is opened.

# *10. DESIGN OF STRUCTURE ELEMENTS*

## 10.1.Design of RC Elements of a Structure

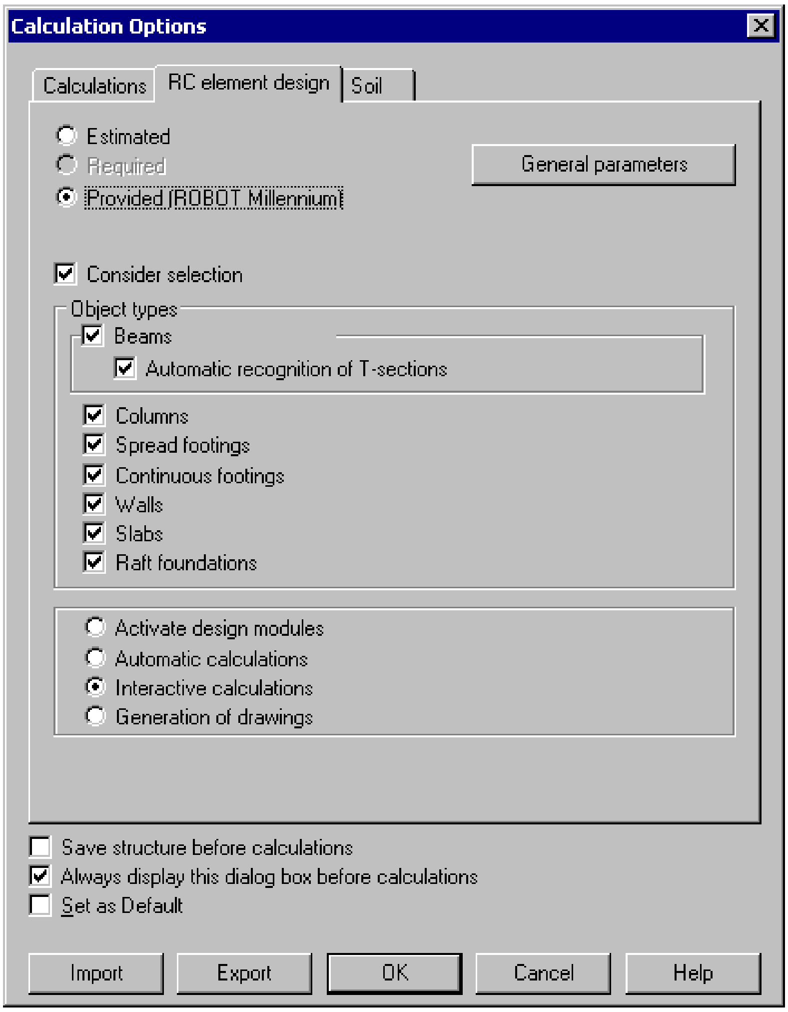
The code calculation of structure elements in the ***CBS Pro*** program can be performed for the RC elements of a structure. To start the code calculation of structure elements, the user should:

- select the *Calculations / RC Element Design* command from the menu
- press the icon .

If in the ***RC element design*** dialog box the option *Always display this dialog box before calculations* is switched on, then the ***RC element design*** dialog box that contains the options of the code calculation of the structure and the soil parameters, appears on the screen; to start the code calculation of the structure elements, press the **Calculate** button.

## 10.2.RC Element Design - Calculation Options

The ***RC element design*** dialog box shown in the drawing below opens after selecting the *Calculation Options* option from the menu (and then moving to the *RC element design* tab) or after selecting the *RC Element Design* option from the menu (with the option *Always display this dialog box before calculations* switched on).



In the top part of the dialog box the following design method may be chosen:
- estimated
- provided (with the use of ROBOT RC modules).

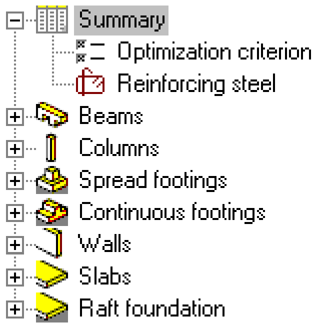
Design parameters are selected in the General parameters dialog box, which opens on pressing the **General parameters** button in the top right part of the dialog box.
The *Consider selection* option makes possible design of the elements selected in the graphic viewer or through the options provided in the Criterion of Selection dialog box. The *Consider selection* option is linked with the mode of selection of the whole structure or the current story accessible in the *Edit* menu or on the *Objects* toolbar ( or ).

The middle part of the dialog box holds a list of objects available in the program, for which the code calculation may be performed; the following types of RC structure elements can be designed in the ***CBS Pro*** program: beams (while calculating the provided reinforcement of RC beams in ***ROBOT***, it is possible to detect the width of a slab that may be taken into account in the element design - the *Automatic recognition of T sections* option), columns, spread and continuous footings, walls, slabs and raft foundations.
After switching on e.g. the *Beams* and the *Columns* options (with the remaining options switched off), the designed objects of an RC structure are only beams and columns.

*NOTE:* *If calculations by means of the exact method have been performed, then it is possible to calculate the required reinforcement of slabs and walls without the necessity to carry our static calculations again.*

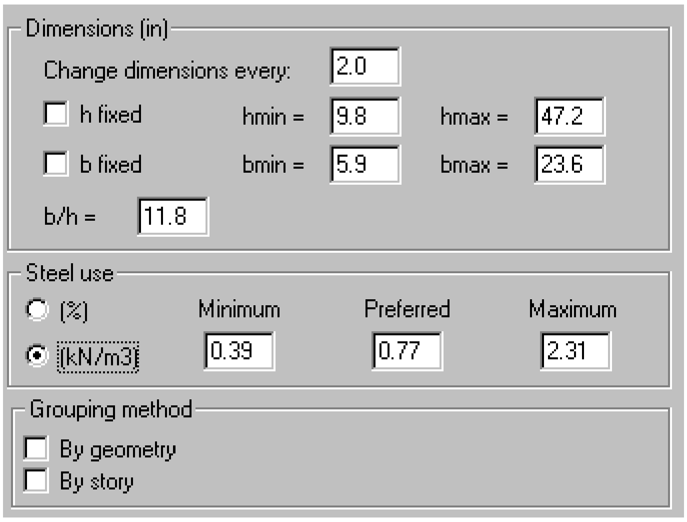
## 10.3.RC Element Design - General Parameters

Design parameters are selected in the ***General Parameters*** dialog box that opens on pressing the **General parameters** button in the RC element design dialog box.
The left-hand part of the dialog box contains the tree of options available in the dialog box, presented in the drawing below.

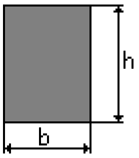



After expanding each option in the tree it is possible to define the optimization criterion and the parameters of reinforcing steel for every type of the element of an RC structure.
The *Summary* option at the top of the tree enables presenting basic, global information about the design options for individual structure objects. These parameters may not be modified on the *Summary* tab.
The successive elements of the tree from the left-hand part of the ***General Parameters*** dialog box make it possible to determine optimization criteria and design parameters for individual types of objects. The following types of elements of RC structures are available: beams, columns, spread footings, continuous footings, walls, slabs and raft foundations.
Below are discussed the options for individual elements of RC structures.

Optimization Criteria
Optimization criteria may be determined for every type of the element of the RC structure.




**BEAMS, COLUMNS**




The following parameters can be specified in the *Dimensions* field:

- *Change dimensions every* – defines a value of the minimum geometry change in course of section optimization

- options switched on: *h fixed*, *b fixed* – optimization of the section dimensions is impossible (the section retains dimensions adopted by the user)

- options switched off: *h fixed*, *b fixed* – it is possible to modify the dimensions b and/or h of the beam cross section and to determine the limit values of the both dimensions: bmin, hmin as well as bmax and hmax

- options switched off: *h fixed*, *b fixed* – it is possible to determine the dimension ratio b/h.

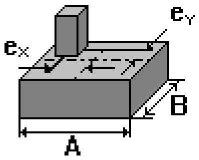
The following parameters can be specified in the *Steel use* field:

- *Minimum*, *Maximum* – in the section optimization it allows determining the bottom and the top values of the interval that represents either the minimal/maximal steel percentage per RC section, or the minimal/maximal mass of steel per one cubic meter of concrete (density)

- *Preferred* – the steel percentage per RC section or the mass of the steel per one cubic meter of concrete (density) preferred by the user and aimed at by the section optimization (if modification of at least one dimension is enabled).

The *Grouping method* field allows selection of the method of grouping RC beams: grouping within the elements of the same geometry and/or grouping within the elements on the same story.
For RC beams it is possible to force uniform height and width of the optimized sections of the spans within one beam using the options: *Uniform height / width of span sections*.

## SPREAD FOOTINGS




The following parameters can be specified in the *Dimensions* field:

- *Change dimensions every* – defines a value of the minimum geometry change in course of the section optimization

- options switched on: *A fixed*, *B fixed*, *ex fixed*, *ey fixed* - optimization of the section dimensions is impossible (the section retains dimensions adopted by the user)

- options switched off: *A fixed*, *B fixed*, *ex fixed, ey fixed* - it is possible to modify dimensions of the cross section and to determine the limit values of the dimensions: Amin, Bmin as well as Amax and Bmax

- minimum steel use (the check of the amount of steel designed by the program and the minimum amount specified by the user only in the case of estimated design).

The *Grouping method* field allows selection of the method of grouping of spread footings (grouping within the elements of the same geometry).

### CONTINUOUS FOOTINGS

The following parameters can be specified in the *Dimensions* field:

- *Change dimensions every* - defines a value of the minimum geometry change in course of the section optimization

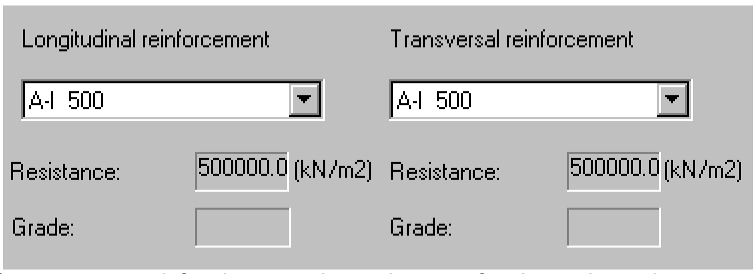
- activated options: *B fixed*, *ex fixed* - optimization of the section dimensions is impossible (the section retains dimensions adopted by the user)

- options *B fixed*, *ex fixed* switched off - it is possible to modify dimensions of the cross section and to determine the limit values of the dimensions $B_{min}$ and $B_{max}$.

The *Grouping method* field allows selecting the method of grouping continuous footings (grouping refers to elements of the same geometry).

Reinforcing steel

Reinforcement parameters may be determined for each type of the element of the RC structure.




For beams, columns, spread footings and continuous footings there is a possibility to select parameters of the longitudinal and transversal reinforcement.

For walls, slabs and raft foundations it is possible to choose parameters of the main and structural reinforcement.

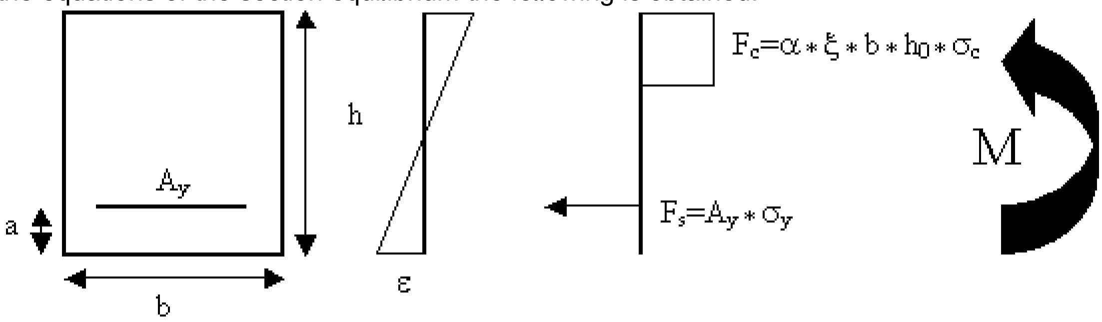
The reinforcement available on the selection lists is the reinforcement defined in the ***Material Database*** dialog box.

## 10.4.RC Element Design - Estimated Calculations

The calculations are performed according to the general rules of the design of RC elements in the Ultimate Limit State and the Serviceability Limit State. Based on the forces for these states as well as the resistance parameters of steel and concrete estimated areas of the element's reinforcement at several characteristic points are obtained. They are the basis for estimation of the steel requirement of the whole element.
These results are sufficient to estimate the steel requirement of the whole structure and allow quick optimization of the element dimensions. In spite of the simplified method of the calculation, an average value of the reinforcement in the element should not differ from the results for the provided reinforcement by more than 10%.

**BEAMS (BENDING)**

Due to positive and negative moments, the calculations are independent. The final reinforcement is an envelope of the reinforcement from the design for positive and negative moments.
Calculation parameters are the section dimensions:
*b* – section width
*h* – section height
*a* – average distance between the edge and the centroid of longitudinal bars of the reinforcement
and material parameters
fck – characteristic resistance of concrete
fyk - characteristic resistance of steel
γc - material coefficient for concrete in ULS
γy - material coefficient for steel in ULS
γc' - material coefficient for concrete in SLS
γy' - material coefficient for steel in SLS.
The first calculation step is to determine the limit moment for unilateral reinforcement. From the equations of the section equilibrium the following is obtained:




$$F_c = \alpha \cdot \xi_{\lim} \cdot b \cdot h_0 \cdot f_{cd}$$
$$M_{\lim} = F_c \cdot h_0 \cdot (1 - 0.5 \cdot \alpha \cdot \xi_{\lim})$$

where:

$$f_{cd} = \frac{f_{ck}}{\gamma_c} \quad f_{yd} = \frac{f_{yk}}{\gamma_y}$$
$$h_0 = h - a$$

For ULS in the calculations the following coefficients are assumed:
ξ lim = 0.5
α = 0.8
γc = 1.5
γy = 1.15.
In the case Mlim ≤ |M $^{ULS}$|, an additional moment is calculated that has to be carried by the top reinforcement.

$$M_{add} = \left|M^{ULS}\right| - M_{\lim}$$

$$A_{y,add} = \frac{M_{add}}{(h - 2 \cdot a) \cdot f_{yd}}$$

$$A_{y\lim} = \frac{F_c}{f_{yd}}$$

The final reinforcement can be expressed by the formulas below:

$$A_y^{(+)} = A_{y,add}$$

$$A_y^{(-)} = A_{y\lim} + A_{y,add}$$

In the case Mlim ≥ |M $^{ULS}$|, the quadratic equation of the following parameters is solved:

$$Ax^2 + Bx + C = 0$$

$$A = -0.5 \cdot b \cdot f_{cd} \quad B = b \cdot f_{cd} \cdot h_0 \quad C = -\left|M^{ULS}\right|$$

$$A_y = \min\begin{pmatrix} x_1 \\ x_2 \end{pmatrix} \cdot \frac{b \cdot f_{cd}}{f_{yd}}$$

however, depending on the sign of the moment, Ay is the top or the bottom reinforcement.

The calculation for ACC proceeds in the identical way as for ULS as described above. In the equations the following are only substituted:
1. moments from the envelope for SLS, acting on the section
2. material coefficients (γc' = 1.0, γy' = 1.0).

**COLUMNS (COMPRESSION)**

In the calculation it has been assumed that the load capacity of the section is a sum of the load capacity of the reinforcement and the reduced load capacity of the RC section for compressive forces

$$N_{Rd} = A_{yc} f_{yd} + \xi \cdot b \cdot h \cdot f_{cd}$$

or the load capacity of steel for tensile forces:

$$N_{Rd} = A_{yc} f_{yd}$$

In the case there occur both the tensile and the compressive forces, the calculation is performed independently for the both load types, and the final reinforcement is an envelope of the reinforcement resulting from these calculations.

The element slenderness is accounted for by the additional reduction factor φ, that increases the action of the axial force:

$$N_{Rd} = \left(A_{yc} f_{yd} + \xi \cdot b \cdot h \cdot f_{cd}\right) \cdot \varphi$$

$$\varphi \le 1.0$$

where:

$$\varphi = \frac{1}{1 + \lambda_w^{\ 3}}$$

$$\lambda_w = \frac{\lambda}{\lambda_{gr}}$$

$$\lambda = \frac{l_c}{i} \quad i = \sqrt{\frac{I}{A}} \quad l_c = \beta \cdot l$$

$$\lambda_{gr} = 50$$

The following designations have been adopted:
I - moment of inertia of the section
A – section area

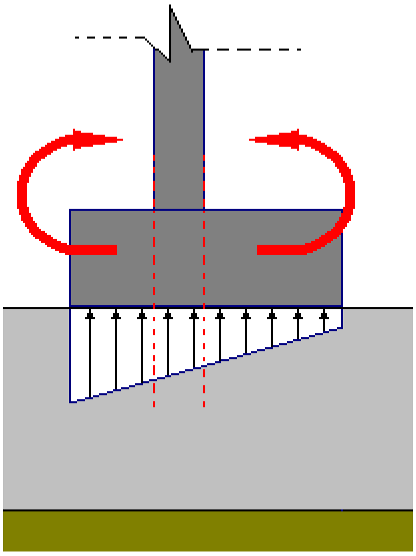
l – column height
$\beta$ - buckling coefficient.

In addition, the following have been adopted in the calculations:
- material coefficients as for the calculation of beams (bending)
- buckling coefficient $\beta$=0.7
- reduction factor for the load capacity of concrete $\xi$ = 0.5.

If bending moments occur in a column, then the bending reinforcement is calculated independently, and afterwards, the bending reinforcement and the compression reinforcement are added up. In the bending calculation the bending moment is also increased by dividing its value by the slenderness coefficient $\varphi$.

**SPREAD FOOTINGS (BENDING)**

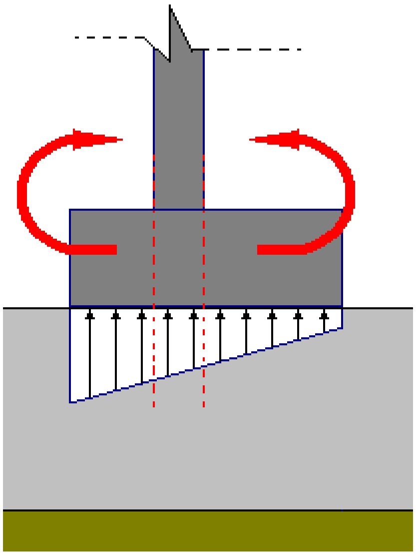
Reinforcement in spread footings is calculated on the basis of the bending moments that are caused by the action of the soil under a spread footing. The impact of the soil on a foundation results from the calculation of the load capacity of the foundation for ULS.



The principle of the bending design is identical as for beams. The calculations are performed independently for both directions.

**CONTINUOUS FOOTINGS (BENDING)**

Reinforcement in continuous footings is calculated on the basis of the transverse bending moments that are caused by the action of the soil under a continuous footing. The impact of the soil on a foundation results from the calculation of the load capacity of the foundation for ULS.



The principle of the bending design is identical as for beams.
NOTE: Longitudinal reinforcement design is not performed for continuous footings.

**SLABS, RAFT FOUNDATIONS, WALLS**

An analysis of these elements in a model calculated by means of simplified methods is not possible.
However, when calculating required reinforcement areas of panels for advanced calculations, the reinforcement mass is determined through estimation - for individual panels the min./max. reinforcement areas are calculated in 1 meter large mesh cells. Thus calculated reinforcement does not take account of the unification of reinforcement zones and lengths of lap splices, therefore, it may diverge from the reinforcement evaluated in ***ROBOT*** RC modules.

## 10.5.RC Element Design - Provided Calculations

In the calculation of provided reinforcement the RC modules (calculation of the provided reinforcement area) of the ***ROBOT*** program are used. This calculation is based on the code requirements. The result of this calculation is a table of the reinforcing steel that satisfies the conditions of the load capacity and the code requirements, together with a reinforcement drawing and a calculation note.
The calculation may be performed in the following modes:

- activation of code modules - a module for design of an RC structure element will be activated and code calculations of this element may be performed there; it is possible to modify parameters of optimization and reinforcement.
- automatic – the code calculation is performed automatically for the assumed parameters of optimization and reinforcement
- interactive – the code calculation is performed for the assumed parameters of optimization and reinforcement, with the possibility to modify these parameters
- with generation of drawings – the code calculation is performed automatically for the assumed parameters of optimization and reinforcement with additional generation of drawings of an RC structure's elements.

*NOTE:* *When designing RC beams analyzed using the advanced method, relative elastic displacements of beams (reduced by displacements of their supports) are transferred to the* **ROBOT** *RC module; it ensures correct design of beams for SLS.*

While calculating the provided reinforcement of RC beams in the ***ROBOT*** program, it is possible to detect the width of a slab that may be taken into account in the element design (the *Automatic recognition of T sections* option). The following assumptions are adopted for the beam design considering the slab that works together with the beam:

- the beam section is rectangular
- the slab width is considered independently for all spans of the beam
- for each span, slabs connected to the span and the neighboring beams are found (only beams that form an angle less than 45 degrees with the span in question are considered)
- an overhang of the slab considered in calculations equals the half of the least distance between the neighboring beam and the span in question, and if there are no neighboring beams, this is the least slab overhang within a given span (not greater, however, than the 10-fold thickness of the slab)
- a section of the designed element is read in the ***ROBOT*** RC module and may have 2 different slab overhangs (on different levels)
- a slab overhang that may be considered in the ***ROBOT*** RC module is compared with requirements of individual RC codes and the overhang allowed by a code is considered in the calculations (the yellow line in a section drawing indicates the range of the slab overhang allowed by a code)
- if element dimensions are changed during its design, then only the dimensions b and h of the rectangular section are modified (and sent back to ***CBS Pro***), while (user-made) modifications of the slab that works together with the element are disregarded.

The calculation of provided reinforcement areas in the ***CBS Pro*** program is performed according to the current settings made in the ***ROBOT*** program with respect to the code as well as the *Standard* preferences for the reinforcement parameters and the calculation parameters not defined in the ***CBS Pro*** program. The user may select a template of calculation options or reinforcement different than the standard one, if it has been defined earlier in ***ROBOT*** RC modules. It may be selected on the selection lists provided in the ***General Parameters*** dialog box for individual object groups.
All the parameters defined in the ***RC element design /General Parameters*** dialog box are transferred to the RC modules of the ***ROBOT*** program.
The provided reinforcement of walls is designed in the ***ROBOT*** RC module **Walls**. For structures calculated using simplified methods, loads applied to walls through distribution of loads are exported to the wall module, while in the case of structures calculated by means of the advanced method, forces reduced in walls for the horizontal section in the center of the wall base are exported to the wall module.
The calculation of the provided reinforcement allows for the design of beams, columns, deep-beams, walls, spread and continuous footings and slabs, taking the requirements of the following codes into account:

| | |
|---|---|
| *Beams*: | ACI 318/99 and ACI 318/99 metric, ACI 318/02 and ACI 318/02 metric, ACI 318/02 Thailand, BAEL 91 and BAEL 91 mod. 99, BS 8110, EC2 (ENV 1992-1-1:1991), EC2 - Belgium NAD, EC2 - Italian NAD, PN-84/B-03264 and PN-B-03264 (2002), EHE 99, NS 3473E, CP 65, CSA A23.3-94, DM 9/1/96, SNiP 2.03.01-84, STAS 10107/0-90 |
| *Columns*: | ACI 318/99 and ACI 318/99 metric, ACI 318/02 and ACI 318/02 metric, ACI 318/02 Thailand, BAEL 91 and BAEL 91 mod. 99, BS 8110, EC2 (ENV 1992-1-1:1991), EC2 - Belgium NAD, EC2 - Italian NAD, EHE 99, PN-84/B-03264 and PN-B-03264 (2002), SNiP 2.03.01-84, NS 3473E, STAS 10107/0-90, DM 9/1/96, CP 65 and CSA A23.3-94 |

*Continuous footings*: ACI, BS 8004, CSA, EC7 (ENV 1997-1:1994), DTU 13.12, Fascicule 64 Titre V, PN-81/B-03020, SNiP 2.02.01-83

*Spread footings*: ACI, BS 8004, CSA, DTU 13.12, EC7 (ENV 1997-1:1994), Fascicule 64 Titre V, PN-81/B-03020, SNiP 2.02.01-83

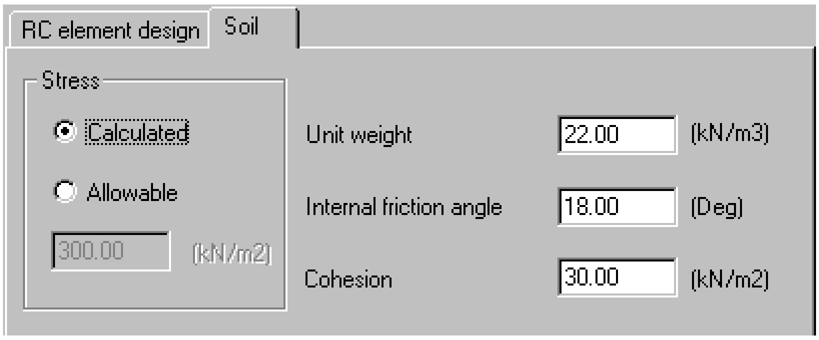
*Slabs*: ACI 318/99 and ACI 318/99 metric, ACI 318/02 and ACI 318/02 metric, ACI 318/02 Thailand, BAEL 91 and BAEL 91 mod. 99, BS 8110, EC2 (ENV 1992-1-1:1991), EC2 - Belgium NAD, EC2 - Italian NAD, EHE 99, PN-84/B-03264 and PN-B-03264 (2002), SNiP 2.03.01-84, NS 3473E, STAS 10107/0-90, DM 9/1/96, CP 65, CSA A23.3-94 and GB 50010-2002

*Deep-beams*: BAEL 91 and BAEL 91 mod. 99, PN-B-03264 (2002)

*Walls*: ACI 318/99 and ACI 318/99 metric, ACI 318/02 and ACI 318/02 metric, ACI 318/02 Thailand, BAEL 91 and BAEL 91 mod. 99.

## 10.6.Soil - Calculation Options

The Calculation Options dialog box also contains the *Soil* tab.




This tab is used to define parameters of the geotechnical calculations of foundations. A value of the allowable stress that is used in the geotechnical calculations of foundations may be determined in this dialog box:

- after selecting the *allowable* option and specifying the value of the allowable stress
- after selecting the *allowable* option and specifying values of the basic parameters of the soil (unit weight, internal friction angle and cohesion); the value of the allowable stress is calculated by the program.

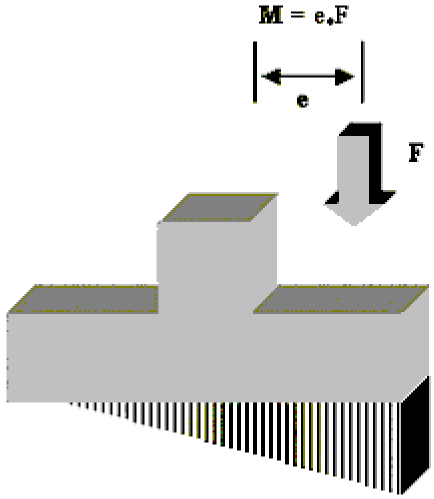
*NOTE:* *The actual foundation depth results from geometry of a model and a terrain level determined by the user (it is constant for the whole model).*
*The minimum foundation depth is not provided on the list of soil parameters since it is not checked by the program if this condition is fulfilled. this condition should be checked by the user.*

## 10.7.Analysis of the Load Capacity of a Foundation

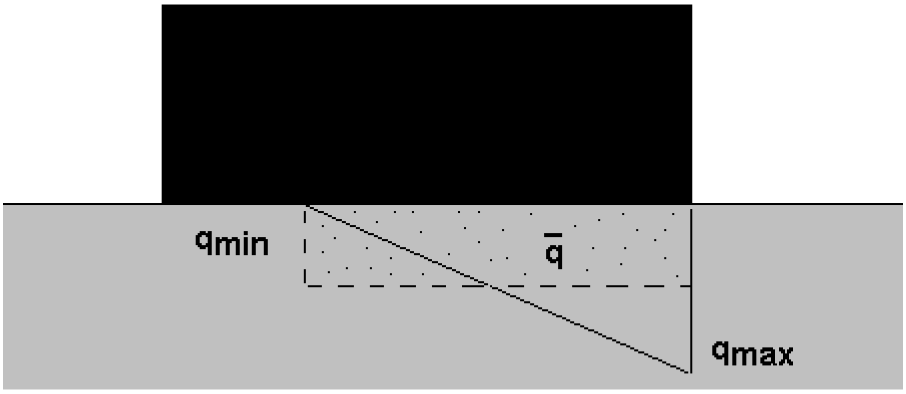
The simplified analysis of the load capacity of foundations in ***CBS Pro*** is based on comparing values of stresses in the soil or forces (for the codes EC7 (ENV 1997-1:1994), PN-81/B-03020, SNiP 2.02.01-83) for a selected code with the allowable value qult modified as in the table below.
The allowable value qult may be determined by the user in the ***Calculation Options*** dialog box on the *Soil* tab or calculated on the basis of the soil properties given in the same dialog box.

| Code | Calculated stress | Allowable stress |
|---|---|---|
| ACI, BS 8004, CSA | $\frac{q_{ult}}{3} \ge \overline{q}_{SLS}$<br>$\overline{q}_{SLS} = 0{,}5 \cdot (q_{max\_SLS} + q_{min\_SLS})$ | $\frac{q_{ult}}{1.0} \ge q_{SLS\_max}$ |
| DTU 13.12<br>Fascicule 64 Titre V | $\frac{q_{ult}}{2} \ge q_{ULS\_med\_max}$<br>$\frac{q_{ult}}{1{,}5} \ge q_{ALS\_med\_max}$<br>$q_{med\_max} = 0{,}75 \cdot q_{max} + 0{,}25 \cdot q_{min}$ | $\frac{q_{ult}}{2} \ge q_{ULS\_med\_max}$<br>$\frac{q_{ult}}{1{,}5} \ge q_{ALS\_med\_max}$<br>$q_{med\_max} = 0{,}75 \cdot q_{max} + 0{,}25 \cdot q_{min}$ |
| EC7 (ENV 1997-1:1994) | $\frac{R_d}{V_d} \ge 1{,}0$ | $\frac{q_{ult}}{q_{max}} \ge 1{,}0$ |
| PN-81/B-3020 | $\frac{m \cdot Q_{fN}}{Q_r} \ge 1{,}0$ | $\frac{1{,}2 \cdot m \cdot q_{ult}}{q_{max}} \ge 1{,}0$<br>$\frac{m \cdot q_{ult}}{\overline{q}} \ge 1{,}0$<br>$\overline{q} = 0{,}5 \cdot (q_{max} + q_{min})$ |
| SNiP 2.02.01-83 | $\frac{F_u \cdot \gamma_c}{F} \ge \gamma_n$<br>$\gamma_c = 0{,}9$ ; $\gamma_n = 1{,}0$ | $\frac{q_{ult} \cdot \gamma_c}{\overline{q}_{ULS}} \ge \gamma_n$<br>$\overline{q}_{ULS} = 0{,}5 \cdot (q_{max\_ULS} + q_{min\_ULS})$ |

To calculate the stresses under a spread footing caused by external loads, the weight of the foundation and the overlying soils, the program employs a linear model that does not allow for accounting for the occurrence of tensile stresses in the soil.




In the calculation an average value of the stresses is applied; this value is understood as the average value of the stresses, not including the zone where zero stresses occur.
For some codes the average maximum value (see the table above) has been used.




The allowable value qult can be determined by the user in the ***Calculation Options*** dialog box on the *Soil* tab, or calculated from the soil properties specified in this dialog box.
For the calculation of allowable stresses based on the soil properties Hansen’s method has been adopted together with the guidelines for this method presented in ‘Foundation Analysis and Design’ Joseph E. Bowles, Fifth Edition, The McGraw-Hill Companies, Inc. 1996.
The basic formula for the load capacity by Hansen has been limited to the following case: it has been assumed that the coefficients responsible for the inclination of the spread footing ‘b’ and the backfill slope ‘g’ equal 1.0.
Since it is not allowed for in the ***CBS Pro*** program to use soils with the friction angle $\phi$ = 0.0 degrees, only the first of the formulas by Hansen is applied. The final formula for calculation of the allowable stresses is presented below:

$$q_{ult} = c \cdot N_c \cdot s_c \cdot d_c \cdot i_c + \bar{q} \cdot N_q \cdot s_q \cdot d_q \cdot i_q + 0.5 \cdot \gamma \cdot B \cdot N_\gamma \cdot s_\gamma \cdot d_\gamma \cdot i_\gamma$$

where the relevant factors equal:

$$N_q = e^{\pi \cdot \tan(\phi)} \cdot \tan^2(45 + \phi / 2)$$

$$N_c = (N_q - 1) \cdot \cot(\phi)$$

$$N_\gamma = N_\gamma = 1.5 \cdot (N_q - 1) \cdot \tan(\phi)$$

$$i_q = \left[1 - \frac{0,5 \cdot H}{V + A' c_a \cot(\phi)}\right]^{2,5}$$

$$i_c = i_q - \frac{1 - i_q}{N_q - 1}$$

$$i_\gamma = \left[1 - \frac{0,7 \cdot H}{V + A' c_a \cot(\phi)}\right]^{3,5}$$

and the effective area A’ = B’ * L’

$$s_q = 1 + \frac{B'}{L'} \cdot \sin(\phi)$$

$$s_c = 1 + \frac{N_q}{N_c} \cdot \frac{B'}{L'}$$

$$s_\gamma = 1 - 0,4 \cdot \frac{B'}{L'} \geq 0,6$$

$$d_q = 1 + 2 \cdot \tan(\phi) \cdot (1 - \sin(\phi))^2 \cdot k$$

$$d_c = 1 + 0,4 \cdot k$$

$$d_\gamma = 1,0$$

where:

$$k = \begin{cases} \frac{D}{B} \le 1 & \frac{D}{B} \\ \frac{D}{B} > 1 & \arctan\left(\frac{D}{B}\right) \end{cases}$$

Whole general Hansen's method presented above is applied for the following codes: ACI, BS 8004 and CSA.
In the case of the remaining codes formulas determining some coefficients have been adapted so that they correspond to code regulations.

*NOTE:* *The ground water impact is not accounted for in the calculations.*

# *11. LINK WITH OTHER PROGRAMS*

## 11.1.Link with the ROBOT Program

The ***CBS Pro*** program offers the possibility of automatic generation of a calculation model for the ***ROBOT Millennium*** program. In order to do that, after defining a model in the ***CBS Pro*** program, the user should choose the option: *Tools / Export to the 'ROBOT' program* provided in the menu or press the icon.

*NOTE:* *If the* **ROBOT Millennium** *program is running, then a structure model will be generated in this program instance; if the* **ROBOT Millennium** *program has not been activated, then it will be run automatically and afterwards, a calculation model will generated.*

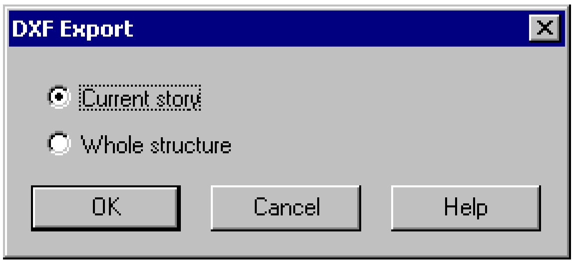
While a structure model is being generated in the ***ROBOT Millennium*** program, individual elements of a structure model of the ***CBS Pro*** program are changed to:
beams and columns to bars
spread footings to fixed supports
continuous footings to beams on elastic foundation with the default elastic foundation coefficient K
walls, slabs and stairs to panels
windows and doors to panel openings
raft foundations to panels on elastic foundation
partition walls to dead linear loads on a panel
floor cut/openings to slab openings
layered materials to dead loads on an appropriate element.
While generating a structure model in the ***ROBOT*** program, beams and columns are ascribed default member types so that it is possible to design these elements in ***ROBOT*** (depending on an element type: design of RC, steel or timber members).
A type of individual objects may be changed by switching off the *Structural element* option; if this option is switched on, such an element will be modeled in the ***ROBOT Millennium*** program only as a load of the value resulting from this element's weight.
Non-structural walls (to which a load has been applied) as well as non-structural slabs (loaded with the self-weight) are modeled in ***ROBOT*** as claddings with loads applied to them.

## 11.2.Link with Other Programs

The ***CBS Pro*** program enables saving / reading a DXF, IFC, ADD and RHG format file.

DXF format files
For the DXF format, both saving and reading options (export and import) are available. In the case of export, successive layers corresponding to object types (layer of beams, columns, etc.) are generated in a file. Objects are represented in the form of blocks (i.e. a set of lines describing e.g. a wall is grouped in a block), which makes their further processing e.g. adding dimension lines, easier.

When exporting to a DXF file, all the stories may be exported simultaneously; after selecting the DXF format, the additional ***DXF Export*** dialog box (see the drawing below) is displayed on the screen; there the user may choose whether the export concerns only the current story or all the stories at the same time (if the latter option is selected, then all the stories are placed in one drawing).




In the case of import, all objects are stored as single lines, block of lines (if they are grouped in a block in an original file) or as texts. Moreover, the program remembers layers to which these objects have been ascribed. These layers are available in the dialog box opened by selecting the menu option*: View / Display / Layers*; the standard mechanisms of filtering object display apply to layers.
A drawing is inserted to the current story; therefore, the user may open simultaneously several projections for consecutive stories. Prior to opening a file, a question is displayed on the screen, about the units which were used in generation of the model in a DXF file; selection of proper units enables them to be converted correctly to units currently applied in the ***CBS Pro*** program.
It is also possible to treat a read-in drawing as layers and to generate a 3D structure on its base. There is an advanced option available, which consists in selecting any number of drawing objects and attempting to change them automatically to 3D objects.
After selecting a line, the options concerned with line conversion to object types such as beam, wall, etc. become accessible (the options are available on toolbars, as well). If the option allowing conversion of lines to walls or partition walls is chosen, then the dialog box opens on the screen, in which the following parameters should be determined:

- maximum wall width
- maximum width of wall opening.

Definition of these parameters is required due to the adopted algorithm of converting lines to walls and automatic recognition of openings (windows, doors) in walls.

*NOTE:* *To the* **CBS Pro** *program arcs and circles are imported and they may be converted to:*
*- slabs*
*- walls, beams, continuous footings (selection of 2 lines at the minimum).*

IFC format files

*NOTE:* *If there is no program reading IFC-format files installed on the computer (IFC files are not linked to any program), then during installation the* ***CBS Pro*** *program is registered automatically in the Windows system as a default* application for IFC files.

For the IFC format, the import option is available in version 1.5.1 (Autodesk Architectural Desktop, Allplan FT), 2.0 (ArchiCAD) and 2.x. The option enables automatic import of a three-dimensional geometry of a structure model from architectural programs. In the current version of the ***CBS Pro*** program the following objects may be read in: beams, columns, slabs, walls (rectangular walls), windows, doors and slab openings.

*NOTE:* *While opening a structure model saved in an IFC format there appears a dialog box where the user should ascribe materials imported from an IFC file to individual types included in the material database of* **CBS Pro** *and determine which materials are structural.*

*NOTE:* *To the* ***CBS Pro*** *program arched walls, beams continuous footings and slabs are imported.*

*RHG format files*
This format is used for data exchange with the ***ROBOT CBS*** program. Note: when the export option is being used, it may be the case that not all the elements are exported; it results from a fact that structure definition capabilities of the ***CBS Pro*** program are more advanced than those of the ***ROBOT CBS*** program.

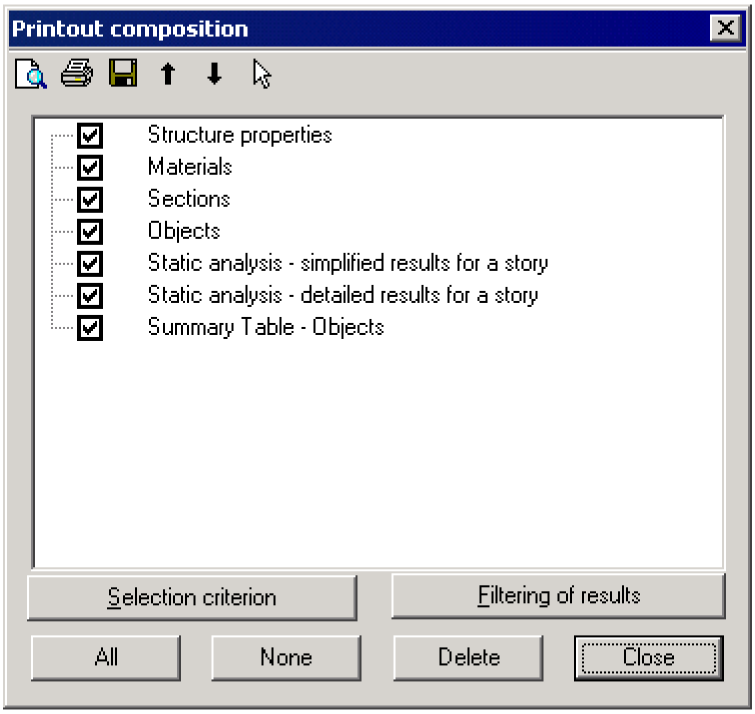
*ADD format files*
This format is used for data exchange with the ***Adcof*** program. Note: When using the export options, it may be the case that not all the elements will be exported; it results from the fact that the possibilities of structure definition in the ***CBS Pro*** program are more advanced than those in the ***Adcof*** program.

# *12. PRINTOUTS*

## 12.1.Printout Composition

The ***Printout Composition*** dialog box is used for composing the documentation of the generated model of a structure. It is possible to select components of the calculation note in the dialog box. It opens after:

- pressing the *Printout Composition* icon
- selecting the menu command: *File / Printout Composition*.




The following icons are located at the top of the dialog box:

- pressing this icon enables previewing a printout
- pressing this icon enables printing a calculation note
- pressing this icon enables saving a calculation note to a text file.
- pressing this icon enables moving a printout component up
- pressing this icon enables moving a printout component down

- activating this icon enables considering an object selection for a whole structure or a chosen story; calculation note components will be generated only for the model objects that have been selected in a structure model.

In the middle part of the dialog box there is a list of components of the calculation note; switching on an option (the √ symbol appears) means that the option will be included in the calculation note. Switching off an option indicates that the option will not be provided in the calculation note.
The following components of the note are available in the current version of the program:

- structure characteristics – information about the lower ('zero') level of the story, level of the soil (with respect to the zero story), number of stories in the building, list of the stories including their names and heights,
- materials – a list of materials used in the structure, including their parameters: resistance, unit weight and price
- sections – a list of sections used in the structure, including their drawings and dimensions
- objects – a list of objects defined in the structure, grouped by type (column, beam, slab, etc.), including their properties (material, section, position)
- loads – a list of loads defined in the structure, grouped by load type (concentrated, linear, planar)
- structure static calculations – short results
- structure static calculations – detailed results (exact distribution of a uniform load for walls, beams and the maximum values Mxx, Myy within each panel)
- structure static calculations – short results for a story
- structure static calculations - detailed results for a story (exact distribution of a uniform load for walls, beams and the maximum values Mxx, Myy within each panel)
- seismic simplified calculations (after selecting the simplified method for seismic analysis) – data adopted in the seismic calculations in the Loads – Default values / Seismic / Seismic analysis dialog box and results of the simplified calculations of the seismic analysis: vibration periods for individual directions and values of horizontal forces and displacements for directions and individual stories, respectively
- results of modal analysis (after selecting the advanced method) – values of frequency, mass participation expressed as percentage for the directions X and Y for each vibration mode
- simplified calculations of a wind load – data for calculation of the wind load according to the simplified method and values of width and height of individual stories for individual directions of action of the wind load, values of the forces applied to a story
- tables saved for printout compositions in the ***Summary table*** dialog box; NOTE: double-clicking a table name on the list of components in the ***Printout composition*** dialog box opens the Summary table dialog box where the selected table can be viewed and edited
- screen captures made in the ***CBS Pro*** program.

At the bottom of the dialog box there are the following buttons:

- **Selection criterion** - possibility of selecting objects; the Selection criterion dialog box opens on the screen

- **Filtering of results** - possibility of selecting load cases, seismic cases or combination envelopes for which results should be included in a note; the program opens the additional dialog box ***Filtering of results*** for selecting cases to be included in a note

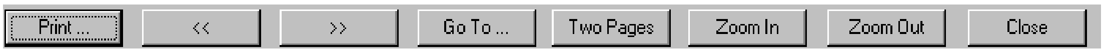
- **All** – after pressing this button, all the components of the calculation note available in the dialog box will be switched on (all the options will be included in the calculation note)

- **None** - after pressing this button, all the components of the calculation note available in the dialog box will be switched off (none of the components will be included in the calculation note)

- **Delete** - after pressing this button a highlighted component of the calculation note will be deleted.

## 12.2.Print Preview

Pressing the  icon in the ***Printout composition*** dialog box opens a preview of a created printout. At the top of the screen there is the menu shown in the drawing below.




The presented menu includes the following buttons:

- **Print** - pressing this button starts printing a created printout

- **<<** - pressing this button moves a created printout to the previous page

- **>>** - pressing this button moves a created printout to the next page

- **GoTo ...** - pressing this button opens the ***Go To ...*** dialog box

- **Two pages / One Page** - pressing this button presents a print preview showing 2 pages or 1 page of the printout

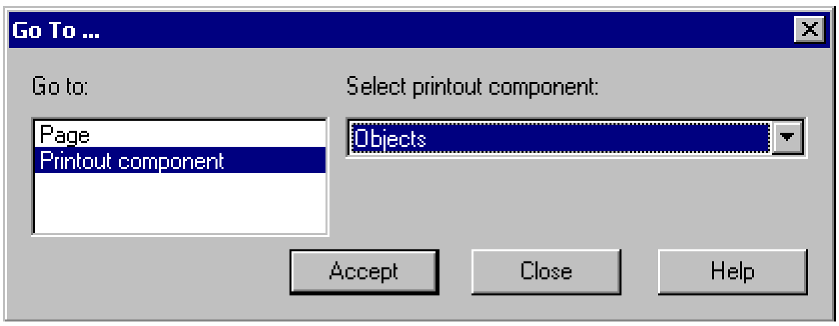
- **Zoom in** - pressing this button zooms in a preview of a created printout

- **Zoom out** - pressing this button zooms out a preview of a created printout

- **Close** - pressing this button closes a printout preview.

## 12.3.Print Preview - Go To

Pressing the **Go To ...** button in the printout preview menu opens the dialog box shown in the drawing below.

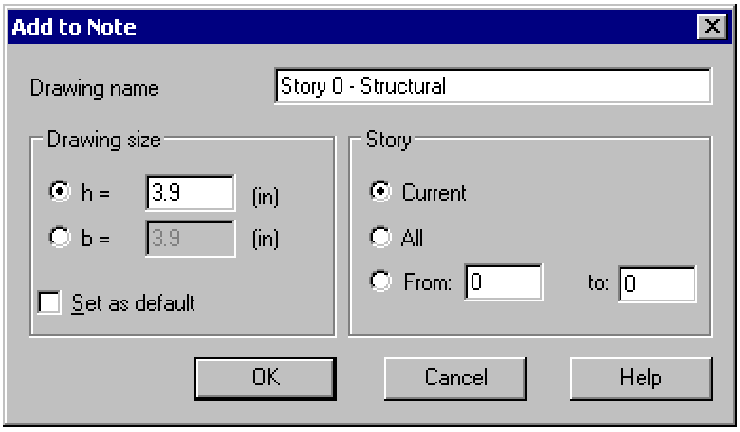


The above dialog box allows selecting the way how to go to a printout element:

- by entering a number of a printout page the user wants to go to
- by specifying a printout component (e.g. structure properties, materials, sections, calculation results, etc.) the user wants to go to.

## 12.4.Add to Note

The option is used to add components to a calculation note created by means of the Printout composition option. The option is available from the context menu (after pressing the right mouse button in any structure view) after choosing the *Add to Note* command.

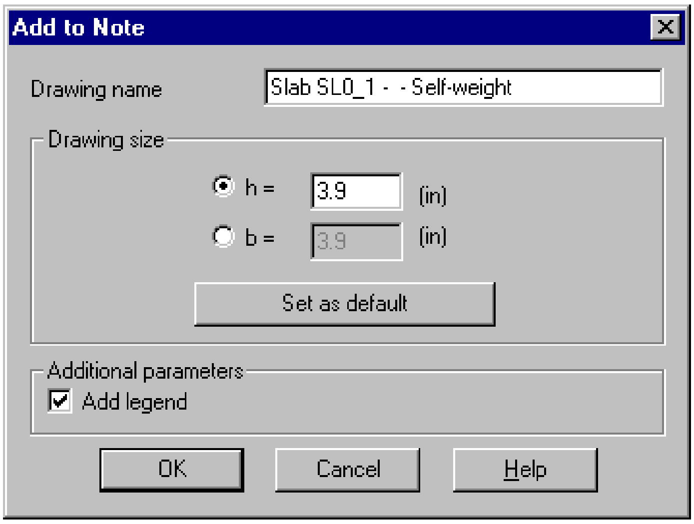



The option allows making a screen capture of the current structure view and adding it to the list of attachments to the note which are available in the ***Printout Composition*** dialog box. The above dialog box makes it possible to:

- assign a name to a generated screen capture (this name will be displayed on the list with the calculation note components) in the *Drawing name* edit field
- set a size of the drawing (screen capture)
  *h* – size definition: determines the drawing height
  *b* – size definition: determines the drawing width
- remember the way of size definition for other projects after pressing the **Set as default** button
- select a building story which may be screen captured; options in the *Story* field are active for a view of one story (2D or 3D view); the following options are available:
  - *current*: a screen capture will be made only for the current story of a building
  - *all:* a screen capture will be made for all stories of a building
  - *from, to:* a screen capture will be made for successive stories (starting with the specified story number no.1 to story no.2).
  If the *All* or *From, to* option is selected, then the variable %p, denoting a number(s) of a story for which a screen capture was made, will be added to the proposed name.

## 12.5.Add to Note - Results

The option is used to add components of calculation and design results for structure elements to a calculation note created by means of the Printout composition option. The option is available after pressing the  icon located on the *Results* tab in the ***Object Properties*** dialog box




The option allows making a screen capture of the current structure view and adding it to the list of attachments to the note which are available in the ***Printout Composition*** dialog box. The above dialog box makes it possible to:

- assign a name to a generated screen capture (this name will be displayed on the list with the calculation note components) in the *Drawing name* edit field

- set a size of the drawing (screen capture)

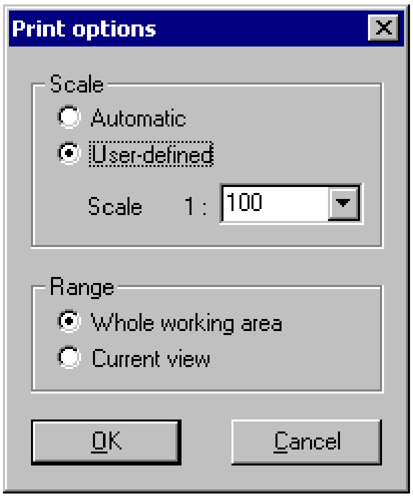
  *h* – size definition: determines the drawing height
  *b* – size definition: determines the drawing width

- remember the way of size definition for other projects after pressing the **Set as default** button.

The lower part of the dialog box holds the option that allows adding a legend (color scale used for map presentation) to the screen capture.

## 12.6.Print Options

The program enables generation of printouts. Printout settings may be defined in the ***Print options*** dialog box which becomes available after selecting the menu option: *File / Print options*.




The following parameters may be defined in the above dialog box:

- *Automatic scale* – after selecting this option, the program adjusts the size of a drawing to the paper format currently set in the printer

- *User-defined scale* – after selecting this option, the user defines a value of the scale (e.g. 1:100)

Range

- *whole working area* - after selecting this option, a view of an entire structure will be printed independently of the current zoom

- *current view* - after selecting this option, a view of the screen part corresponding to the current structure zoom will be printed.

*NOTE:* *If a width of printout space is less than 18 cm (this is a default value for a new project), then before displaying the print preview and before starting printing, the program shows a message which suggest that margins should be reduced so that the printout width is at least 18 cm.*

*NOTE:* *There is a possibility to print from 3D view of a structure.*

# *13. PROBLEMS*

## 13.1.Lack of 3D View with Rendering

3D view with rendering is supported by DirectX library – version 8.1 or higher. Such a library is made available as a separate installation on the installation disk of the ***CBS Pro*** program; its latest version is also provided on the Microsoft Internet pages.
If 3D view with rendering is not available in the program, it may be caused by the following:

- DirectX library has not been installed or has been installed incorrectly – in this case DirectX library should be installed again

- the user works in the system Windows NT – this system does not support DirectX libraries, thus work with 3D view with rendering is impossible

- a graphic card is not set correctly; to improve the 3D view, the user should:
  - reduce the color palette (it should be kept in mind that the 'high color 24 bit' color palette is not DirectX-supported)
  - reduce the size of the 3D window in the ***CBS Pro*** program
  - change the screen resolution from 1024*768 to 800*600

- graphic card drivers are not appropriate (they do not support DirectX library); the user should install the latest card drivers available on the Internet pages of the card manufacturer.