

# Tower 8

An instruction manual for operating with the program  
[Updated for Build 8572]

This guide explains only new features of the program, that is, orders that were not in the Tower 7 program, so it is primarily intended for users who operated in the Tower 7 program.

# Contents

<b>1. INTRODUCTION.....</b>	<b>6</b>
1.2 Program Installation.....	6
1.2.1 Local Program Installation.....	6
<b>2. GRAPHIC INTERFACE.....</b>	<b>8</b>
2.2 Arrangement and Size of Windows for Drawing.....	8
2.6 Drawing the Polygonal Line.....	13
2.6.5 Offset.....	13
2.9 Commands for Manipulating the Elements of Drawings.....	14
2.9.16 Offset.....	14
2.10 Visibility.....	15
2.10.1 Adjusting the Visibility by Types of Elements (Visibility).....	15
2.10.2 Visibility filter by entity sets.....	15
2.10.3 Adjusting the Visibility by Selecting (Hiding).....	18
2.13 3D Graphic Control (menu "3D View").....	24
2.13.1 Rotation of Isometric Presentation (Orbit).....	24
2.14 Commands for Changing the Current Presentation in "2D View" Window (menu "2D View").....	24
2.14.10 Transition to perpendicular view.....	25
<b>3. ENTERING DATA.....</b>	<b>28</b>
3.1 Structure.....	28
3.1.1 Slab/Wall.....	28
3.1.4 Beam.....	28
3.1.10 Area Support.....	32
3.1.11 Linear Support.....	35
3.1.12 Point Support.....	36
3.1.15 Link.....	38
3.1.17 Review of cross section optimisation.....	38
3.2 Loading.....	39
3.2.1 Defining the Load Cases.....	39
3.2.2 Defining the Load Combinations.....	45
3.2.3 Area Load.....	48
3.2.3.1 Area Load on Evolved Surfaces.....	48
3.2.4 Linear Load.....	49
3.2.10 Moving Load.....	49
3.2.10.2 Time history of movable loads.....	50
3.2.15 Turning walls into load.....	53
3.3 Changing and Controlling the Previously Entered Data.....	53
3.3.4 Set Control.....	53

3.4	Commands for Entering the Elements of Drawings other than Parts of the Structure .....	56
3.4.1	Auxiliary Axes.....	56
3.4.1.1	Defining the Axes.....	56
3.4.1.7	Current layer.....	57
3.4.2	Selecting the Layer for Drawing .....	58
3.4.6	Layers visibility .....	59
3.4.7	Item mark.....	60
3.4.7.1	Item marking .....	60
3.5	Adjusting the Parameters Used by the Program in its Operating ("Setup" menu) .....	61
3.5.1	Settings.....	61
3.5.2	Functionality.....	66
3.5.3	Toolbars.....	68
3.6	Commands for Operating with Data Files .....	69
3.6.5	Export of model into Tower 7 .....	69
3.6.7	Importing the Drawings from AutoCAD (DXF Import).....	70
3.6.10	IFC Export.....	70
3.6.11	SAF Import .....	71
3.6.12	SAF Export.....	73
<b>5.</b>	<b>Stability Design .....</b>	<b>75</b>
5.2	Discovery of instability of the model.....	75
5.2.1	The review of instability of the model .....	77
<b>7.</b>	<b>CALCULATING THE MODEL .....</b>	<b>82</b>
7.3	Calculation Methods .....	82
7.3.4	Seismic Analysis .....	82
7.3.4.1	EUROCODE .....	82
7.3.4.2	EUROCODE (Lateral forces method) .....	84
7.3.4.4	Swiss Regulations (SIA).....	86
7.3.4.6	Direct dynamic analysis .....	86
7.3.4.7	Italian Regulations (NTC 2018).....	88
<b>8.</b>	<b>POST-PROCESSING.....</b>	<b>93</b>
8.1	Purpose and Description of Module Operation .....	93
8.5	Influences in Slab/Wall .....	98
8.5.2	Influences in Arbitrary Cross-section (Section).....	98
8.5.6	Review of influences in slab sections.....	99
8.6	Influences in Beams .....	101
8.6.1	Individual Diagrams .....	101
8.6.3	Diagrams in Cross-section.....	102
8.10	Resultant of the reactions.....	104
8.12	Proxy entity .....	106
8.12.2	Influences in the proxy entity .....	106

8.13	Influences in soil halfspace .....	107
8.17	Bill of quantities.....	107
8.18	Structural Design Results Textual Representation .....	109
8.19	Generating the report for entity groups .....	111
8.20	Removing the Presented Influences from the Screen Instantly (Reset).....	112
8.21	Numbering the Finite Element Mesh Nodes.....	113
8.22	Entity search.....	113
8.23	Set Control .....	118
8.24	Creating the Design Documentation .....	120
8.24.1	Export of Graphic Blocks .....	120
8.24.3	Defining the Paper Format (Page format) .....	122
8.24.6	Commands for Operating with Blocks in Tree .....	128
8.24.18	Exporting the block to the clipboard .....	130
8.24.23	Automated update of the report.....	130
8.25	Direct Dynamic Analysis (available only in the Expert Configuration of the Program) .....	137
8.25.2	Time history of selected entities.....	137
8.26	Columns buckling length calculation .....	140
8.26.1	Viewing of columns buckling coefficients .....	142
8.26.2	Columns Grouping.....	144
8.27	Show area entities extremes.....	145
8.28	Show diagram values in 3D .....	146
<b>9.</b>	<b>Concrete Design .....</b>	<b>147</b>
9.1	Selecting the Codes for Design (Codes) .....	147
9.2	Defining the Load Combining Pattern .....	148
9.3	Slab Design .....	153
9.3.1	Input Data .....	153
9.3.1.1	Input data - Global.....	153
9.3.3	Presenting the Design Results.....	154
9.3.6	Manipulating the Reinforcement Regions .....	156
9.3.7	Creating the Textual Report .....	159
9.3.11	Slab punching verification .....	159
9.3.12	Review of slab punching verification .....	163
9.4	Beam Design.....	165
9.4.1	Input Data .....	165
9.4.1.1	Input data - Global.....	165
9.4.3	Presenting the Design Results.....	166
9.4.4	Adopting the Reinforcement .....	167
9.4.5	Generating the textual report .....	174
9.4.10	Interaction diagram of the beam.....	175
9.4.11	Designing the concrete according to EC8 (Capacity design) .....	178
9.4.11.2	Method of implementing dimensioning of concrete beams and columns according to EC8 EN.....	178
9.4.11.5	(EC8 CD) Critical Regions.....	179

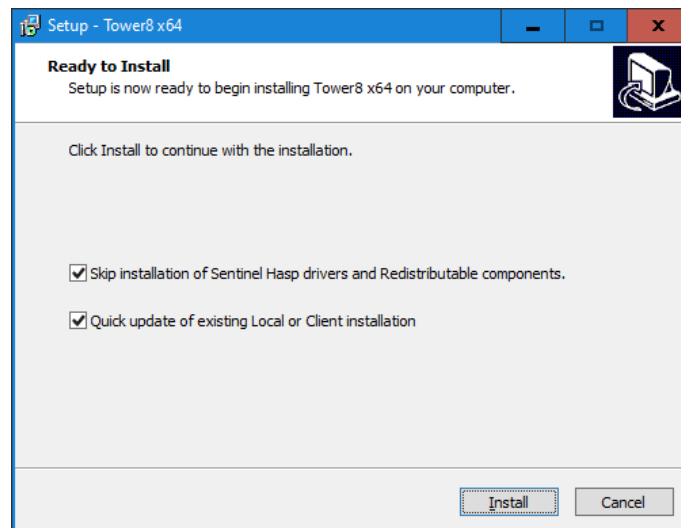
9.5	Walls Design (Shear walls) .....	180
9.6	Dimensioning a series of walls .....	181
9.7	Review of walls designing .....	182
9.8	Cross-section Calculation .....	188
9.9	Masonry wall stress control.....	189
9.11	Design of proxy entity .....	192
9.12	Review of proxy entities.....	210
9.13	Quantities of reinforcement .....	211
<b>10.</b>	<b>Steel Section Design .....</b>	<b>214</b>
10.3	Stress control.....	214
10.4	Input Data .....	217
10.6	Viewing results .....	218
10.9	Stability control - calculator .....	221
10.10	Optimisation of cross sections .....	221
10.11	Review of cross section optimisation .....	226
<b>11.</b>	<b>Wood Constructions Dimensioning process .....</b>	<b>229</b>
11.8	Stability control - calculator .....	229

# 1. INTRODUCTION

## 1.2 Program Installation

### 1.2.1 Local Program Installation

For users installing the new Build version, the ability to quickly install the Tower program via an existing local or client installation has been introduced. This installation method does not require you to enter an installation number or select a directory. The program also allows you to skip installing Sentinel Hasp drivers and Visual Studio components. Running the file "**\Tower8\Tower8\_win32\_Setup.exe**" if you install the 32-bit version of the program, or the file "**\Tower8\Tower8\_x64\_Setup.exe**" if it is a 64-bit version of the program will open the following dialogue:



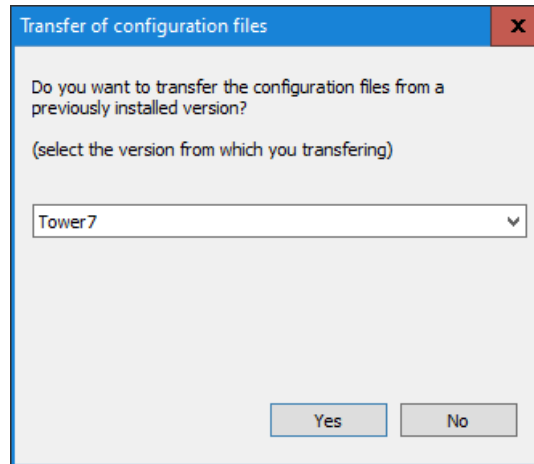
#### **Skip installation of Sentinel Hasp drivers and Redistributable components.**

By setting the check box to on, the installation allows the program to skip installing Sentinel Hasp drivers and Visual Studio components.

#### **Quick update of existing Local or Client Installation**

Setting the check box to on will enable quick installation of the program, via an existing local or client installation, without entering the installation number and selecting a directory, while maintaining all existing parameters and settings and without changing the license (approved modules and number of jobs). If there is a need to change the content of the license (you have purchased new modules or new jobs and received a new installation number), you need to uncheck this check box to have the opportunity to enter a new installation number.

If the program is installing for the first time, and some of the earlier versions of Tower are installed on the computer, the configuration files created in it can be transferred to Tower 8. In this case, after selecting the installation directory, a new dialog will appear as follows:



In the closed list will be proposed all versions of the Tower installed on the computer. Supported versions: Tower6, Tower6 x64, Tower7, Tower7 x64, Tower8, Tower8 x64. If you wish to keep the configuration files, you need to select the desired version of the program from the closed list and press the "Yes" button. After that, all the configuration files created in the selected program will be copied to the corresponding Tower 8 folder.

Selecting the "No" button will indicate that the configuration files are not downloaded, but are created completely new, with the default configurations.

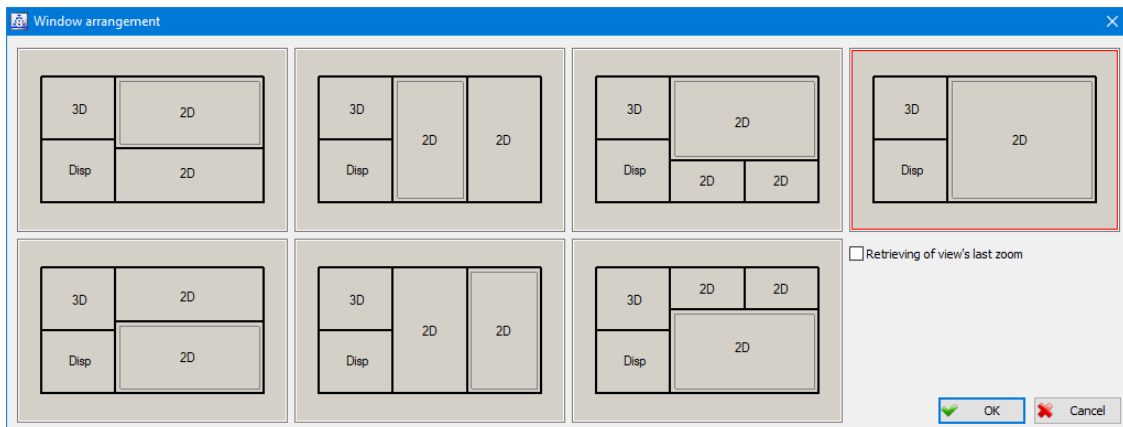
Configuration files that appear for the first time in Tower8:

<b>Tower.\$arm</b> -	Configuration data for beam reinforcement adopting
<b>Tower.\$kke</b> -	Configuration for parameters of graphical blocks
<b>Tower.\$omp</b> -	Configuration data for favorite cross sections
<b>Tower.\$ppp</b> -	Configurations for steel design cross sections optimisation
<b>Tower.\$tsc</b> -	Configuration data of reinforcement concrete section calculator ("Cross-section calculation" command)
<b>Tower.\$tst</b> -	Configuration data for single member stability control (timber)
<b>Tower.\$tss</b> -	Configuration data for single member stability control (steel)
<b>Tower.\$tpp</b> -	Configuration data for slab punching control calculator
<b>Tower.\$svod</b> -	File with recorded configuration of proxy entity arm mode
<b>Tower.\$dopt</b> -	Parameters of dynamic load

## 2. GRAPHIC INTERFACE

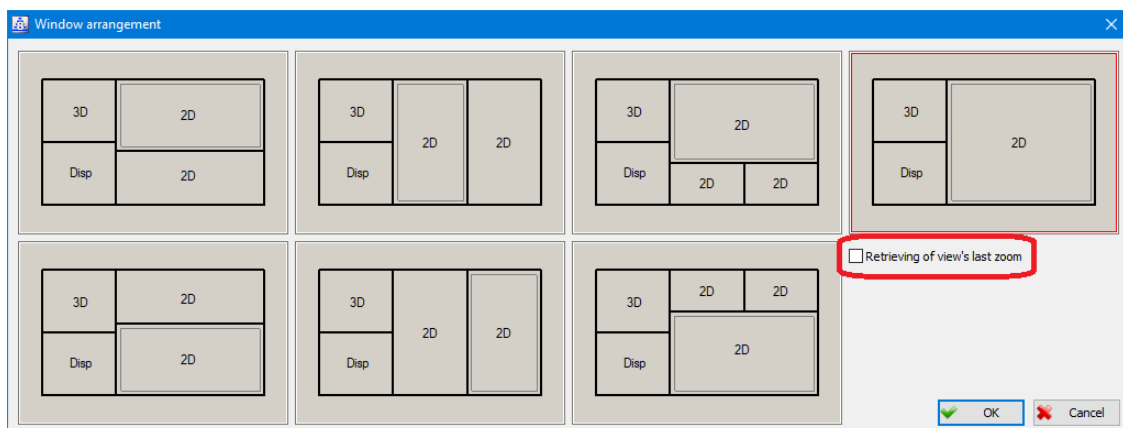
### 2.2 Arrangement and Size of Windows for Drawing

Tower 8 program can have more than one 2D windows. Various assemblies can be displayed at the same time: Frames, levels, auxiliary views. This feature of the program significantly makes easy the selection of points in drawing model, as well as reviewing and comparing the results of different parts of the model. The selection of the number and the inter-arrangement of the 2D window is done from the dialog **"Window arrangement"**:



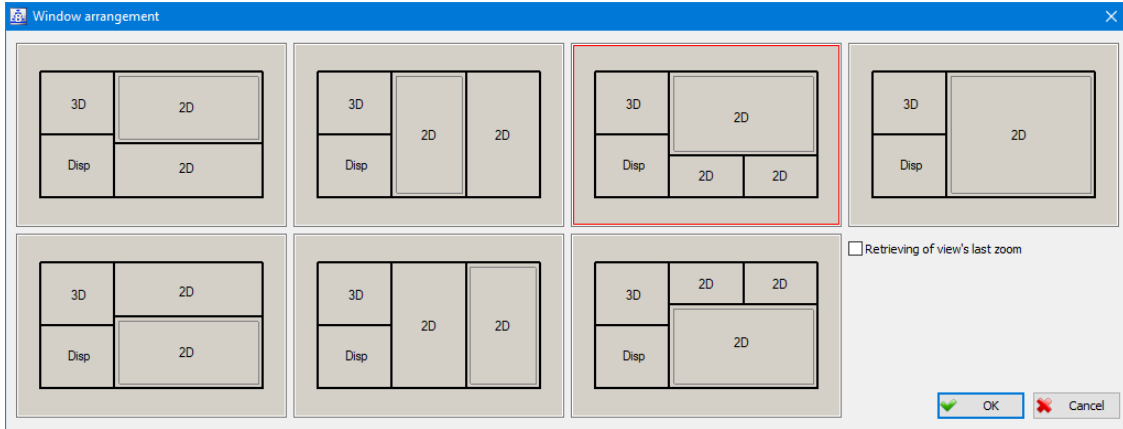
Appearance of the dialog is used to select the number and interchange of the 2D windows

**Retrieving of view's last zoom** - It is possible to return to the previous zoom state when returning to the view that was previously set in the 2D view. If the check box is set to the on state, the view will have the previous zoom state when changing the view. If the check box is set to off, the program will set the default view zoom.

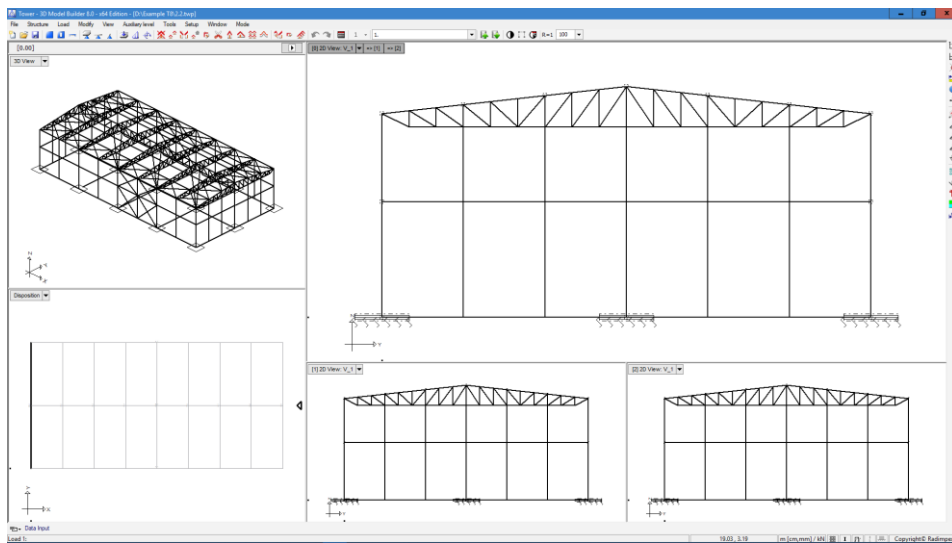


Check box that allows you to return to the output zoom view when changing assemblies

The one 2D window is always the main one and it behaves in exactly the same way as the program behaves when only one 2D window is selected. On the offered pictures it is specially framed. Other 2D windows are called auxiliary. Choosing the desired number of windows is done by clicking the mouse over the corresponding image in the dialog, after that it becomes framed in red.

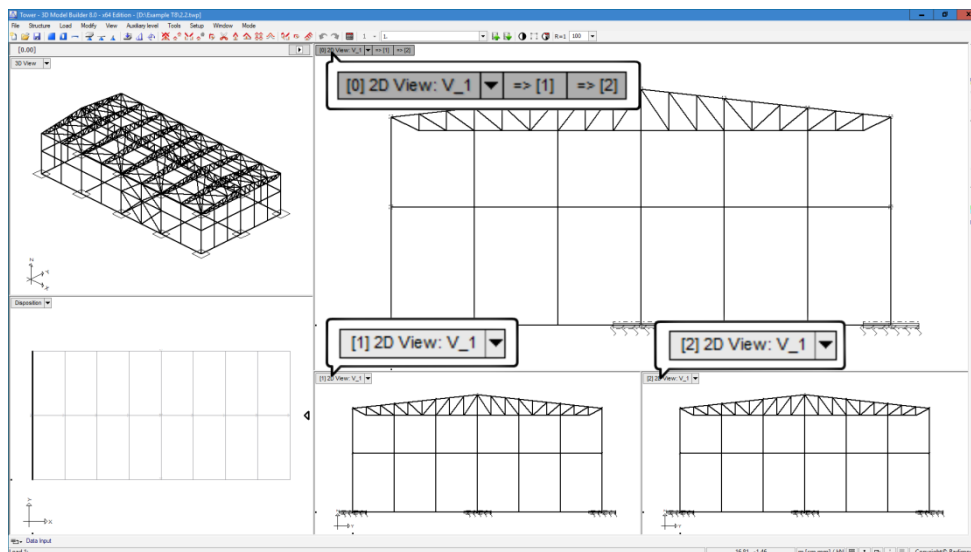


It has chosen three 2D windows. The big one which is also the main, while two auxiliaries are below it.



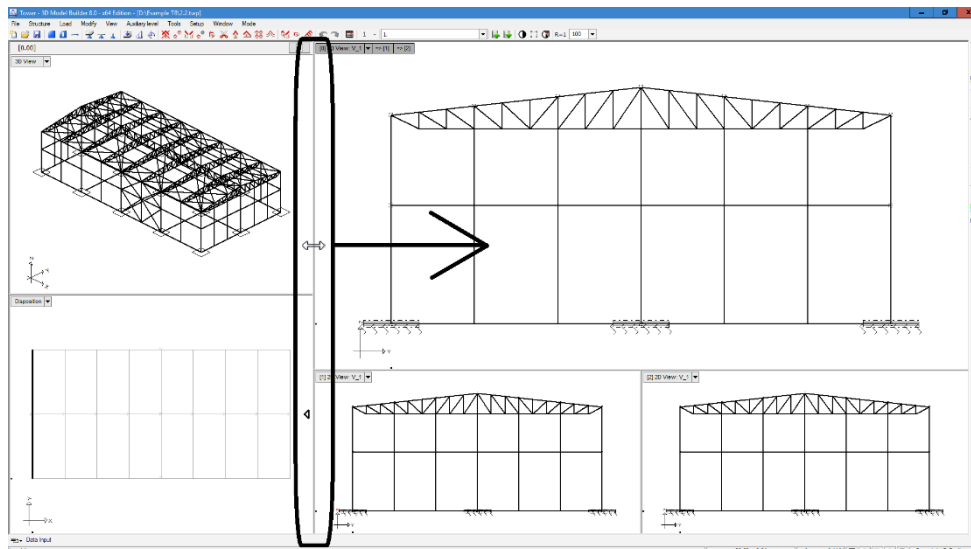
Layout the chosen disposition of 2D windows

In order to know which 2D window is the main one, and which is an auxiliary one, in front of the name of the main 2D window zero is written in brackets, and in front of the name of each auxiliary view is written its ordinal number.

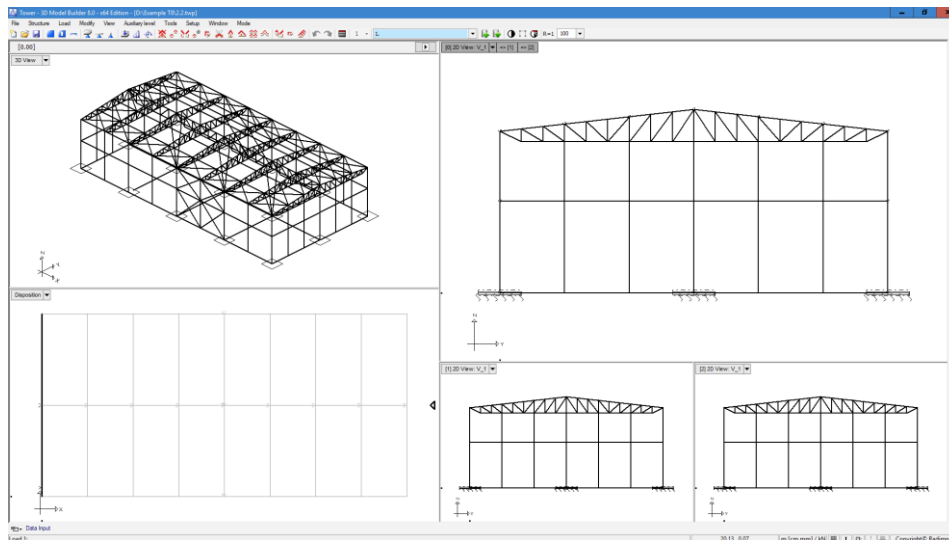


Marking the main and auxiliaries 2D layout

The size of the selected windows can be changed by moving their borders with the mouse. It is permissible to move the border that separates the 2D window from the 3D window and Disposition, as well as the boundaries between the main and auxiliary 2D windows. In order to change the size you need to move the mouse pointer over the desired boundaries by changing its shape in the double arrow, then press the left mouse button and move the border to the new position.



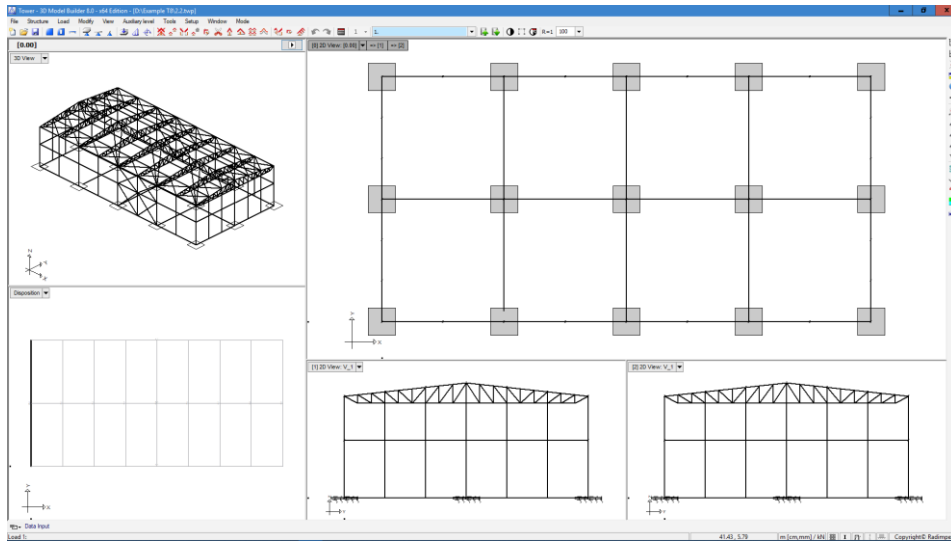
Moving the border that separates 2D windows from 3D windows and Disposition



Window size is changed by moving the border

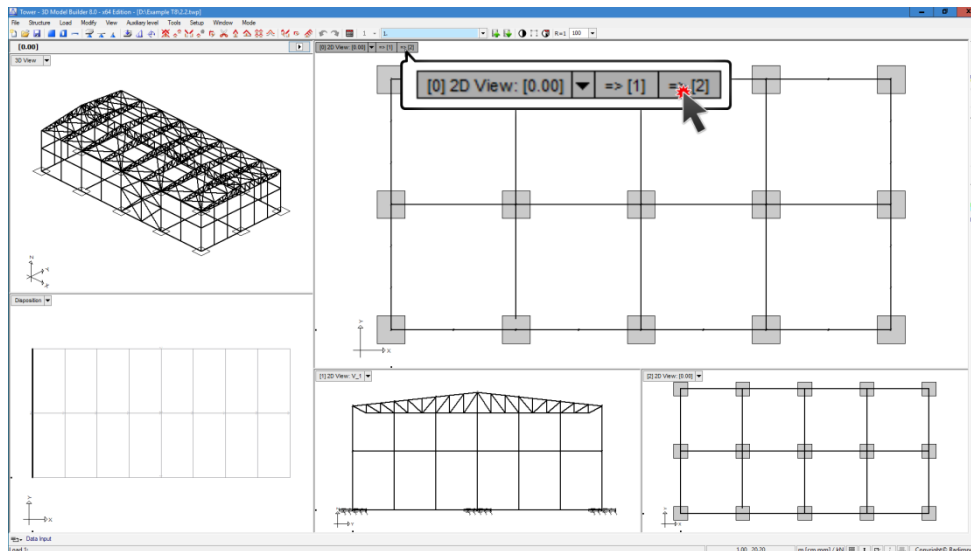
Restoring the borders between the windows to the default position can be done by choosing the **“Non widescreen monitors”** command from the **“Window”** drop-down menu.

When you operate with more than 2D windows, the change of the current frame, level, or auxiliary view is only displayed in the main 2D window, while the display in the auxiliary windows is determined using the commands provided for it. To display a layout in auxiliary 2D window, it is necessary to select it to be displayed in the main 2D window in one of the methods described above.



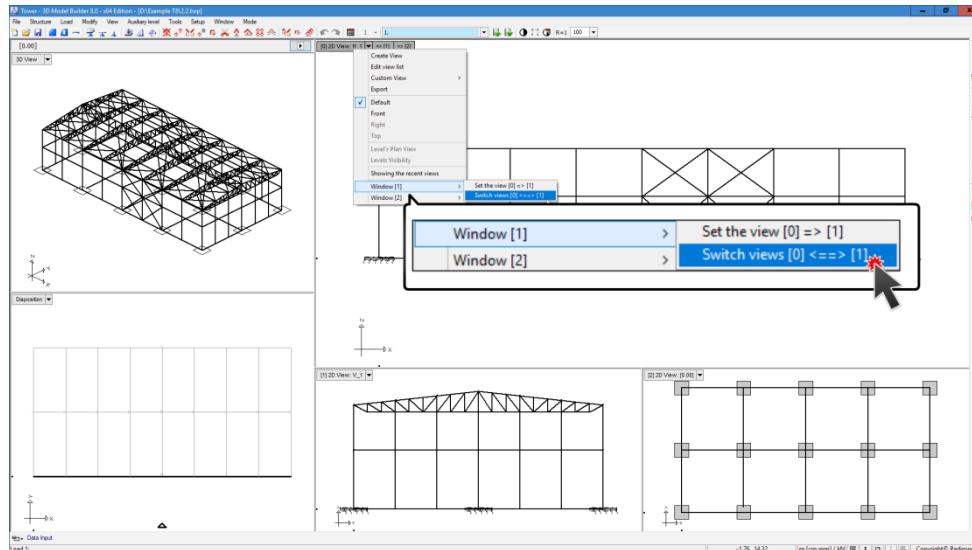
By selecting from the list of levels in the main 2D window is displayed the content of level 0.00

Behind the name of the main 2D window, there are as many control boxes as the number of auxiliary windows, and on each of them is printed the ordinal number of the corresponding auxiliary window. By clicking the mouse over one of these boxes, it is intended to display a current layout from the main 2D window in the auxiliary window which ordinal number is printed on it.

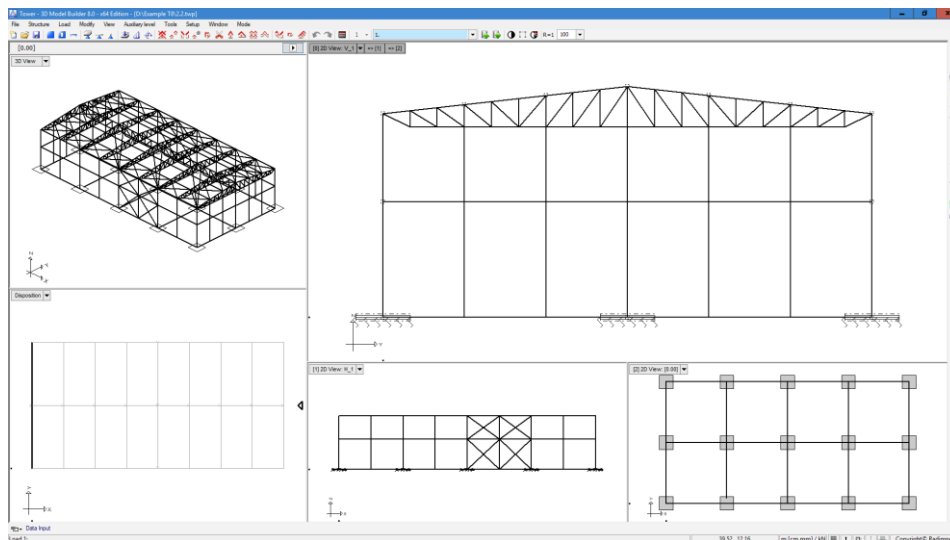


By clicking the mouse over the command box with the ordinal number 2 the current layout from the main 2D window is displayed in the corresponding auxiliary window

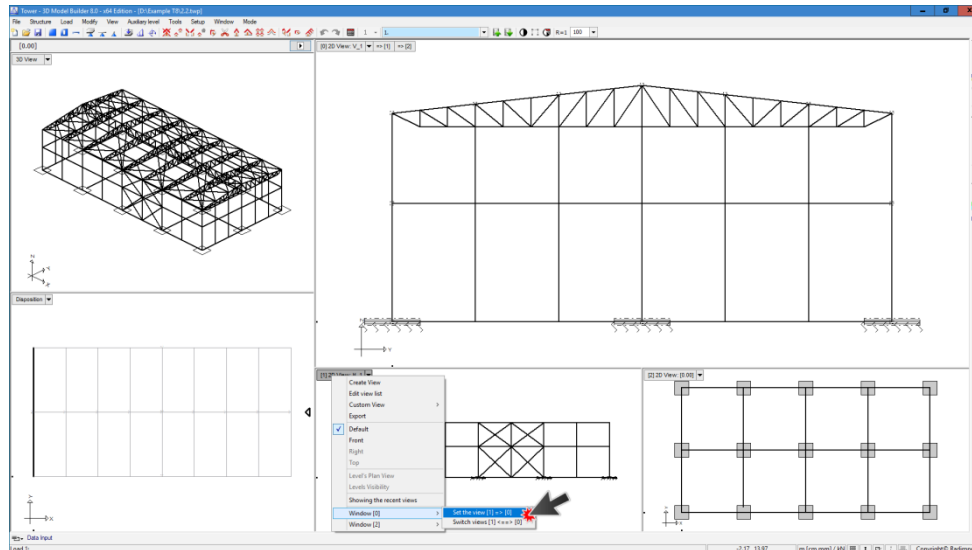
Another way to get a view of auxiliary windows is using commands from the drop-down menu, which opens with the right mouse click over the name of the main 2D window. In this menu are the names of all the auxiliary windows, and by placing a mouse pointer above one of them, a new menu with two commands is opened: **"Set the view"** and **"Switch views"**. Using the Set up layout command, you assign the current layout from the main 2D view to the selected auxiliary view, and the "Switch views" command replaces the layout of the main and the selected auxiliary 2D view. Behind the commands are printed the ordinal numbers of the main and the auxiliary view, and you can clearly see what will happen with the choice of that command.



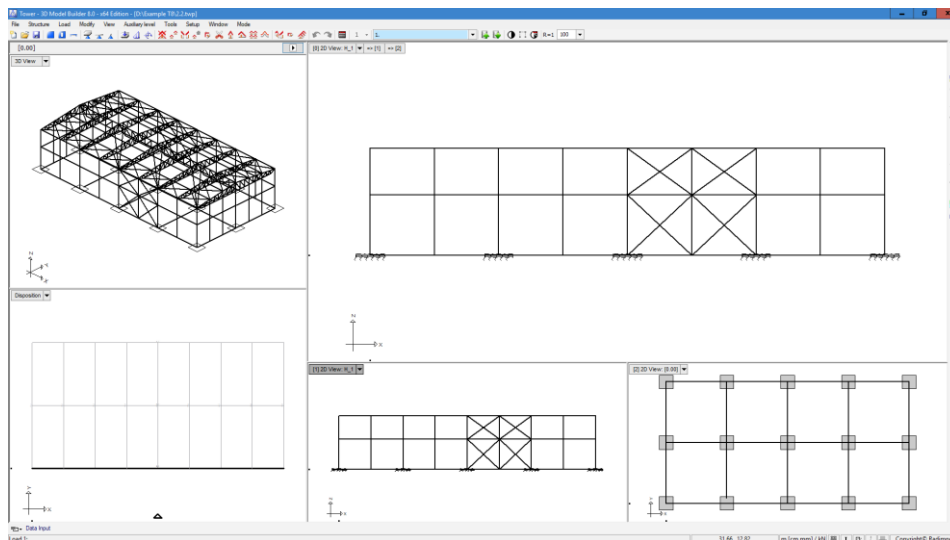
Selecting the command "Switch views [0] <=> [1]", replacement is done between the main layout [0] and auxiliary [1]



These commands are also in the drop-down menus that can be opened by right-click of the mouse over the names of the auxiliary windows. It practically means that in the same way, the current layout from the selected auxiliary window can be assigned to the main 2D window or some other window, or to switch their layouts.



Selecting the command "Set the view [1] <==> [0]", the layout of the auxiliary window [1] is assigned to the main layout [0]



## 2.6 Drawing the Polygonal Line

### 2.6.5 Offset

Distance <1>:

The program remembers the last given distance and shows it in parentheses. Pressing the Enter key on the keyboard or right-clicking will indicate the adoption of this value. The distance can be set as well as by entering values from the keyboard or selecting two points from the drawing.

## 2.9 Commands for Manipulating the Elements of Drawings

### 2.9.16 Offset



With this command, the geometry of the existing linear and surface entities can be changed by drawing lines parallel to the existing at the given distance and on the given side, then, the final operation will be deleted. The influence is the same at linear entities as it has been moved in the direction of the perpendicular entity to the selected entity. In the case of superficial entities, the contour determines them, which means that the length of lines that make up the contour are also changing. So, all contour lines are drawn again from the inner or outer side, at a set distance, and their new length is obtained by section with the adjacent lines of the new contour.

After activating the command, the program from the command line first requires selecting an entity that will be offset:

**<0 sel.> Offset - Select (Window / <End>):**

When an entity is selected it is necessary to specify the distance to which the entity will be offset:

**Distance <1>:**

The program allows you to remember and quickly select the last used distance value. Clicking on the right mouse button will indicate acceptance of the offered distance value "1", which is in the square bracket.

Distance can be assigned in two ways: by entering the values from the keyboard and selecting two points from the drawings, so that their shortest distance between them determines the offset distance. After setting the first one, the program from the command line will require to enter the second point:

**Second Point:**

Finally, the point that determines the side to which the offset is made is entered:

**Point on side to offset (Delete source entity / <End>):**

Selecting the "End" option while the "Delete source Entity" option is displayed on the command line, deletes the entities selected for offsetting. Left click on the "Delete source entity" command line to get a new look:

**Point on side to offset (Keep source entity / <End>):**

Selecting the "End" option while the "Keep source entity" option is displayed on the command line allows selected entities not to be deleted from the drawing after the offset is completed.

## 2.10 Visibility

### 2.10.1 Adjusting the Visibility by Types of Elements (Visibility)

#### Beam

- ✓ Description - The display of the beam description is now regulated by the check box, instead of the switch from the previous versions. This means that both the set and the description of the beam can be displayed at the same time.

#### Auxiliary level

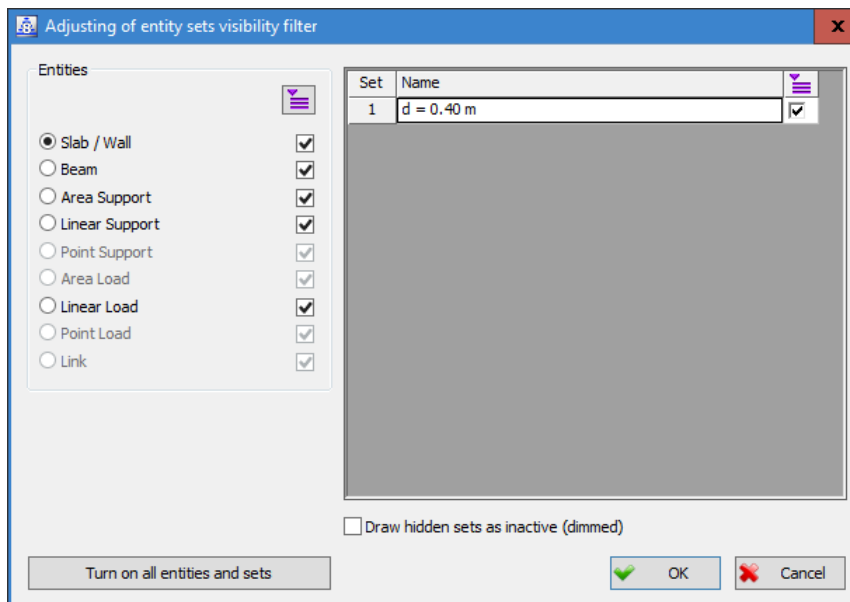
- ✓ Layers - In this part of dialog it's possible to change the visibility and locking/unlocking the layers both individually and simultaneously for all layers, in exactly the same way as in the command "Layers visibility", which is explained in detail in Chapter 3.4.6.

### 2.10.2 Visibility filter by entity sets



The visibility of the drawing elements can also be determined by the data assigns that are associated with them in the current model. This type of visibility is switched on by choosing the "**Visibility filter by entity sets**" command from the "View" drop-down menu or by the left-click of the mouse above the icon displayed. When the command is active, this icon is embedded, that is, framed by a blue square (the mode of displaying active icons depends on the operating system used).

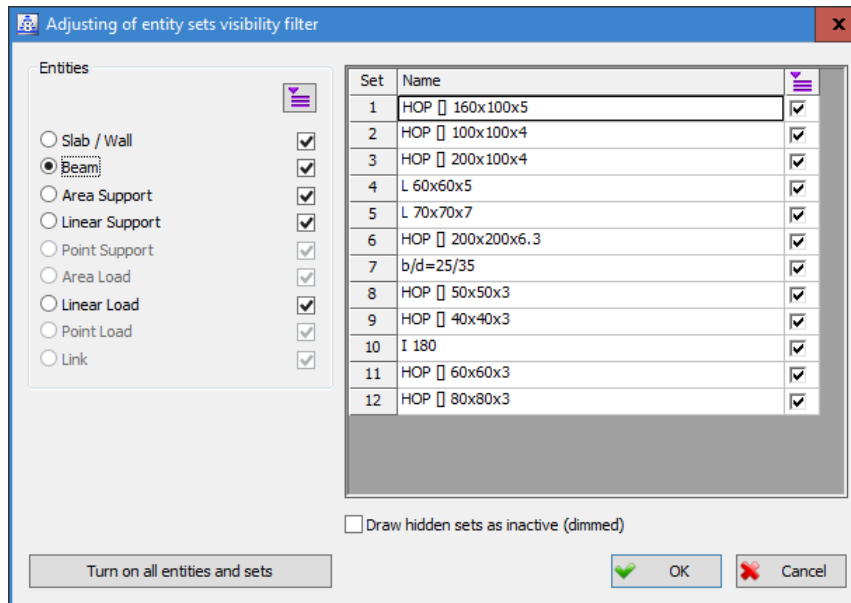
The selection of sets that will be visible or invisible will be done in the dialog "**Adjusting of entity sets visibility filter**", which starts from the "View" drop-down menu or right-click the icon above.



Dialog for defining visibility by sets

In the left part of the dialog, the names of all entities whose visibility can be changed using this command are displayed. We will note that only those entities that exist in the current model are available. The choice of the entity is done by the left mouse click over the name, after which the

switch that is in front of it is switched on, and in the right part of the dialog all data assigns of the selected entity are displayed.



In the right part of the dialog, all data sets are displayed which are joined to beams in the current model

To the right of the name of each entity, there is a check box that can change its global visibility. For example, if this check box is turned off for the beam entity, no beam will appear on the screen, whether it belongs to a set that is visible or not. Changing the state of this switch also it changes the state of the switch that adjusts the visibility of the entity in the "Visibility" dialog, and vice versa.



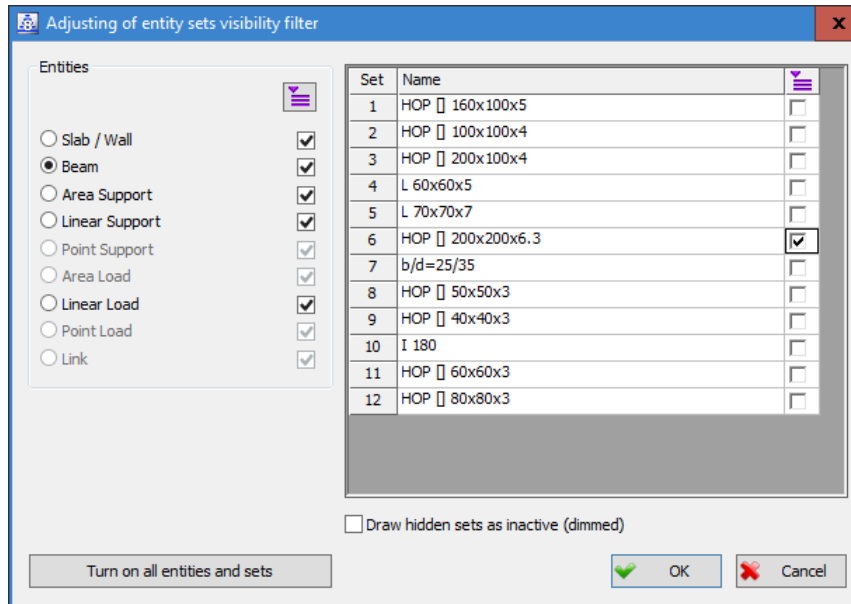
- Clicking the mouse above this button opens the menu with the "All on" and "All off" options, which can enable/disable all check boxes, and hence the global visibility of all entities in the model.

In the right part of the dialog, all data sets of chosen entity are displayed The table contains three columns:

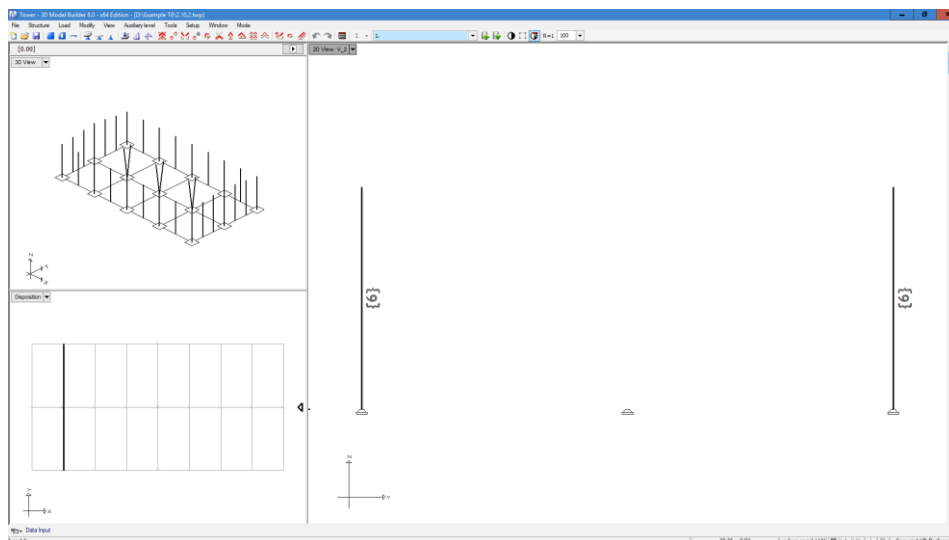
**Set** column is for marking the ordinal number of the set.

**Name** column is for marking the description of the set. For example, at the slabs, this is its thickness, at the beam dimensions are the dimensions of the section, and at the load is displayed the intensity.


In the third column there are check boxes that determine the visibility of the sets. So, if the check box is set to on, the display will display the entities that belong to that set, and if it is turned off, it will not be displayed.



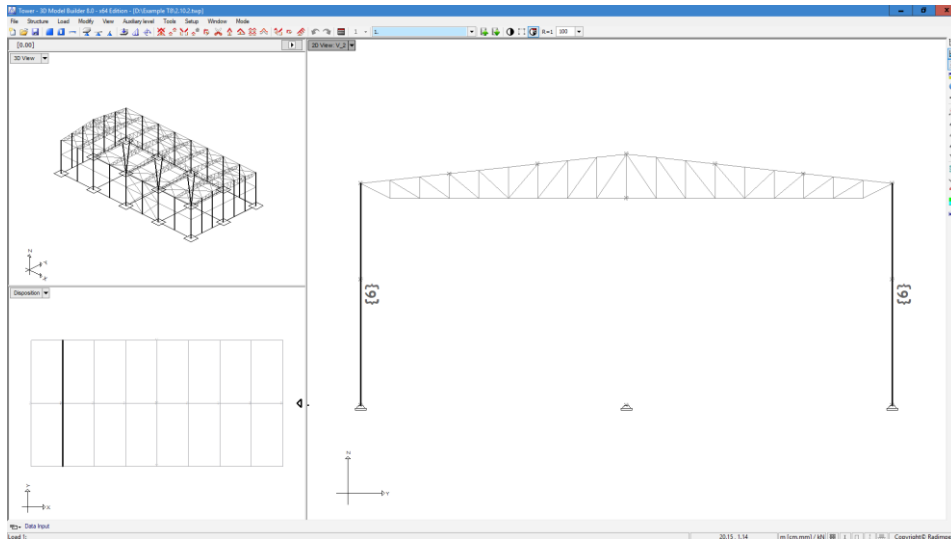
The visibility of all sets of beams except for set 6 is turned off



Only the columns belongs to set 6 are displayed on the screen

 - Clicking the mouse above this button in the header of the third column of the table, the "All on", "All off", and "Invert selection" options menu opens. With the first two options, all the check boxes in this column can be turned on/off, and the check boxes are turned off/on using the "Invert selection" option.

**Draw hidden sets as inactive (dimmed)** - When this check box is set to on, the entities belonging to the sets whose visibility is turned off are plotted with the parameters for plotting inactive entities that are set in the "Settings" dialog box (3D View => Wireframe => Set Control => Non-active Elements).



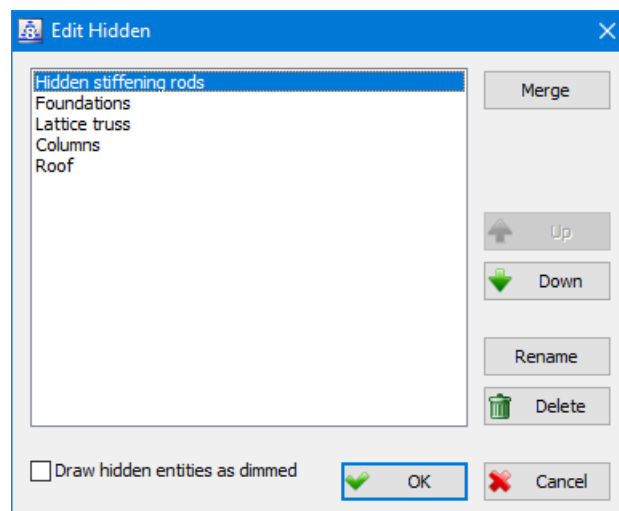
Beams belongs to turned off sets (all sets except set 6),  
are drawn on the display as inactive

**Turn on all entities and sets** - By clicking this button, all the visibility settings in this dialog are canceled by setting all check boxes that determine the global visibility of the entities as well as check boxes to determine the visibility of the sets of each of these entities.

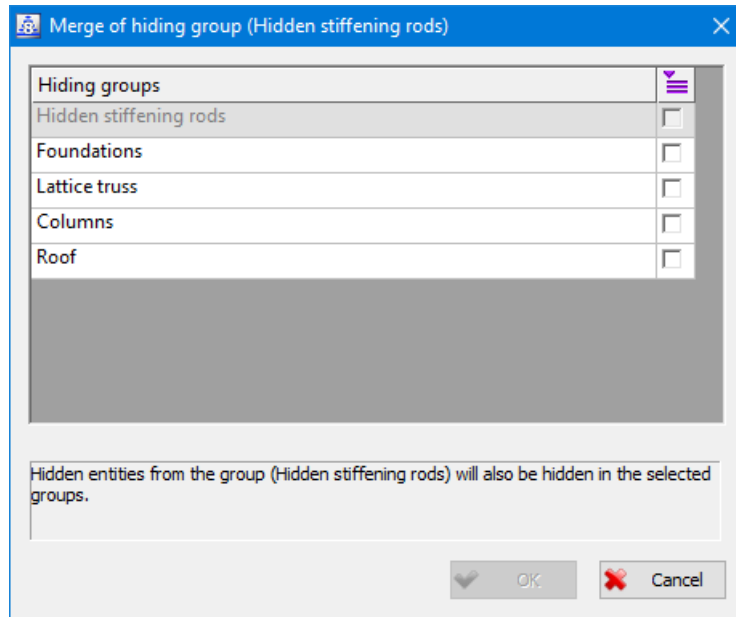
If the model is recorded after setting visibility across sets, all settings will be memorized and loaded with this model. It means that all models can have their own visibility settings by sets.

The "Visibility filter by entity sets" command is transparent, which means it can also start when another command is active.

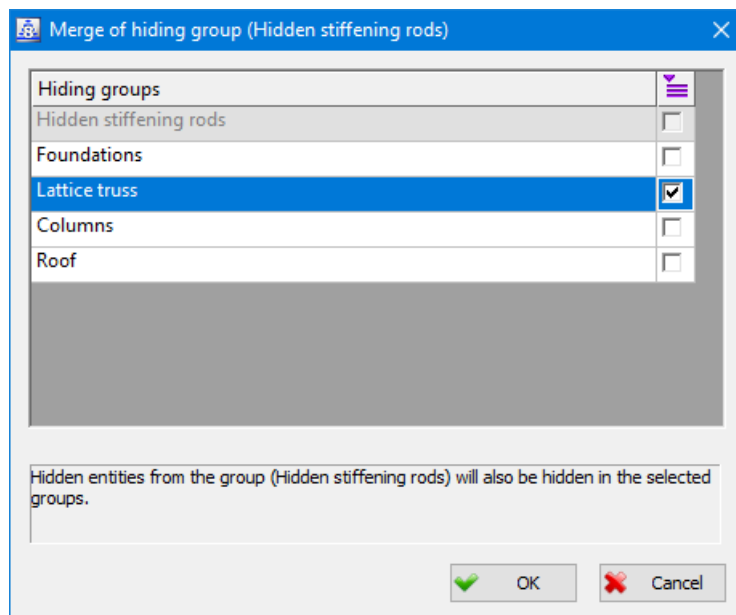
### 2.10.3 Adjusting the Visibility by Selecting (Hiding)



There is often a need to hide some entities that are not currently hidden in already defined groups. For this purpose, the "**Merge**" button is provided, with the help of which the union of hidden elements of one hiding group with other groups can be performed. Selecting it opens a dialog of the following appearance:




The name of the dialogue also displays the name of the group that was selected when the command was run. This group is specially marked in the table. In addition, the table shows all other hiding groups that are defined for the current model. In this dialogue, groups are selected, in order to unite their hidden elements with hidden elements from a specially marked group. The selection of groups is done by including the check boxes located in the column to the right of their names.



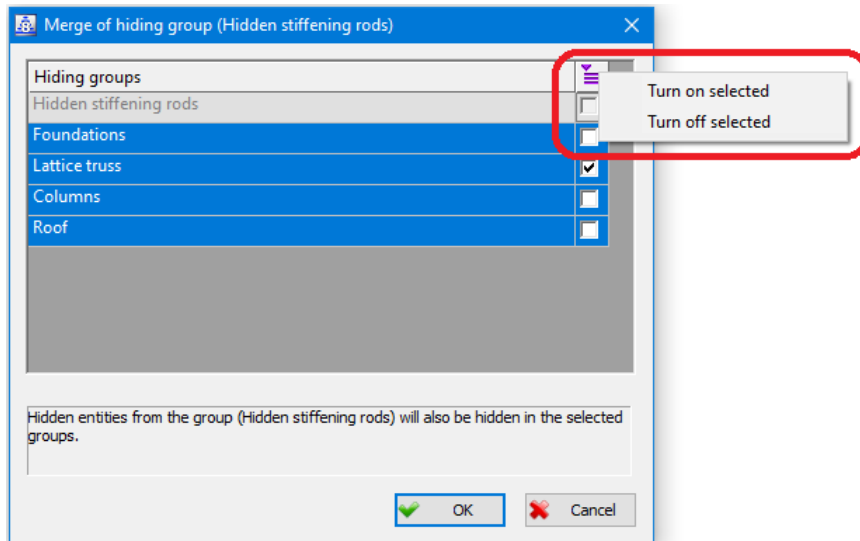
By turning on the check box, to merge to the group "Hidden stiffening rods" the group "Lattice truss" is selected

Multiple selection of groups using the "Shift" and "Ctrl" keys is also enabled in the dialog. Pressing the "Ctrl" key and clicking the mouse on one of the groups will select that group, while the previously selected groups will remain selected. Clicking the mouse on a previously selected group will exclude it from the selection. When the "Shift" key is pressed, a mouse click on a group selects not only that group, but also all other groups in the table that are located between the previously selected and that group.

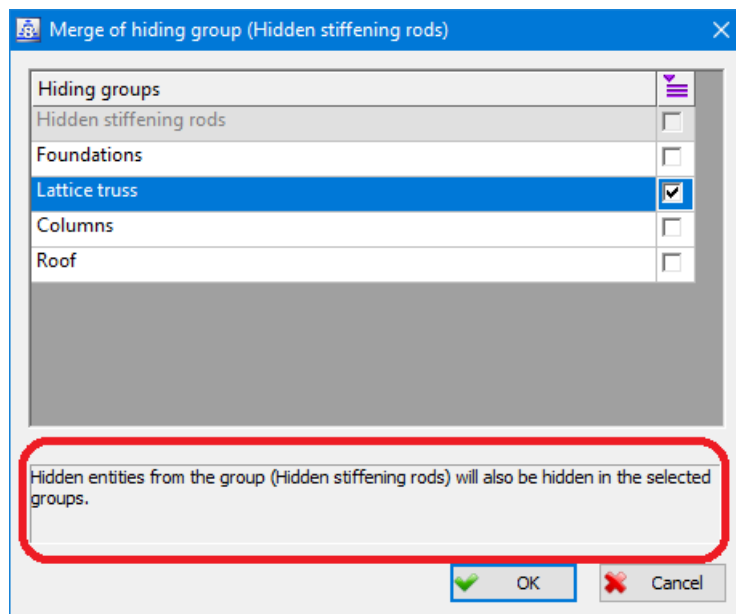
Right-clicking on the “” icon, located at the top of the check boxes column, will open a drop-down list with the following options:

**Turn on selected** - selecting this option will activate check boxes for all selected groups from the table.

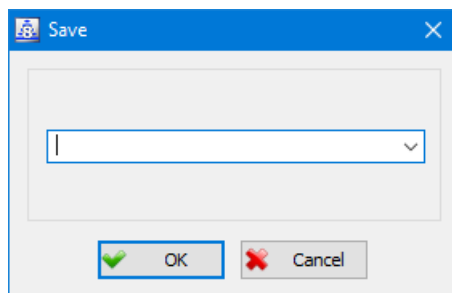
**Turn off selected** - selecting this option will turn off the check boxes for all selected groups in the table, if they were previously turned on.



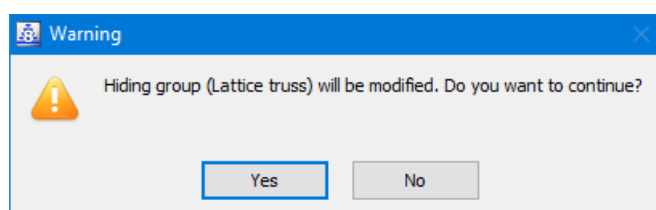
The lower part of the dialog also shows a brief explanation of the result of working with this command.



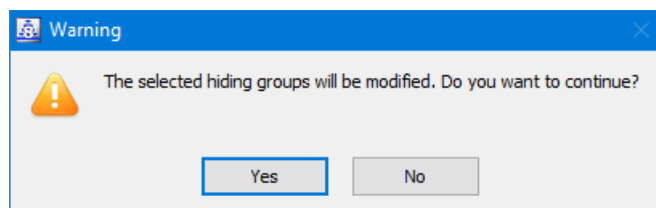
If only one hiding group is selected, after activating the “OK” button, a dialogue will open in which the name of the new group is obtained, which is obtained by merging it with the group that is specially marked.



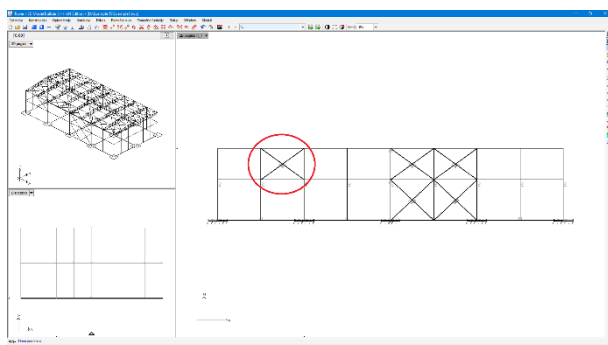
By selecting from the list that opens by clicking on the arrow at the right end of the name entry field or by entering a name from the keyboard, you can also specify the name of an existing group. In this way, the existing group will be replaced with a new one, and the program will display the corresponding message.



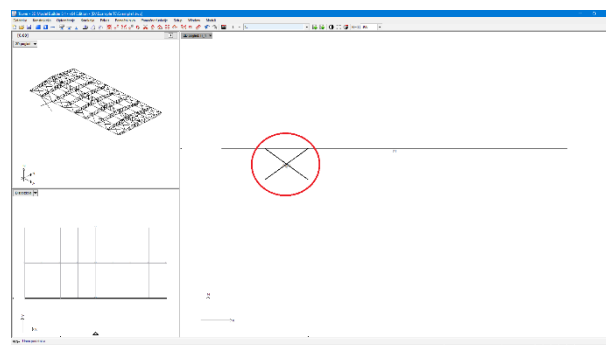
If more than one hiding group is selected, each of them will be modified, so that the hidden entities from the specially marked group will become hidden in each of the selected groups as well. When exiting the dialog, the program will display the appropriate message.



The most common example of using this command is when new elements are added to an existing model. Let's suppose that new stiffening rods have been added to the model of the hall, which should not be seen in some previously defined hiding groups, for example in those in which only elements belonging to the roof or mezzanine structures are visible ...

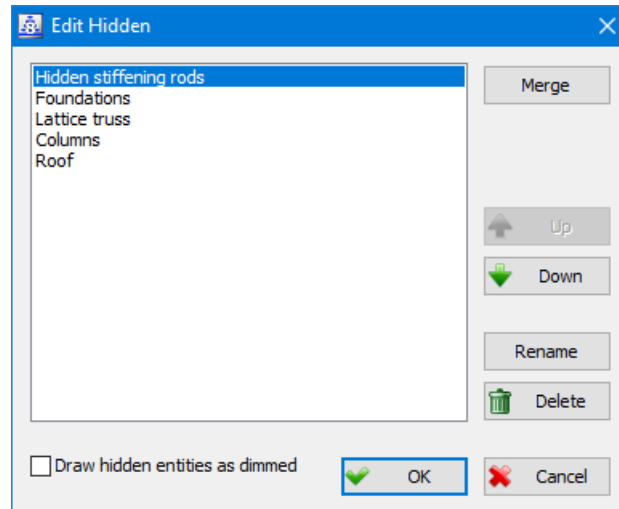


Stiffening rods drawn

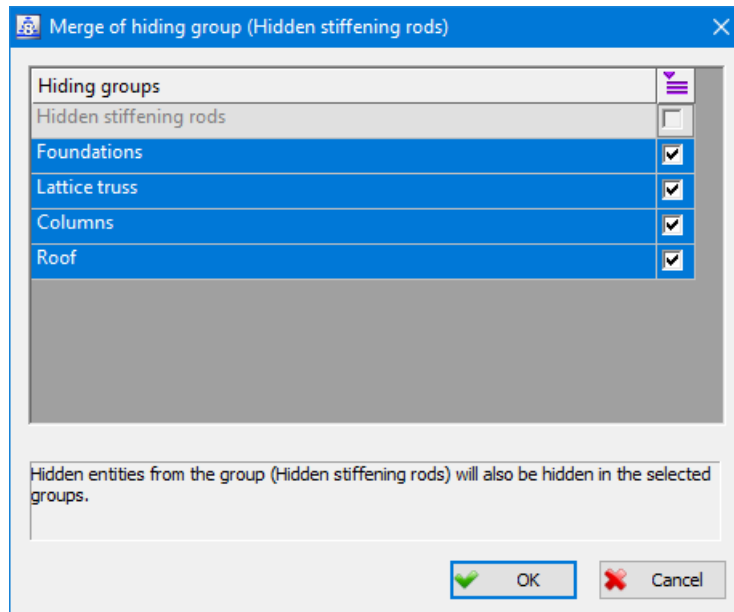


It can also be seen in the group "Roof", and it should not be

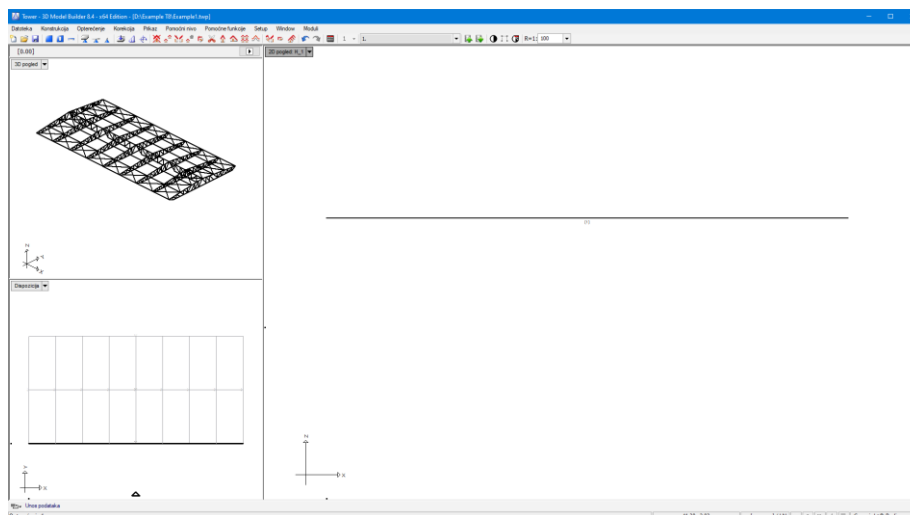
First you need to create a group in which these rods are invisible, and then run the command "Edit hide list".



In this dialog you should select the just created hiding group "Hidden stiffening rods" and click on the "**Merge**" button. In the new dialog, all hide groups will be selected to merge.



Upon completion of the command, the newly added rods will no longer be seen in the selected hide groups.



The drawn stiffening rods are no longer visible in the "Roof" group.

## 2.10.4 Beams visibility



By selecting the command "**Beams visibility**", which is located in the drop-down menu "Layout" or by left-clicking the mouse above the displayed icon, the visibility of the beams is changed. When the command is active, this icon is pushed-in, i.e. framed by a blue square (the way active icons are displayed depends on the operating system used) and then beams will be displayed on the screen. The term "beam" refers to all girders which inclination with the horizontal plane is less than  $45^\circ$ .

## 2.10.5 Columns visibility



By selecting the command "**Column visibility**", which is located in the drop-down menu "Layout" or by left-clicking the mouse above the displayed icon, the visibility of the columns is changed. When the command is active, this icon is pushed-in, i.e. framed by a blue square (the way active icons are displayed depends on the operating system used) and then columns will be displayed on the screen. The term "column" refers to all girders which inclination with the horizontal plane is greater than  $45^\circ$ .

## 2.10.6 Slabs visibility



By selecting the "Slabs visibility" command, located in the drop-down menu "Layout" or by left-clicking the mouse above the displayed icon, the slabs visibility is changed. When the command is active, this icon is pushed-in, e.g., framed by a blue square (the way active icons are displayed depends on the operating system used) and then the panels will be displayed on the screen. The term "slab" means all slabs which inclination with the horizontal plane is less than  $45^\circ$ .

## 2.10.7 Walls visibility



By selecting the command **"Walls visibility"**, which is located in the drop-down menu "Layout" or by left-clicking the mouse above the displayed icon, the visibility of the walls is changed. When the command is active, this icon is pushed-in, that is, framed by a blue square (the way active icons are displayed depends on the operating system used) and then the walls will be displayed on the screen. The term "wall" refers to all slabs which inclination with the horizontal plane is greater than 45°.

## 2.13 3D Graphic Control (menu "3D View")

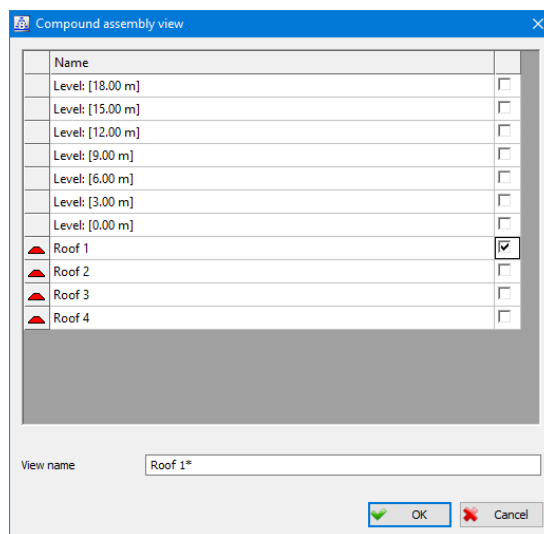
### 2.13.1 Rotation of Isometric Presentation (Orbit)

The "Direct orbit" check box is located in the "Functionality" command dialog in the "Other" folder (chapter 3.5.2.Functionality). When this check box is checked, the "Orbit" command switches to direct operation mode, which means that it is practically constantly active. In this case, the model can be rotated at any time while working with the program, by pressing the left mouse button and moving the cursor in the desired direction of rotation. By turning on the "Direct orbit" check box, the "Orbit" command in the menu that opens when you right-click the mouse over the name of the 3D view becomes unavailable.

## 2.14 Commands for Changing the Current Presentation in "2D View" Window (menu "2D View")

### 2.14.2 Edit view

It is possible to create a compound view from only one auxiliary view, which will project the influences in the selected oblique plane onto the XY plane. In the "View name" edit field, the program automatically adds the symbol "\*" after the name of the auxiliary view. You can change the name of the compound view created in this way by entering the desired text in the given edit field.



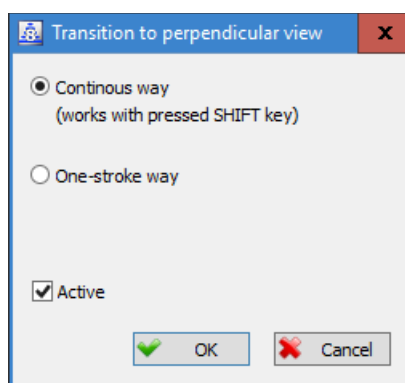
Creating a compound view from just one auxiliary view

## 2.14.10 Transition to perpendicular view




Changing the layout that is displayed in the 2D window can also be done directly from the 2D window by clicking on the entity that belongs to the layout that is being placed. This type of visibility is switched on by choosing the **"Transition to perpendicular view"** command from the "Tools" or by the left-click of the mouse above the icon displayed. When the command is active, this icon is embedded, that is, framed by a blue square (the mode of displaying active icons depends on the operating system used).

All parameters that affect the way the command operates are set in the dialog that opens with the right mouse button above the icon displayed:

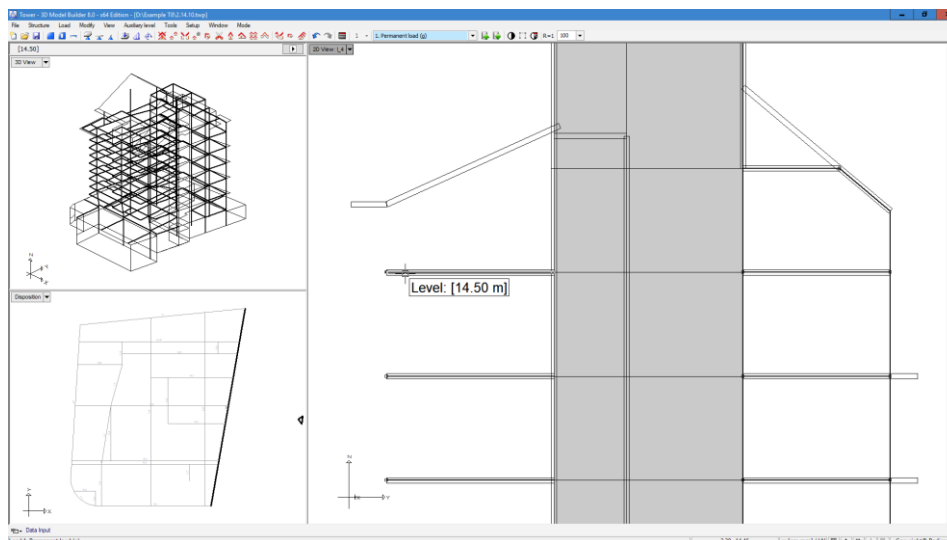


**Continous way** - When this switch is selected, the Shift key must be pressed to make the command work.

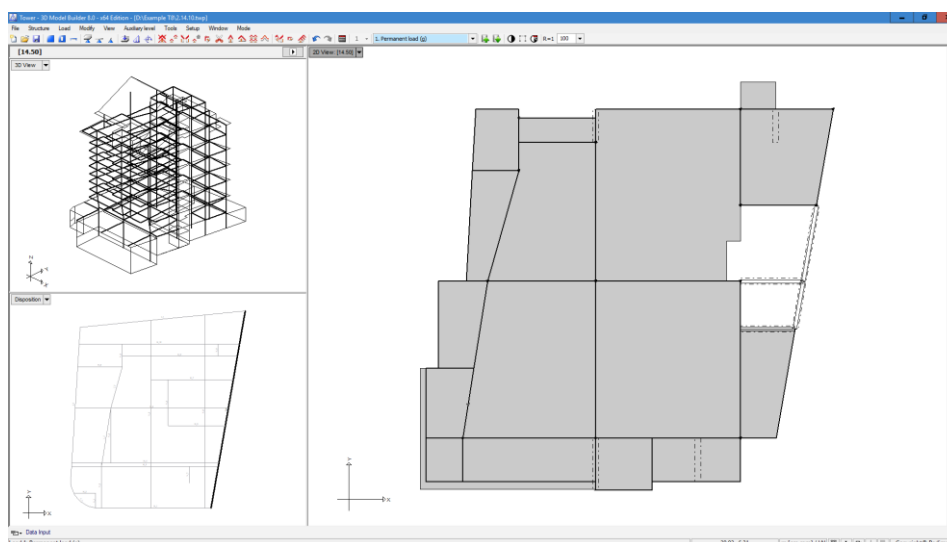
**One-stroke way** - the command works without pressing additional buttons, but only one change of layout can be made, after which the command is turned off. In order to make a new change of layout, the command must be switched on again every single time. When this mode is selected, it changes the picture of the icon that the command turns on .

**Active** - check box whose status determines whether the command is on or not.

Regardless of which mode is selected, the further running of the command is the same. First, in the 2D window, the mouse pointer should be placed above the entity that, in addition to the current layout, belongs to the layout that is being placed. The program displays a tool tip with the name of that layout, and a mouse click causes it to be placed in the 2D window.



The tool tip displays the name of the level where the slab is in section above which is the mouse arrow



Clicking the mouse above the slab in section is changed the layout of 2D window on the level where is the slab

If the mouse arrow is placed above the cross-section of the surface entity (slab, area support) and the current layout in the 2D window, the mouse click will set the plane to which the surface entity belongs to a new layout. Since the slab can belong to only one level, it is not necessary to have an assembly. In such a situation, the program in the tool tip, and later in the name of the 2D window, prints the name of the "<no name>" as the name of the layout. If the mouse arrow is placed above the cross-section of several surface entities with a current layout, the one that looks at the current layout will lock the angle that deviates at least 90 degrees.

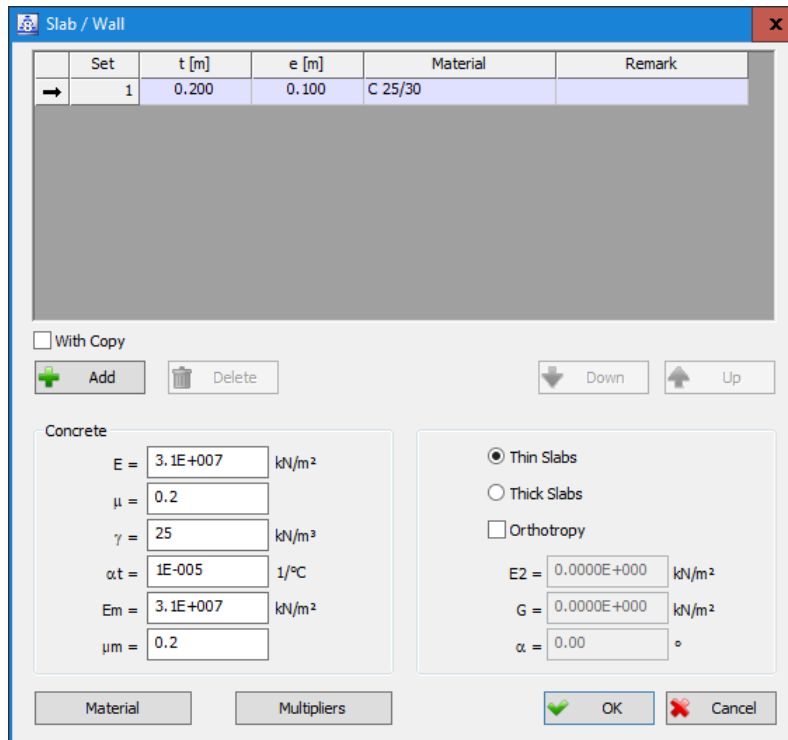
At line entities, there is a slightly different situation, since they can be at several planes simultaneously. It is therefore necessary that the line entity belongs to some previously created circuit assembly (frame, level, custom view). If this condition is not satisfied, tool tip will not be displayed when setting up the mouse arrow above the entity, which means that the change of layout by mouse click is impossible. If the mouse arrow is placed above the line entity which is the part of a number of assemblies, the one that looks at the current layout will lock the angle that deviates at least 90 degrees.

We will also note that the change of layout can also be performed when another command is active.

# 3. ENTERING DATA

## 3.1 Structure

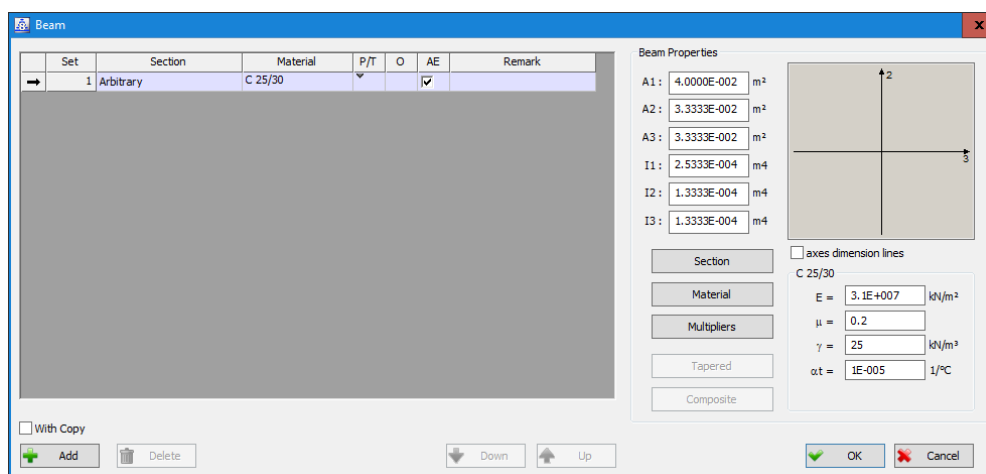
### 3.1.1 Slab/Wall



Appearance of a dialog box for numerical data on plates defining

**Remark** Edit remark entry field for plate sets. The default remark will be displayed in the plate set control dialog and in the set filter visibility dialog.

### 3.1.4 Beam

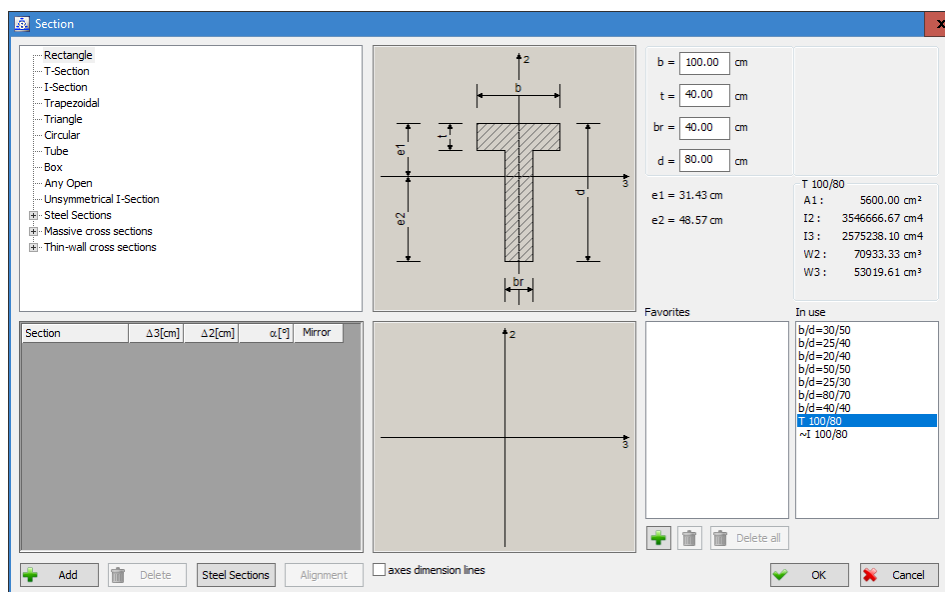


Basic appearance of the dialog box for defining the beam sets

**Remark** Edit remark entry field for beam sets. Added remark will be displayed in the beam sets report, in the beam sets control dialog, and in the sets filter visibility dialog.

### Shaping the Cross-sections

**In use** - in this part of dialog is displayed the list of all sections in use in the current model. When there is a need, each of these sections can be easily selected from the list by clicking the mouse over its name. In this way, it is possible to avoid re-definition of the sections that are already being used in the model. It is also very easy to create new sections similar to someone in the list.

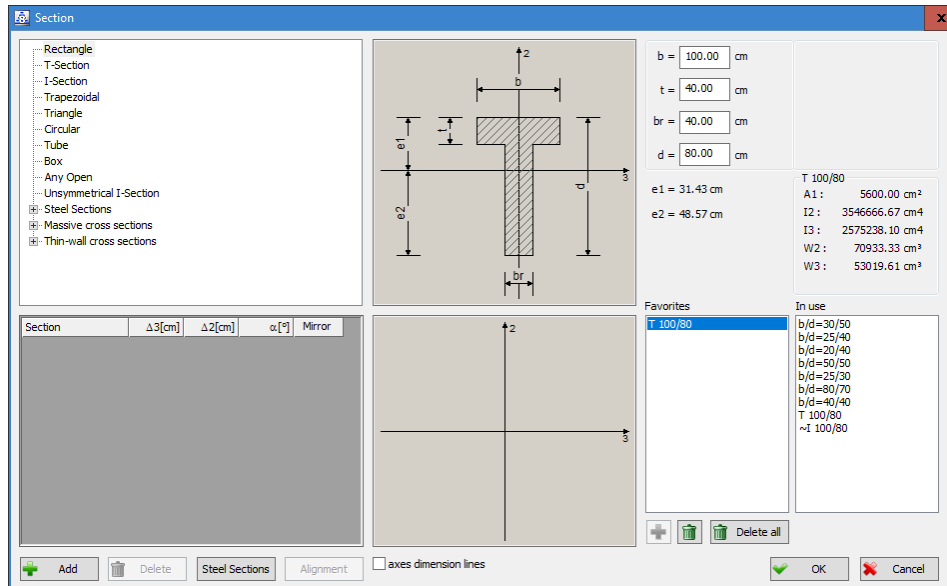


The T section of  $b/d=100/80$ cm is selected from the section in use

**Favorites** - While operating with the program, you can add all the sections you use frequently to your favourite section list. With one click of the mouse over its name, these sections can be selected at any time from the list and assigned to the current set or edited to get a similar section. This avoids the constant setting of the same sections from the beginning, and therefore shortens the time required to define the beam sets in which they are used.



- a button used to insert a cross section into a favorite list. The geometry of the intersecting section must be correctly specified, otherwise the program will display the corresponding message when attempting to insert. If the section already exists in the list, the insertion button will be unavailable.

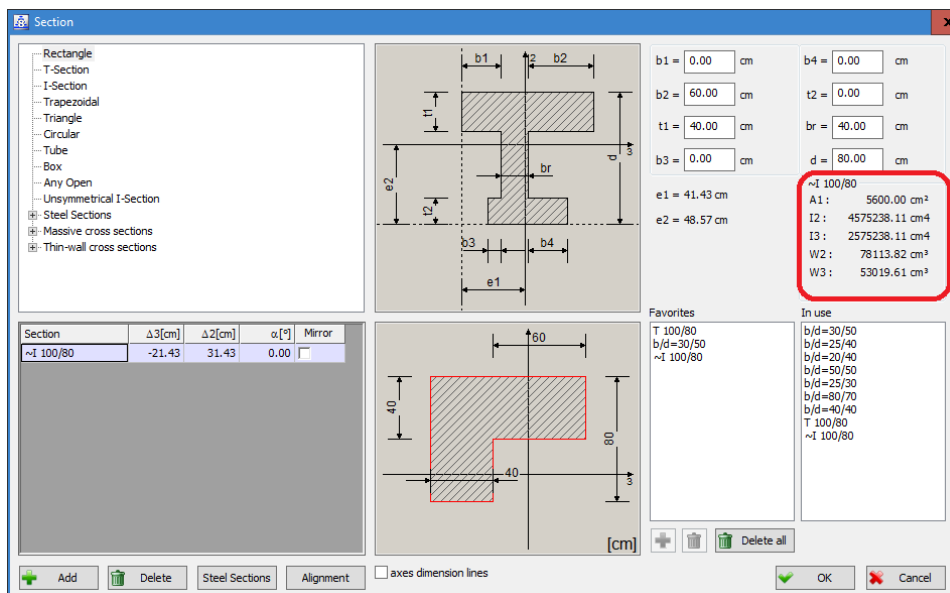


In the list of favorites, the T section of the dimension  $b/d=100/80$ cm is inserted

 - clicking on this button deletes the selected section from the list of favorite sections.


 Delete all - a button used to delete all sections from a favorites list.

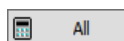
**Geometric characteristics of the sections** - after defining a new cross section, that is, after selecting one of the previously defined cross sections from the favorite list and cross-sections in use, the program calculates its geometric characteristics and displays them in a dialog.



Display geometric characteristics of an assigned section

### Editing the Steel Section Library

 -the geometric profile characteristics of the program can be calculated on the basis of its dimensions so that they do not have to be set when adding new profiles or, when editing the existing. By clicking on the button, the calculation of the geometric characteristics of the selected profile in the table is performed.



All

- By clicking this button, the calculation of the geometric characteristics is performed simultaneously for all profiles from the selected profile group.

The data that must be given when creating the profiles are marked in white in the table, while the geometric characteristics of the profile that the program can calculate based on the entered dimensions in the table is marked with a special colour.

Steel Sections	h[mm]	b[mm]	s[mm]	t[mm]	r1[mm]	r2[mm]	A1[cm <sup>2</sup> ]	A2[cm <sup>2</sup> ]	A3[cm <sup>2</sup> ]	I1[cm <sup>4</sup> ]	I2[cm <sup>4</sup> ]	I3[cm <sup>4</sup> ]
I 80	80.00	42.00	3.90	5.90	3.90	2.30	7.57	3.12	4.45	0.87	6.29	77.80
I 100	100.00	50.00	4.50	6.80	4.50	2.70	10.60	4.47	6.13	1.60	12.20	171.00
I 120	120.00	58.00	5.10	7.70	5.10	3.10	14.20	6.14	8.06	2.71	21.50	328.00
I 140	140.00	66.00	5.70	8.60	5.70	3.40	18.20	7.92	10.28	4.32	35.20	573.00
I 160	160.00	74.00	6.30	9.50	6.30	3.80	22.80	10.06	12.74	6.57	54.70	935.00
I 180	180.00	82.00	6.90	10.40	6.90	4.10	27.90	12.41	15.49	9.58	81.30	1450.00
I 200	200.00	90.00	7.50	11.30	7.50	4.50	33.40	14.92	18.48	13.50	117.00	2140.00
I 220	220.00	98.00	8.10	12.20	8.10	4.90	39.50	17.77	21.73	18.60	162.00	3060.00
I 240	240.00	106.00	8.70	13.10	8.70	5.20	46.10	20.83	25.27	25.00	221.00	4250.00
I 260	260.00	113.00	9.40	14.10	9.40	5.60	53.30	24.34	28.96	33.50	288.00	5740.00
I 280	280.00	119.00	10.10	15.20	10.10	6.10	61.00	28.21	32.79	44.20	364.00	7590.00
I 300	300.00	125.00	10.80	16.20	10.80	6.50	69.00	32.36	36.64	56.80	451.00	9800.00
I 320	320.00	131.00	11.50	17.30	11.50	6.90	77.70	36.75	40.95	72.50	555.00	12510.00
I 340	340.00	137.00	12.20	18.30	12.20	7.30	86.70	41.46	45.24	90.40	674.00	15700.00
I 360	360.00	143.00	13.00	19.50	13.00	7.80	97.00	46.81	50.19	115.00	818.00	19610.00
I 380	380.00	149.00	13.70	20.50	13.70	8.20	107.00	52.08	54.92	141.00	975.00	24010.00

The geometric characteristics of the section that the program can calculate

### 3.1.9 Generate thin-wall section girder



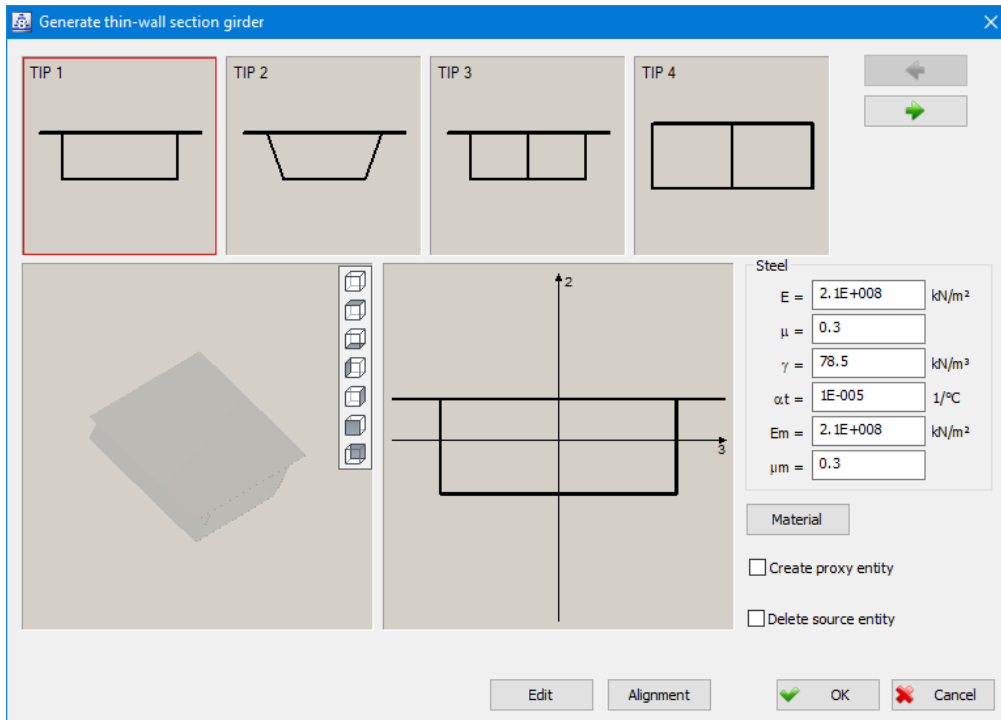
Using this command, an entire beam, straight or curved paths can be generated from the selected thin-walled section, so that the program creates a slab of the appropriate geometry from each segment of the given section. After running the command, the following message appears on the command line:

First point of axis (Select beam):

In addition to specifying the axis of the beam, it is also possible to select a beam, which avoids entering the geometry of the beam with a local coordinate system, but instead uses the existing geometry of the selected beam. By selecting the "**Select beam**" sub-option from the command line, you enter the beam selection procedure:

<0 sel.> Beams - Select (Window / Polygon / Section / selection Groups / Resent selection / <End>):

After selecting the beam, the following dialog opens:

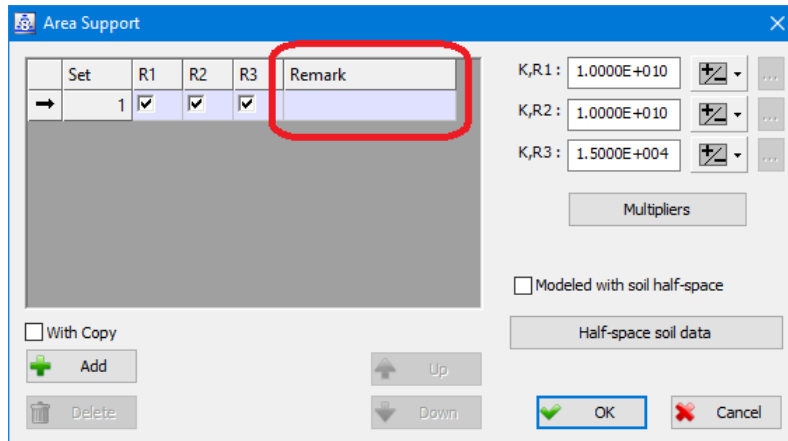


Layout of the dialog in which the parameters for generation are defined thin-wall section girder

### Delete source entity

By turning on the check box, after the generation of the thin-wall section girder, the beam whose geometry has been taken over will be deleted.

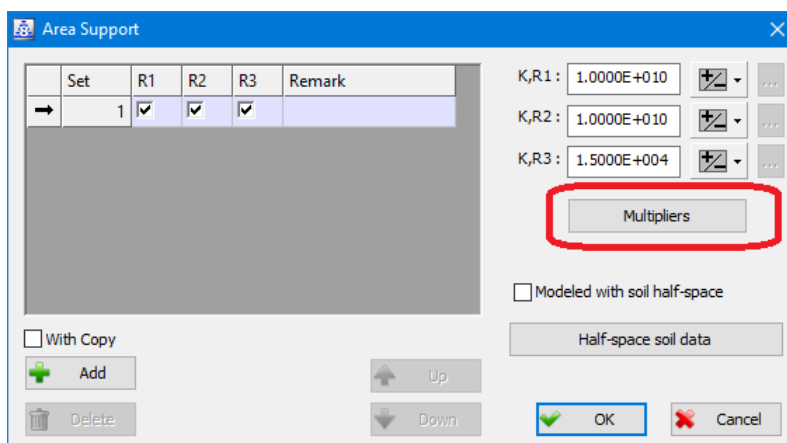
### 3.1.10 Area Support



Dialogue layout for defining sets of surface supports

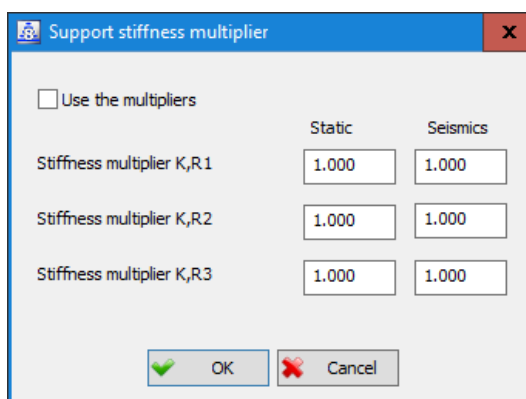
**Remark** Field for entering Remark for surface support sets. The assigned remark will also be displayed in the dialogue for controlling the sets of surface supports, if the check box "**display remarks**" is turned on (see chapter "8.23") and in the dialogue for setting the visibility filters by sets.

## Surface stiffness multipliers



Button for setting stiffness multipliers surface supports

Each set of surface supports can be assigned multipliers of the stiffness of the supports independently for the static and seismic model. For this purpose, a button "**Multipliers**" is provided, which activates a dialog of the following appearance:



The appearance of the dialogue in which they are assigned surface stiffness multipliers

Multiplier descriptions are located in front of edit boxes where their values are set independently for static and seismic models. The following can be specified:

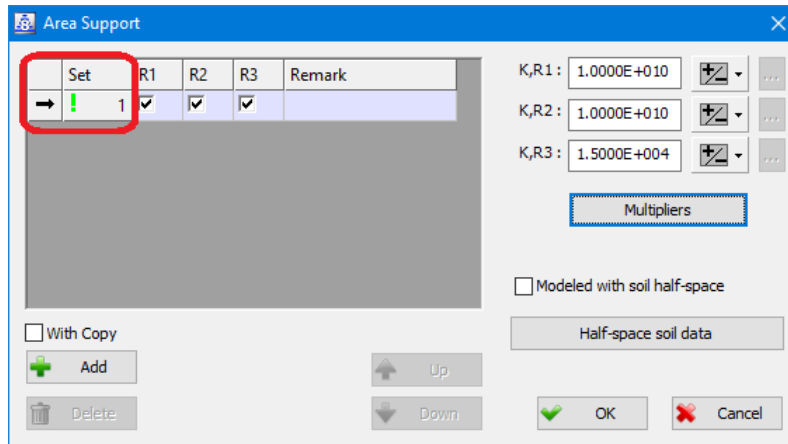
- **Stiffness multiplier K,R1**

- **Stiffness multiplier K,R2**

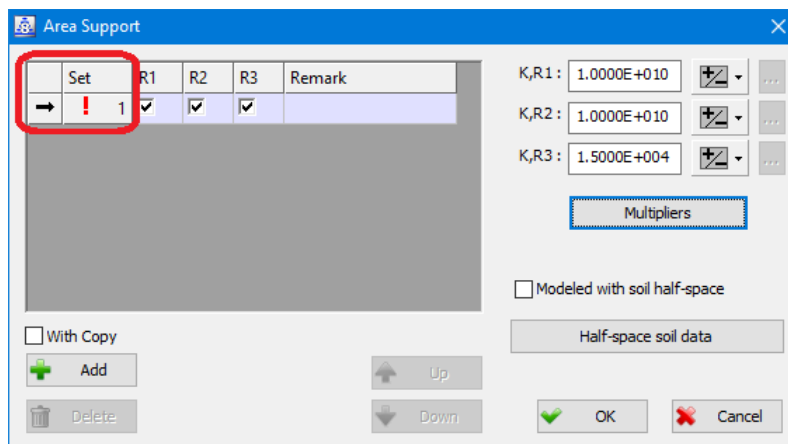
- **Stiffness multiplier K,R3**

- **Use the multipliers** - A check box whose on state indicates that the multipliers defined in this dialog will be applied when calculating the model.

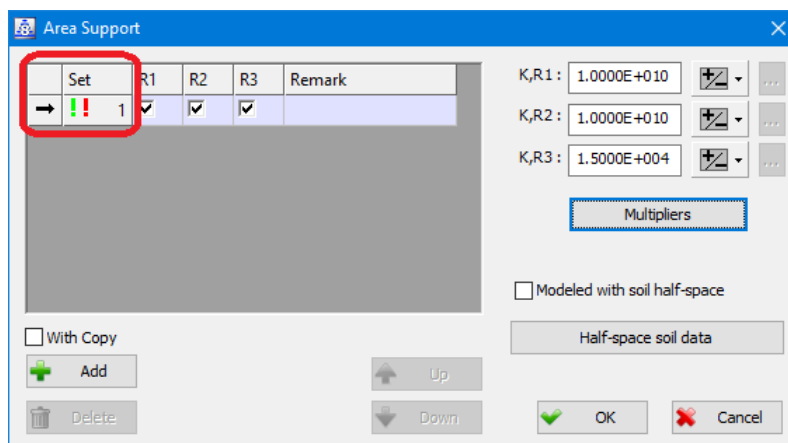
Exiting the dialog box on the "**OK**" button, the program returns to the dialog for surface support sets, and in a row with ordinal numbers, sets of surface supports that include the use of multipliers sets a special icon to distinguish it from the others. Depending on whether the use of multipliers for static, seismic only, or for both static and seismic models is included, three different frames can be found in addition to the sequence number.



An icon indicating that this set is a surface support included use of multipliers for the static model

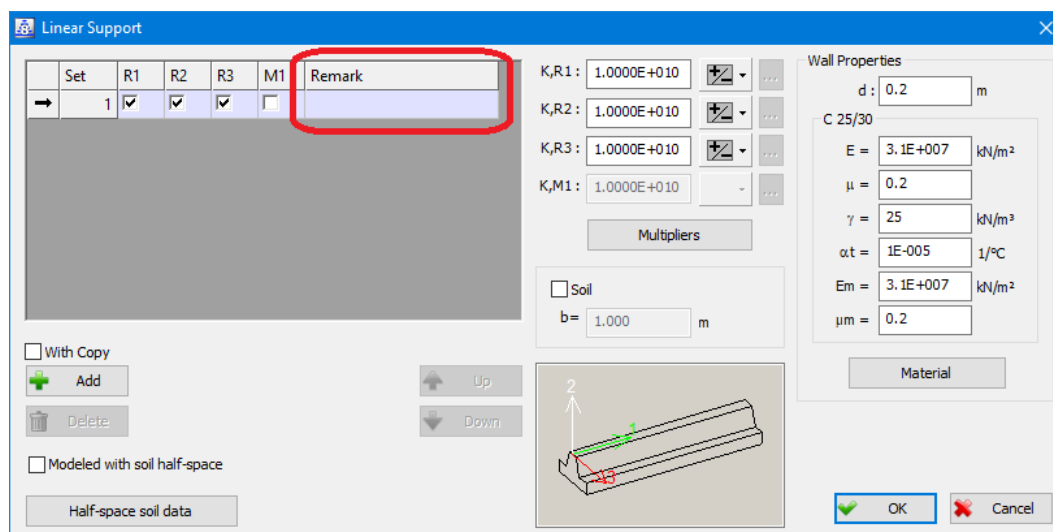


An icon indicating that this set is a surface support involved the use of multipliers for the seismic model



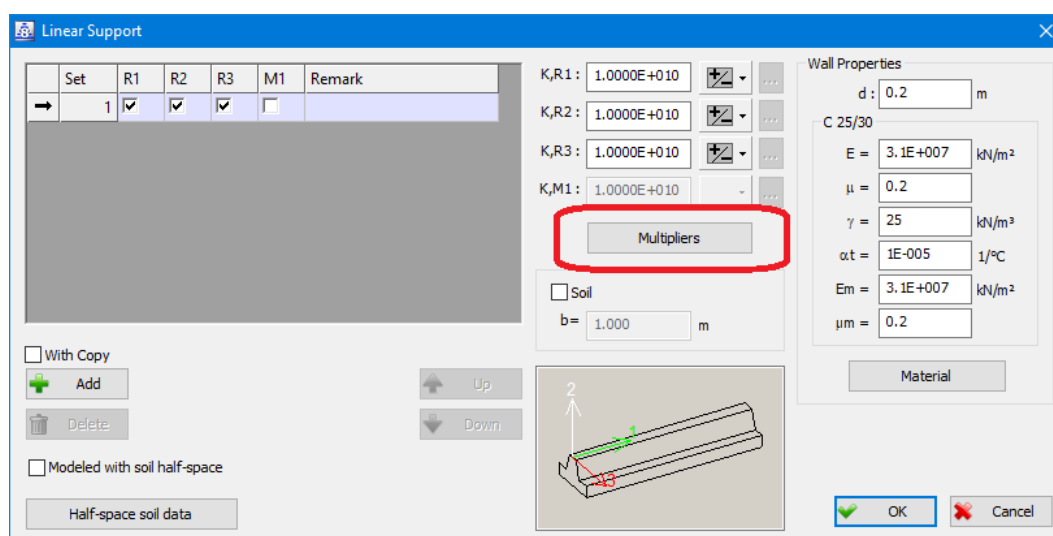
An icon indicating that this set is a surface support the use of multipliers for both static and seismic models is included

### 3.1.11 Linear Support



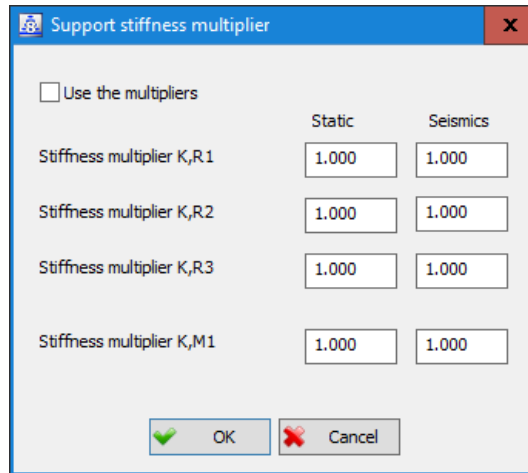
**Remark** Field for entering Remark for linear support sets. The assigned remark will also be displayed in the dialogue for controlling the sets of linear supports, if the check box **"display remarks"** is turned on (see chapter "8.23") and in the dialogue for setting the visibility filters by sets.

#### Support stiffness multiplier



Button for setting the line stiffness multipliers

Each set of line supports can be assigned multipliers of the stiffness of the supports independently for the static and seismic model. For this purpose, a button **"Multipliers"** is provided, which activates a dialog of the following appearance:



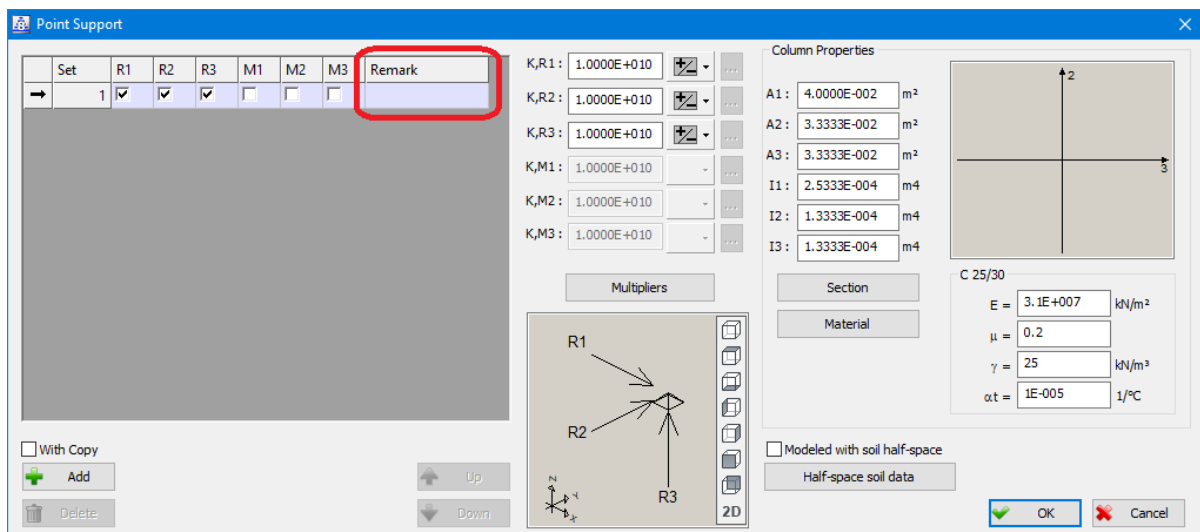
The appearance of the dialogue in which they are assigned stiffness multipliers of line supports

Multiplier descriptions are located in front of edit boxes where their values are set independently for static and seismic models. The following can be specified:

- **The stiffness multiplier  $K,R,1$**
- **The stiffness multiplier  $K,R,2$**
- **The stiffness multiplier  $K,R,3$**
- **The stiffness multiplier  $K,M,1$**
- **Use the multipliers** - A check box whose on state indicates that the multipliers defined in this dialog will be applied when calculating the model.

The icons that are placed next to the ordinal numbers of the sets of line supports with the given multipliers have the same meaning as for the sets of surface supports (see section 3.1.10).

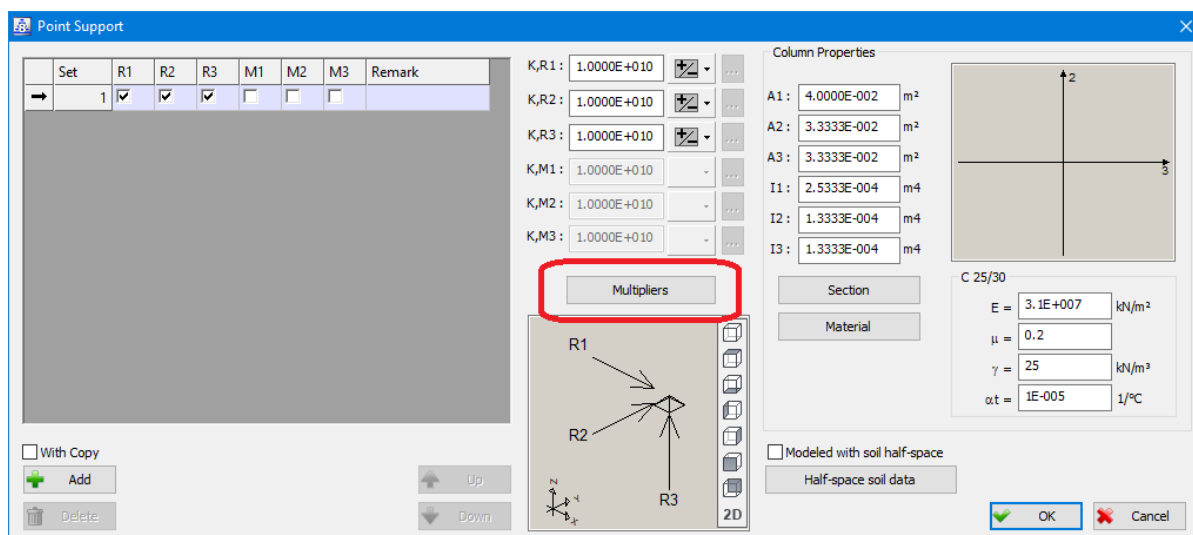
### 3.1.12 Point Support



Dialogue layout for defining sets of point supports

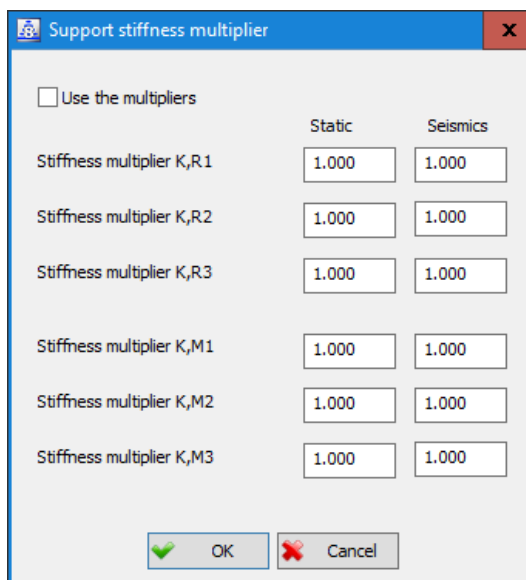
**Remark** Field for entering Remark for point support sets. The assigned remark will also be displayed in the dialogue for controlling the sets of point supports, if the check box "**display remarks**" is turned on (see chapter "8.23") and in the dialogue for setting the visibility filters by sets.

### Support stiffness multiplier



Button for setting point stiffness multipliers

Each set of point supports can be assigned multipliers of the stiffness of the supports independently for the static and seismic models. For this purpose, a button "**Multipliers**" is provided, which activates a dialog of the following appearance:



The appearance of the dialogue in which they are assigned point stiffness multipliers

Multiplier descriptions are located in front of edit boxes where their values are set independently for static and seismic models. The following can be specified:

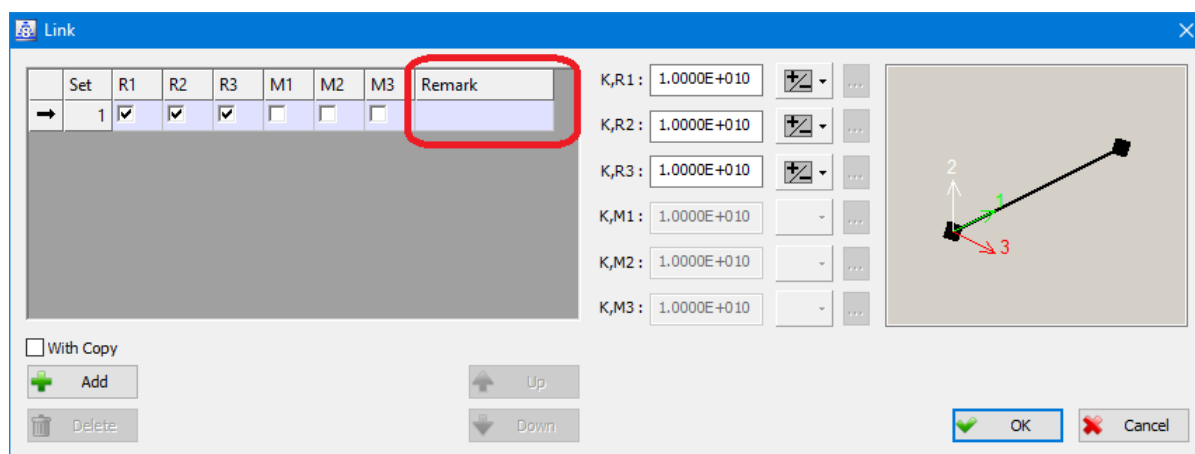
- **The stiffness multiplier K,R,1**
- **The stiffness multiplier K,R,2**

- The stiffness multiplier  $K,R3$
- The stiffness multiplier  $K,M1$
- The stiffness multiplier  $K,M2$
- The stiffness multiplier  $K,M3$

**Use the multipliers** - A check box whose on state indicates that the multipliers defined in this dialog will be applied when calculating the model.

The icons that are placed next to the ordinal numbers of the set of point supports with the given multipliers have the same meaning as for the sets of surface supports (see section 3.1.10).

### 3.1.15 Link



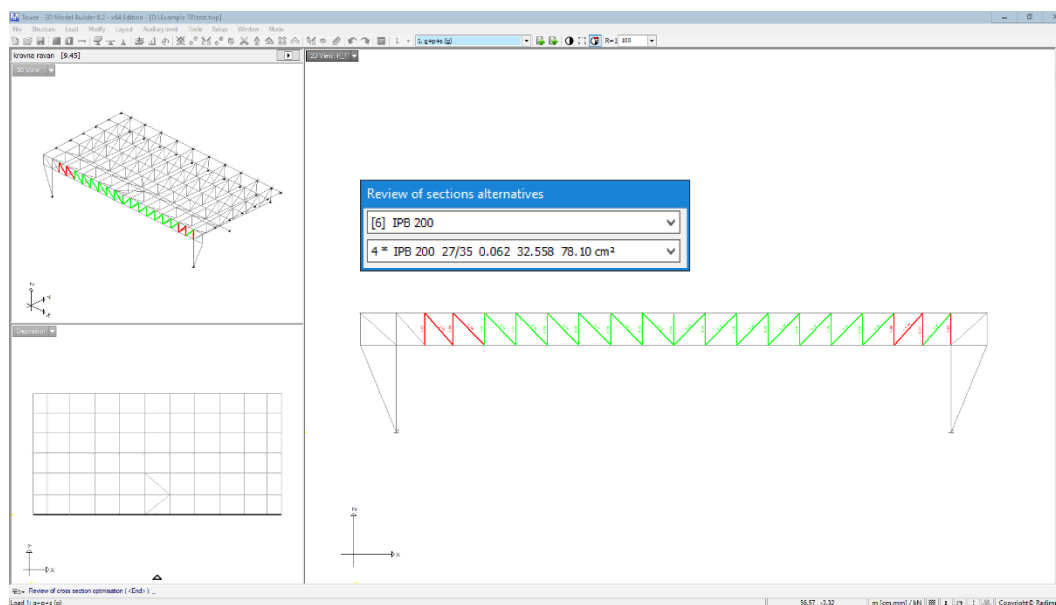
Dialogue layout for defining connection sets

**Remark** Field for entering Remark for link sets. The assigned remark will also be displayed in the dialogue for controlling the sets of links, if the check box "**display remarks**" is turned on (see chapter "8.23") and in the dialogue for setting the visibility filters by sets.

### 3.1.17 Review of cross section optimisation

The cross-section optimization process allows checking the dimensioning of the adopted cross-sections and provides the user with the information whether it can use another profile instead of one profile used in the construction.

By selecting the "**Review of cross section optimisation**" command, located within the "Structure" drop-down menu, which is available if the cross-section optimization process has previously been performed in the result processing module, the user can easily and quickly find the location and number of rods which do not satisfy the stability control or are oversized and then replaced with new cross sections.



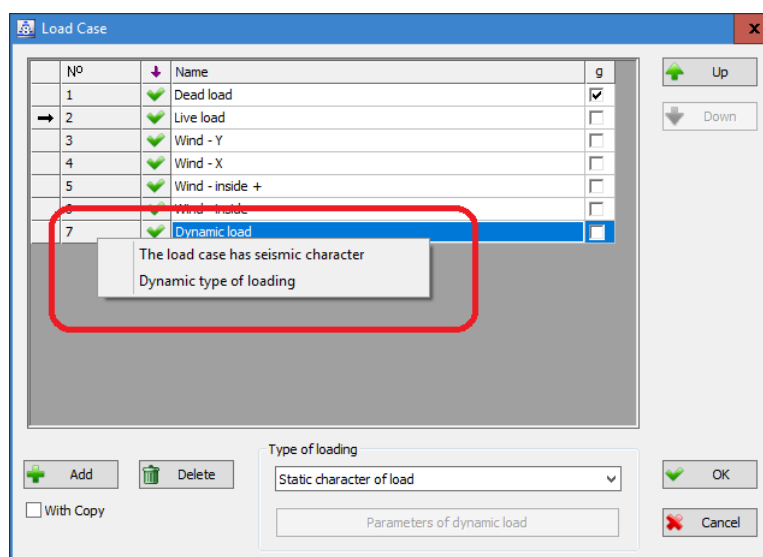
Review of cross section optimisation in the data entry module

Operating with command "Review of cross section optimisation" is described in chapter 10.11, and we will not explain it again.

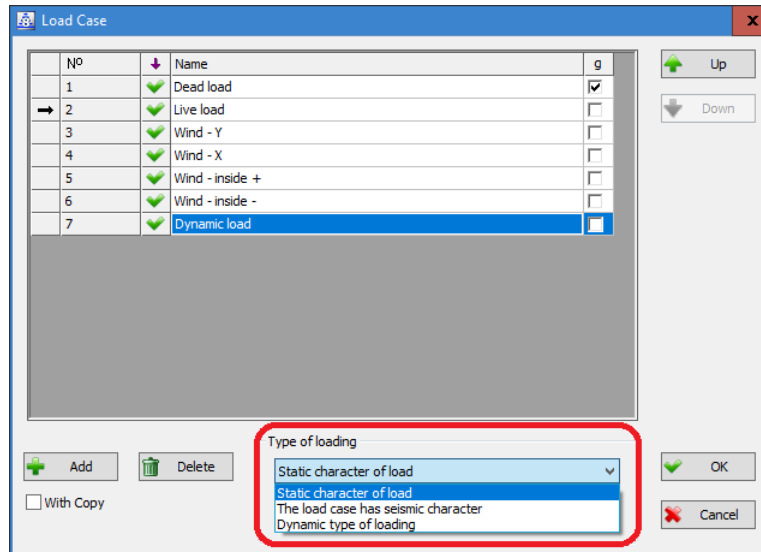
## 3.2 Loading

### 3.2.1 Defining the Load Cases

If the mouse arrow is placed above the regular number of a load in the list, and the right button is pressed, the drop-down menu with options "The load case has seismic character" and "Dynamic type of loading" will open.

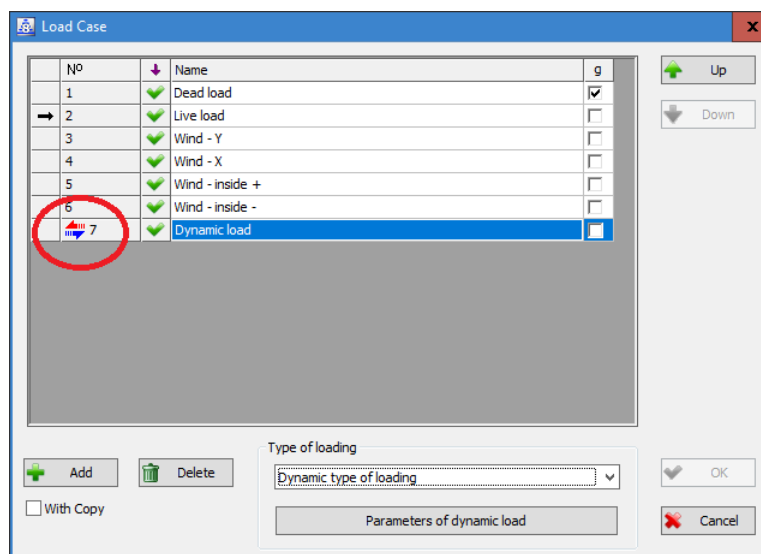


The load characteristic can also be selected from the closed list below the table of load cases:



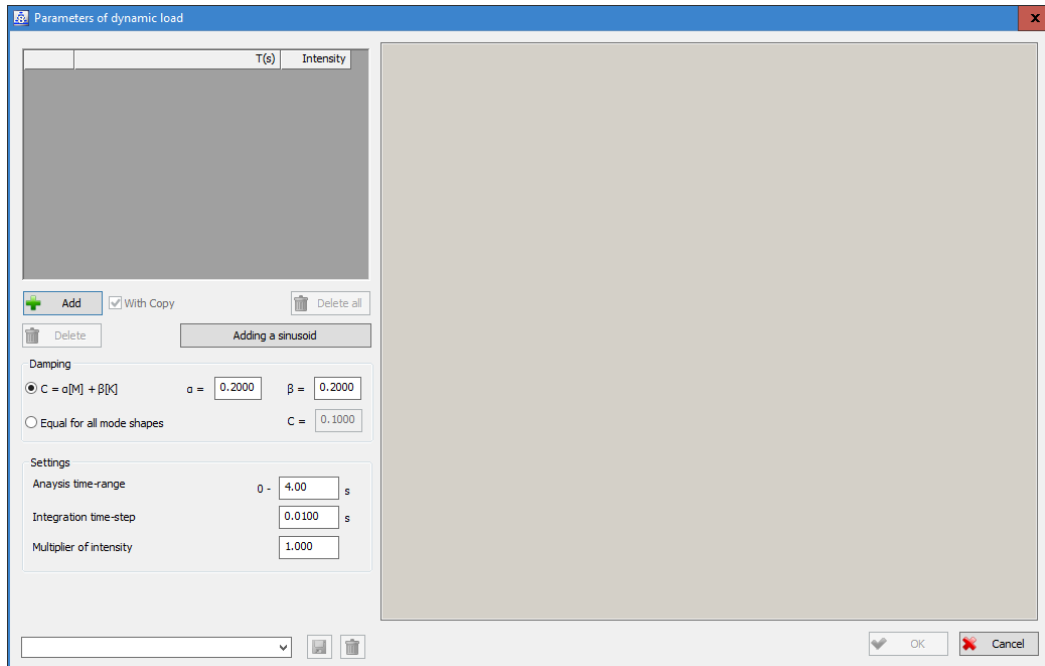
**Static character of load** - This is the default character of the load and is automatically assigned to each load case when creating it. If a load case has been changed by mistake, the choice of this item from the list can restore the default value.

**Dynamic type of loading** - by selecting this option, the load is assigned a dynamic character, and in front of its regular number, a special mini picture is placed to distinguish it from other load cases. The "Dynamic type of loading" option is only available with the "Expert" license for the Tower program.



A mini picture indicating that the load is the case included dynamic character

Whenever you select such a load from the list, the "**Parameters of dynamic load**" button becomes available. With its choice, a dialog opens in which all the data required for the dynamic load calculation is obtained.




Dialog where is set the data of dynamic load

In the upper left corner of the dialog there is a table defining the dynamic load. It consists three columns:

- ordinal number
- **T(s)** - a column where the time moments of the dynamic load are added.
- **Intensity** – a column in which the intensity of the dynamic load is given at a given moment of time. You can enter values from -1 to +1 and multiply with them all the loads that are in the given load case.

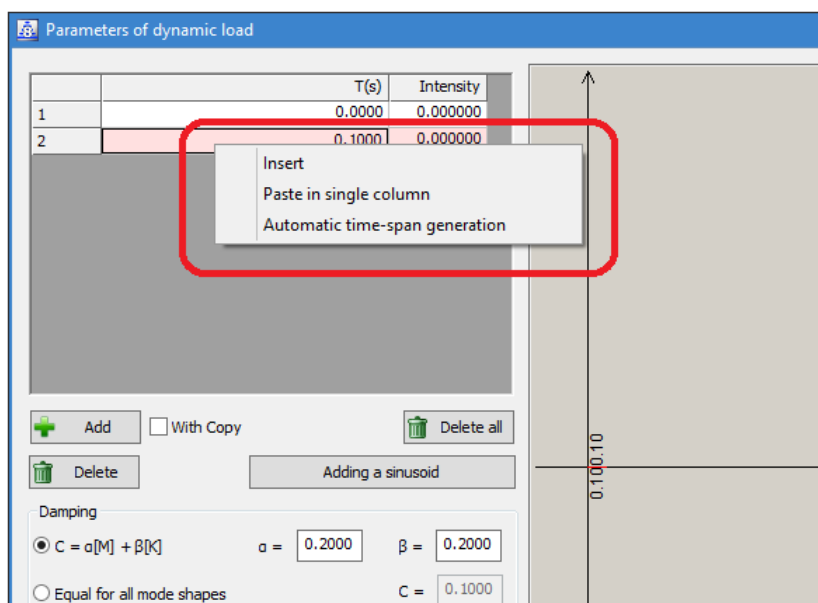
In this way, arbitrary load behaviour can be set, such as oscillation of machines or various types of impacts (impact of vehicles in the bridge column, etc.).

 **Add** A button that activates the new row in the table. If the check box "With Copy" is enabled, adding a new row is done by copying all the data entered in the currently selected row.

 **Delete** The button which activation throws the currently selected row from the table.

 **Delete all**  
The button which activation removes all rows from the table.

The table can also contain content copied from another table, which greatly accelerates data entry. Right-clicking the mouse above any cell in the table opens the drop-down menu with options "**Insert**" and "**Paste in single column**".



Drop down menu which is opened by right clicking of the mouse over the cell

**Insert** - In case that a data is copied, activating the "Insert" option will extract the copied content into the selected cell. If more data is copied, they will be inserted in the table in the horizontal, starting with the currently selected cell.

**Paste in single column** - When this option is selected, all the data is copied into one column of the table starting with the currently selected cell.

In case there is not enough number of rows in the table, the program automatically generates as many rows as needed to insert all the data extracted.

**Automatic time-span generation** - This item appears in the drop-down menu only if it opens above the column "T (s)" and if there are at least two time moments in it. Its choice is to automatically generate time intervals starting from the selected moment of the time, with the generated intervals equal to the difference between the selected and the previous moment of the time.

**Sort in ascending order** - Right-clicking the mouse above the column header "T (s)" opens the drop-down menu in which this command is located. With its choice, all time moments in the column is edited in a rising order.

**Adding a sinusoid** - Dynamic load can also have a sinusoidal shape. Clicking on this button opens a dialog where all the parameters necessary for the program are set to create the desired sinusoid based on them.

dialog for sinusoid defining

**Starting time** - the field where the moment of time is set in which starts the sinusoid.

**Step** - the field where the length of time interval is set. The total time of sinusoid will be divided by the length of time interval.

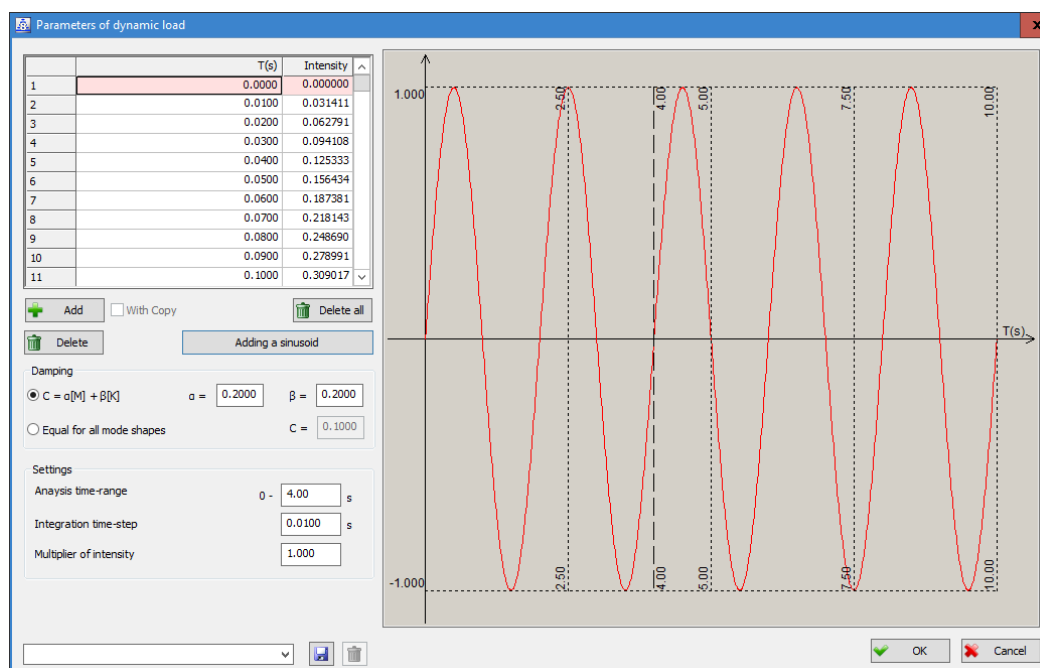
**Total duration** - the field in which the total duration of the sinusoid is set.

**Full cycle duration** - the field in which the duration of a sinusoidal period is determined.

**Cycle start position** - the field in which the shift of the starting position of the sinusoid is set. The entered value represents a percentage of the duration of a single period.

**Amplitude** - the field in which the value of the sinusoid amplitude is set.

After exiting the dialog, the program generates all the intervals of the sinusoid in which the dynamic load functions and inserts them into the table on the basis of the set data.



Oscillation of the dynamic load in sinusoid shape is set

## Damping

The program allows defining the damping in one of two ways:

### **C = $\alpha$ [M] + $\beta$ [K]**

Formula defines the damping. Edit fields for entering the damping coefficient become active if the switch is set to ON:

**$\alpha$**  = Edit field to enter the damping coefficient.

**$\beta$**  = Edit field to enter the damping coefficient.

### **Equal for all mode shapes**

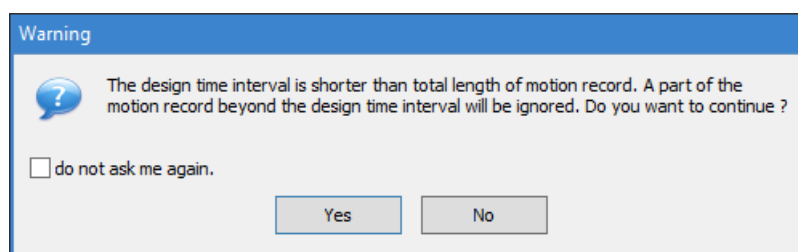
Activating this switch defines the damping that will be equal for all the tones.

**C** = Edit field to enter the damping.

## Settings

### **Calculation interval of time**

edit field where is set the interval of the time and it will be calculated. If the calculating interval of time is shorter than the whole length of period of dynamic load, program will display the following alert:



### **Integration time-step**

the field where is set the length of time intervals, which will be used for calculation. It's recommended to set "fine" or shorter (at least the same) intervals than the intervals on which is set the dynamic load, to have more precise results.

### **Multiplier of intensity**

the edit field for entering a value that will multiply the intensity of the dynamic load.

In the lower left part of the dialog there is a list with all the dynamic loads that were previously defined and recorded so that it could be used in other models. Any of them can be selected and set as a current program at any time.

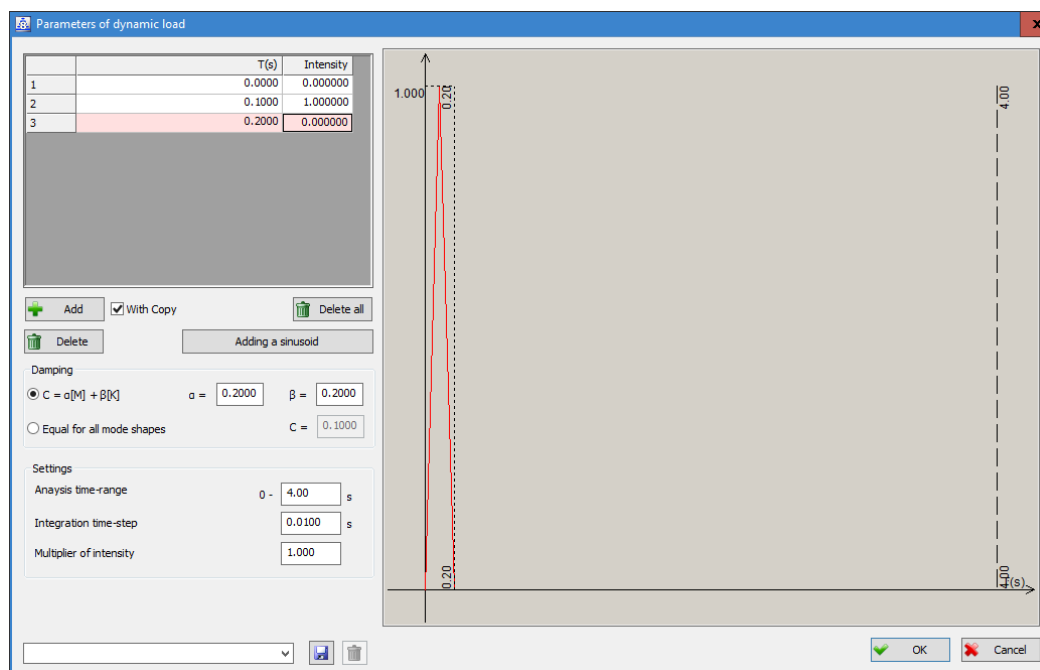


A button which activation records the given dynamic load, or records the current state of all parameters in the dialog.



The button which activation deletes the selected dynamic load from the list.

In the right part of the dialog, a graphic interpretation of the given dynamic load is displayed.



A graphic layout of a set dynamic strike

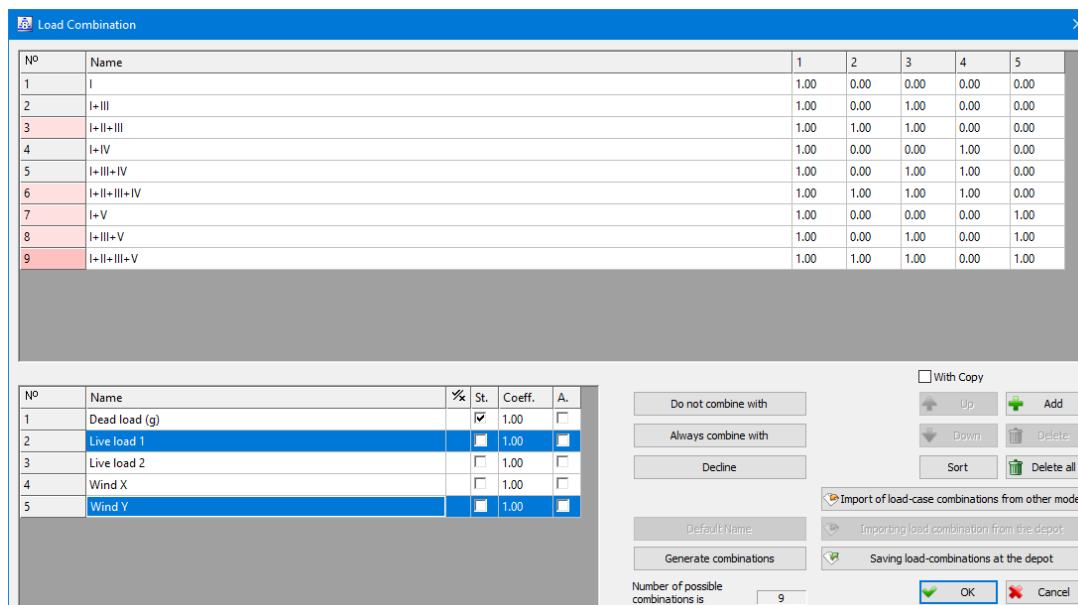
By choosing the “**OK**” command box, the program returns to the basic dialog of the command to define load cases, and all the given data in this dialog joins the current dynamic load case.

### 3.2.2 Defining the Load Combinations

After selecting a basic load case, cells with ordinal numbers of all combinations that contain it are marked with a special color.

By selecting the basic load case “Live load 1” the ordinal numbers of combinations containing it (3, 6, 9) are colored

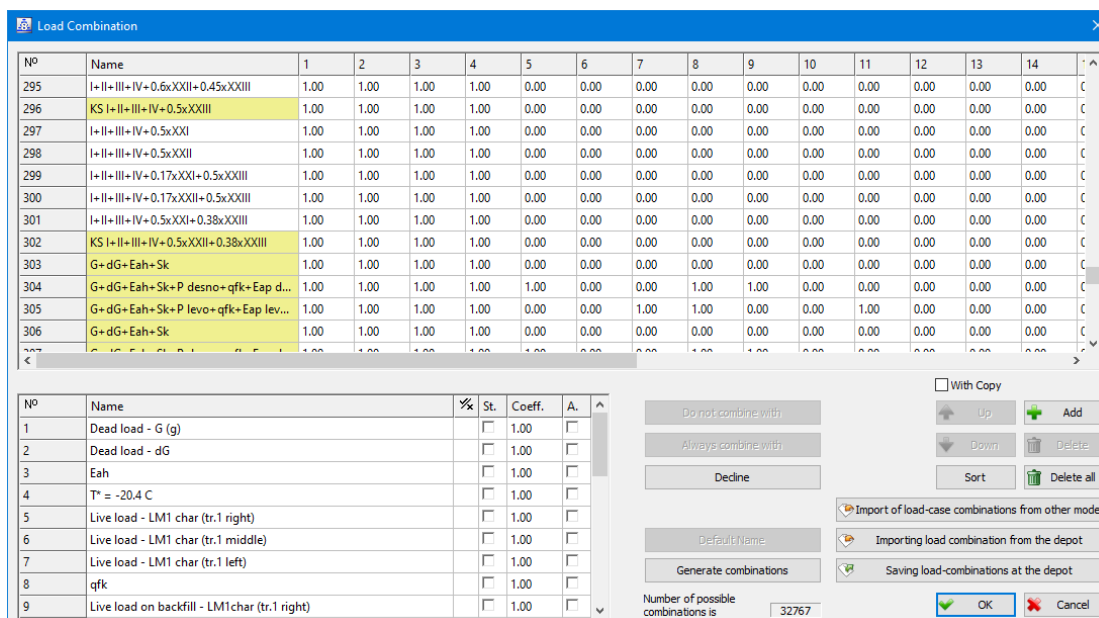
If several basic load cases are selected, the combinations that contain all of them will be marked with one color, and the combinations that contain just some of them, with another color.



Only the combination with ordinal number 9 contains both selected basic cases load and it is marked with a darker color

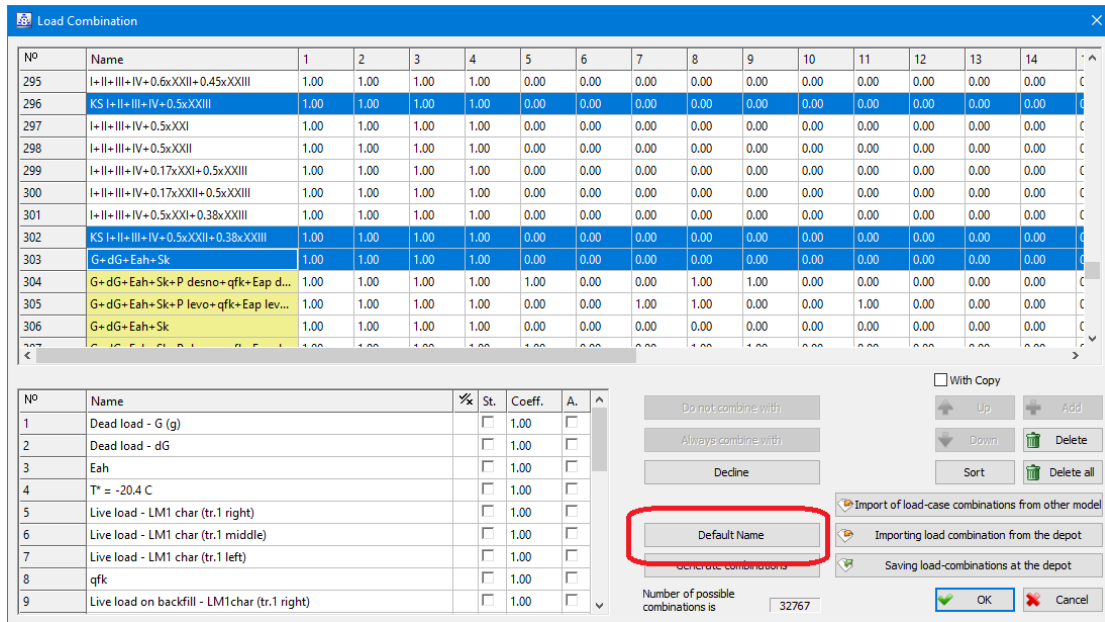
It is possible to simultaneously move several selected load combinations using the “ Up” and “ Down” command fields, provided that they form a continuous set.

In the list with load combinations, names of combinations that are different than “generic” are marked with a special color:



Names of combinations that are not “generic” are marked in yellow

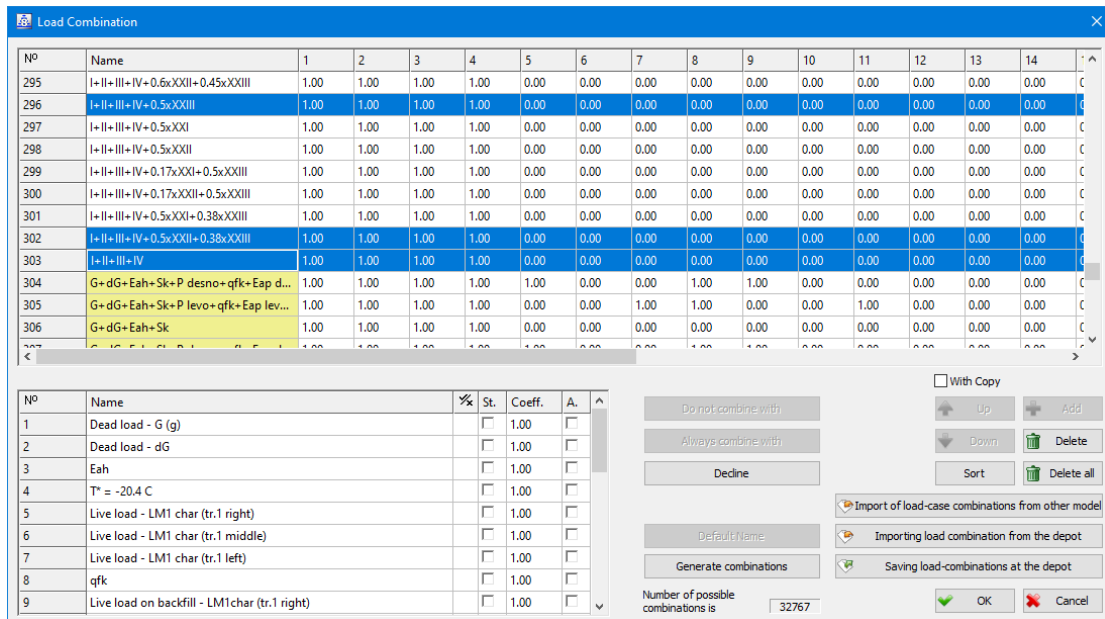
It is possible to restore user-defined combination names to “generic” names.



Button to generate name combinations

**Default Name**

Button with which the program generates names with selected combinations based on the given values of the coefficients with the basic load cases.

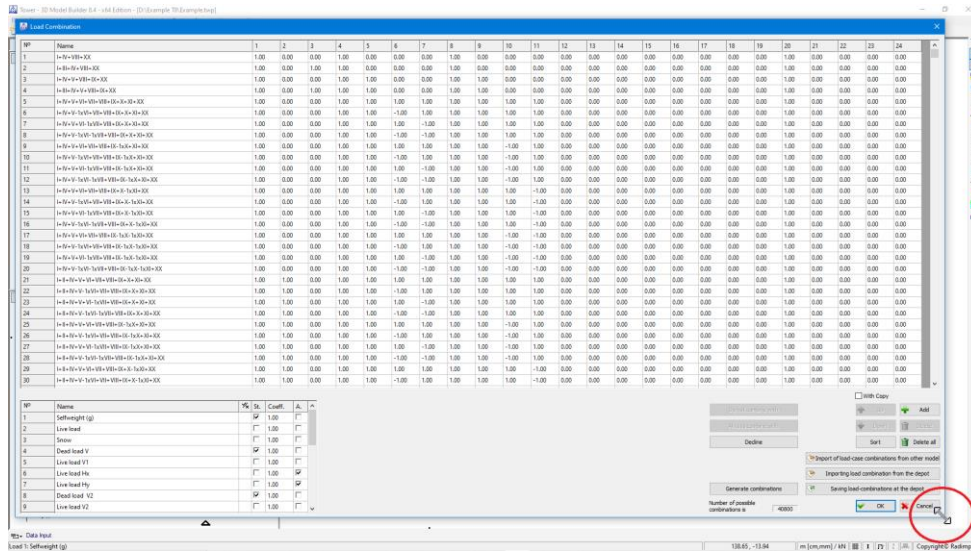


Names of selected combinations after the command "Default Name" were reverted to "generic"

**Decline** - the button by which they are cancelled, or set to the default state, all the coefficients and combination conditions specified for load cases. Columns "St." (Status), "Coeff." (Coefficient), "A." (Alternatively) and the conditions "Do not combine with" and "Always combine with" are cancelled.

In order to display as many columns with basic load cases as possible, the size of the dialogue can be changed by stretching. Firstly, it is necessary to place the mouse pointer on one of the edges of the dialogue or on one of its corners, where it will change its shape into an arrow that

shows possible directions of stretching. When this happens, you should press the left mouse button and stretch the dialogue to the desired position without releasing it.



Stretching the lower right corner changed the size of the dialogue

### 3.2.3 Area Load

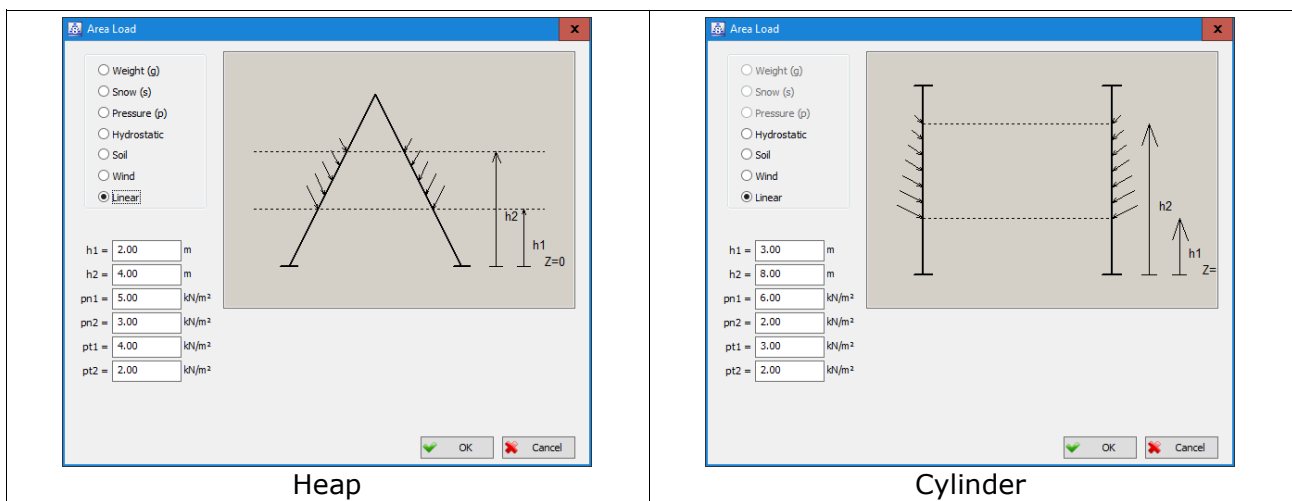
Load is distributed over the linear system

In addition to line entities, it is possible to select the edges of the slabs that are in the same assembly where the load is located.

#### 3.2.3.1 Area Load on Evolved Surfaces

##### Linear

This type of load can be assigned to the heap and cylinder.



The load is defined by providing the following data:

- h1** – global Z coordinate of linear load onset.
- h2** – global Z coordinate of end of linear load action.

**pn1** – force intensity in the direction of normal to the surface, at the beginning of the action of the load.

**pn2** –the intensity of the force in the direction of normal to the surface, at the end of the action of the load.

**pt1** –the intensity of the force in the direction of the derivative surface, at the beginning of the action of the load.

**pt2** –the intensity of the force in the direction of the derivative surface, at the end of the action of the load.

### 3.2.4 Linear Load

#### Local Coord. System of Beam

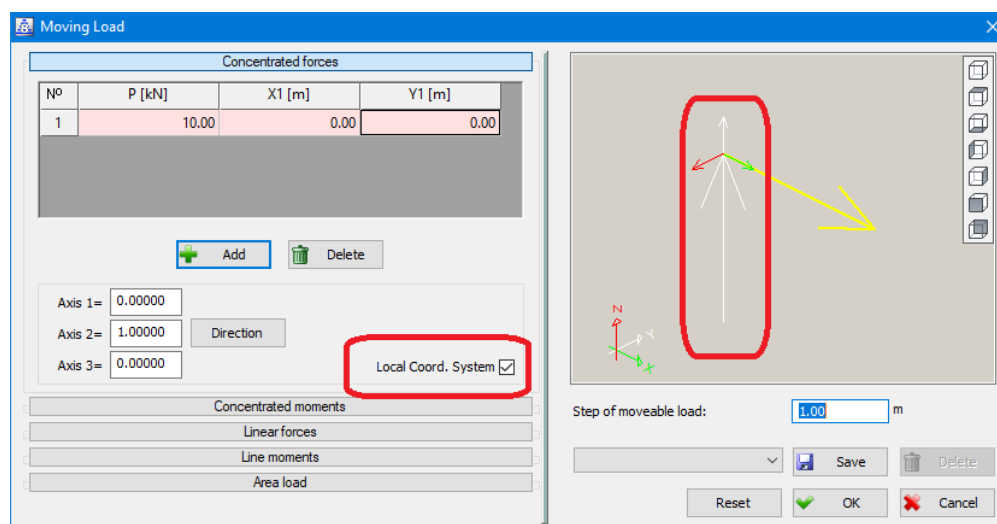
The vector of action of the line load is placed in the direction of one of the local axes of the beam on which the load falls. This switch will only be available if all the specified segments of the load's polyline are on the beams, along their entire length. The direction in which the local axes of the beam will be directed is the load vector determined in the same way as with the option "Local Coord. System of 2D View".

### 3.2.10 Moving Load

In the dialog for defining numerical data on moving load, in the section for specifying concentrated forces, linear forces and area loads, new check boxes "**Local Coord. System**" have been added.

#### Concentrated forces

**Local Coord. System** – When this check box is set to the on state, the direction of action of the selected force is set in relation to the local coordinate system of the polyline that determines the path of the moving load. From the moment the LKS axis check box is turned on, the trajectories are displayed on the drawing of the given load in the right part of the dialog.



A concentrated force acting in the direction of the local axis 2 is given moving load paths

Everything that has been said here about concentrated forces is also valid for line forces and surface loads.

Improvement in setting the moving load: it is allowed that two loads have the same geometry if their vector of the direction of action is different. This applies to concentrated forces, linear forces and area loads.

### 3.2.10.2 Time history of movable loads

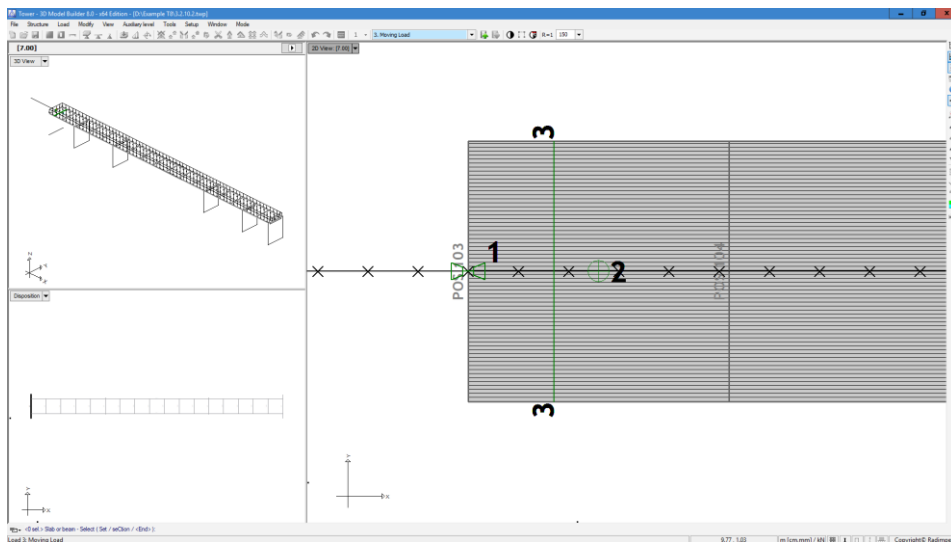
<0 sel.> Sab or beam - Select (Set / seCtion / <End>):

**Section** - the history of the impact of the mobile load can also be calculated in the set sections on the slab. By selecting the sub-option "seCtion" from the command line, enter the section procedure by selecting 2 points from the drawing.

First Point (<End>):

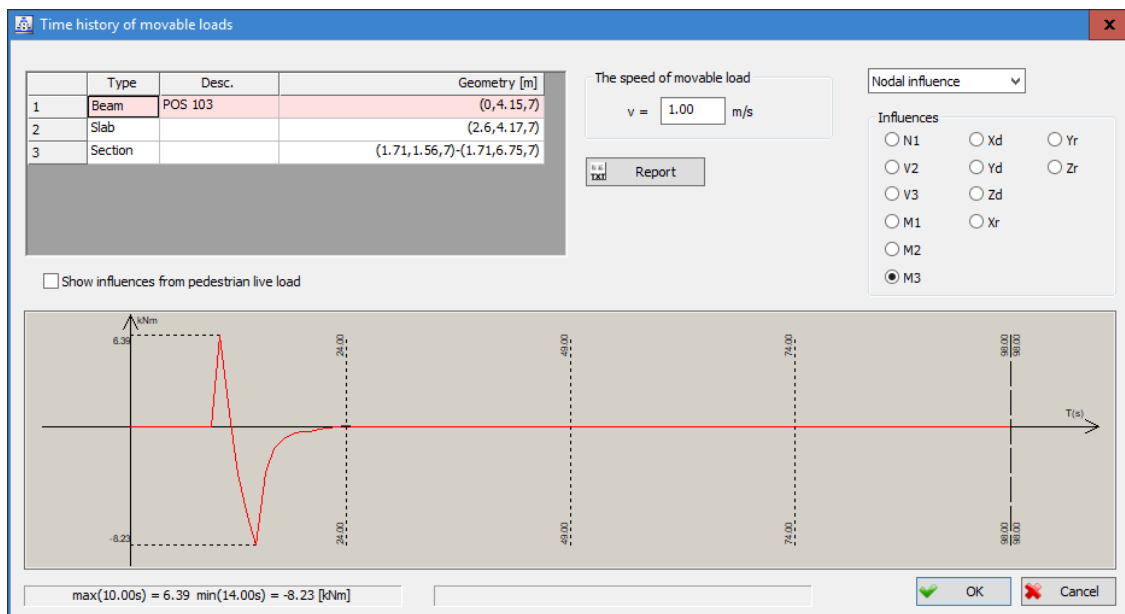
Second Point (<End>):

After setting the second point, the line program sets a section and returns to the basic command line shape.

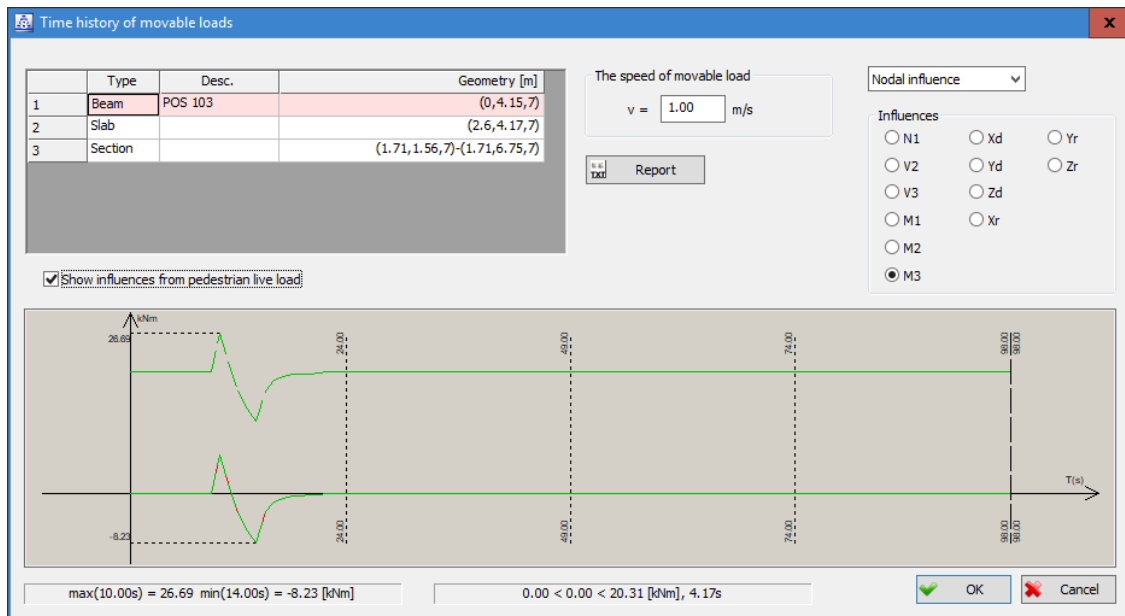


The set section is marked by the line which joins the chosen points

Selecting this command from the "Influences" drop-down menu, within the calculation result module, a dialog for displaying the history of the influence of the mobile load in the selected entities opens:

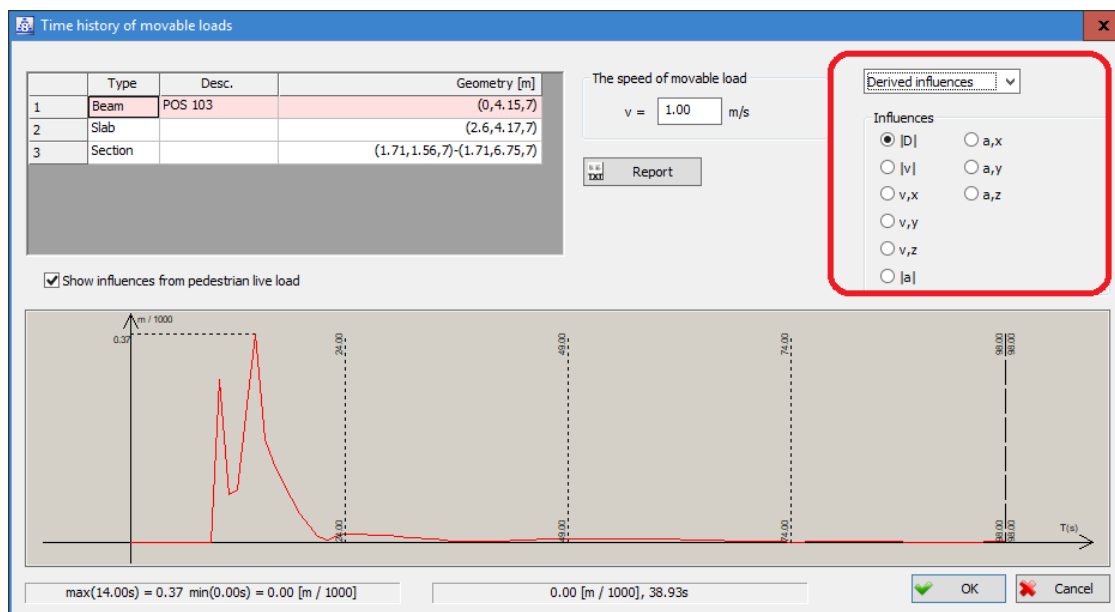


**Show influences from pedestrian live load** - a check box that appears in the dialog only if there is still some load in the selected load case in addition to the moving one. This case most often occurs when, in addition to the load from the vehicle, which is assigned as a moving one, the pedestrian infiltration, which is assigned as surface or line load, is taken into account. When the check box is turned off, only the history of the impact from the moving load is displayed, and when it is switched on the influences of the pedestrian infiltration are also displayed.



The layout of the history of diagram of pedestrian infiltration

**Influences** - In the dialog, two types of influences can be displayed: **"Nodal influence"** and **"Derived influences"**. The choice of the type of influence is done from a closed list, which is above the part of the dialog with the names of the available influence. After each change in the type of impact, the influences that are available in this part of the dialog are also changed.

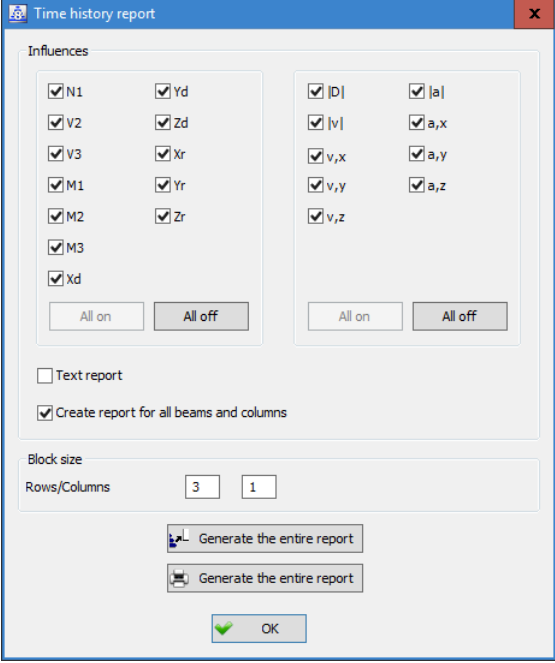


The calculated influences at the set point of the beam are selected from the list

- |D|** - absolute shift (the root of the sum of the squares of the component displacement)
- |v|** - absolute speed
- v,x** - speed towards the global X axis

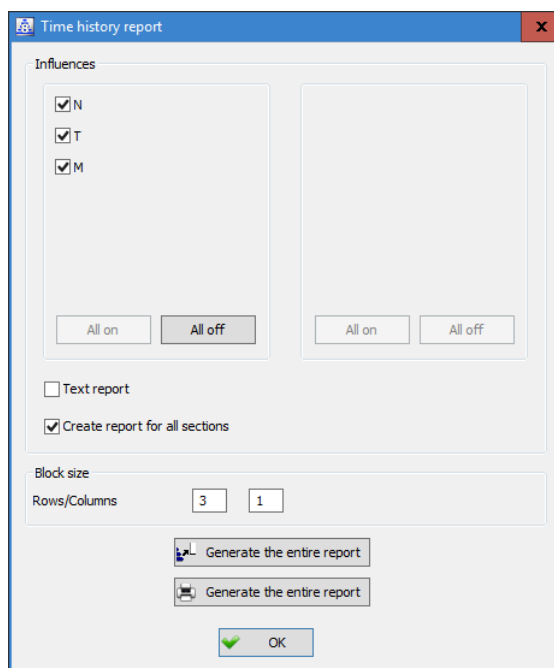
- v,y** - speed towards the global Y axis
- v,z** - speed towards the global Z axis
- |a|** - absolute acceleration
- a,x** - acceleration towards the global X axis
- a,y** - acceleration towards the global Y axis
- a,z** - acceleration towards the global Z axis

## Report



**Text report** – if this check box is set ON, the program will, in addition to the graphic blocks, generate tables with values of influence at the certain moments of time for the selected influences.

The mode of operation with this command is exactly the same, and when the table is selected, the only difference is in the type of influence that can be found in the report:



The layout of dialog for generating the history report of influences in the sections

### 3.2.15 Turning walls into load



By choosing the command "Turning walls into load", from the "Load" drop-down menu, you enter the procedure for selecting the walls that will be converted into a load:

<0 sel.> Walls - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End>):

After selecting the desired walls, a new message appears on the command line:

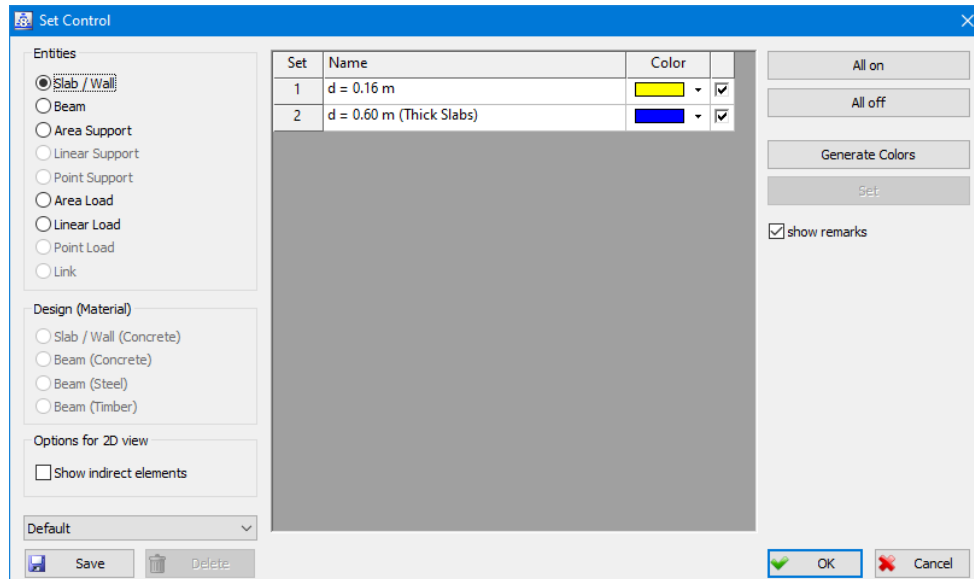
Delete selected walls (<No> / Yes):

By selecting the "Yes" option, the selected walls will be replaced with the appropriate line load, while by selecting the "No" option, the selected walls will be converted into a line load without deletion, that is, both the walls and the replacement line load will remain in the model.

## 3.3 Changing and Controlling the Previously Entered Data

### 3.3.4 Set Control

A particularly important option in the program is the visual perception of the accuracy of the entered numerical data, ie the possibility to visually check whether all elements of the construction are associated with accurate sets of numerical data. The "**Set Control**" command has been developed for this purpose. It can be called from the drop-down menu that opens by right-clicking on the name of the window "3D View", as well as from the drop-down menu "Layout".



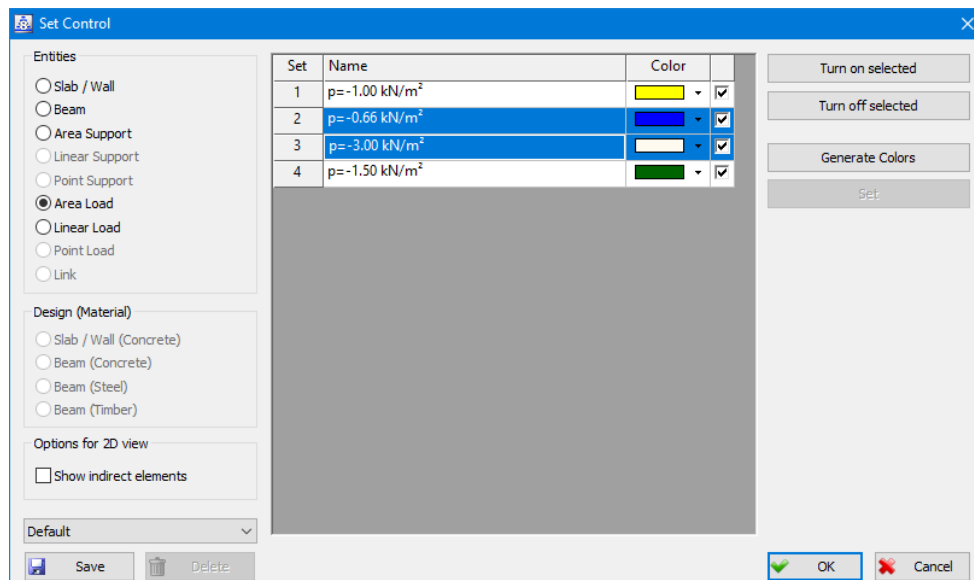
Appearance of the dialog within the command "Set control"

### show remarks

The program allows you to set remarks for sets of plates, beams, surface, line and point supports and connections. If this check box is set to the on state, the default note will be displayed in the dialog and in the set control legend.

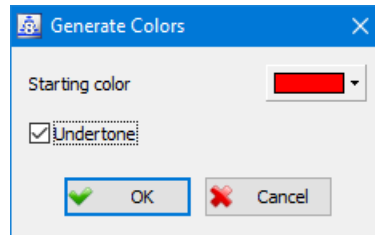
### Generate a color palette

Multiple selection is enabled in the set control dialog, so that only for selected entity sets colors will be generated. The selection is done as in all other commands, using the Shift and Ctrl keys on the keyboard and the left mouse button.

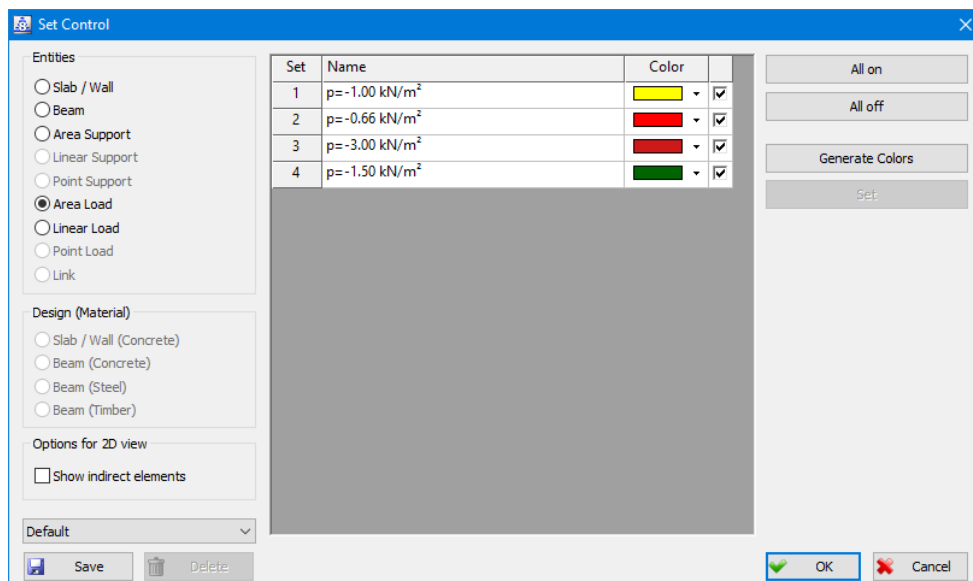


Multiple selection of sets in the set control dialog

Activating the "**Generate Colors**" button opens the following dialog:



After activating the command field "OK", the program will return to the basic appearance of the dialog, and new colors will be generated only for selected entries from the list from the specified palette.



The selected sets are associated with different shades of the color chosen for the initial one

If one or no sets are selected in the list, using the "All on" and "All off" command fields, you have the option to turn all sets of a given entity on or off to display. If multiple sets of sets are made in the dialog, then the command fields become "Turn on selected" and "Turn off selected" and you have the option to turn on or off to display only selected sets of a given entity.

After selecting the command field "OK", this dialog will be closed, and the selected sets of the given entity will be painted with the appropriate colors on the screen.

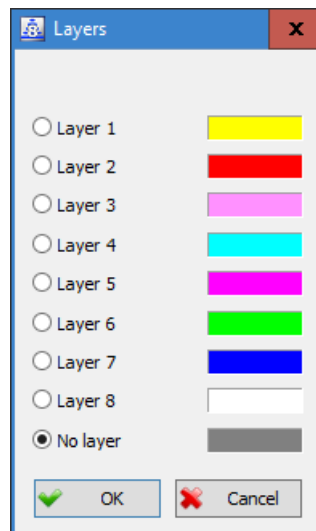
The "Design (Material)" part of dialogue box contains switches that allow entities to be sorted and displayed according to the design material used. These switches are unavailable in the data entry module, so they will be discussed in more detail in the chapter "8.23 Set Control".

## 3.4 Commands for Entering the Elements of Drawings other than Parts of the Structure

### 3.4.1 Auxiliary Axes

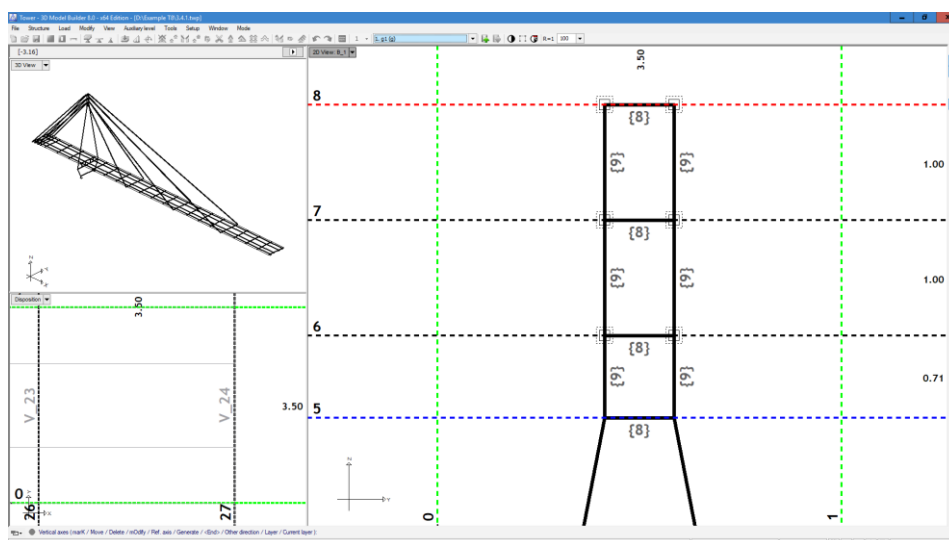
#### 3.4.1.1 Defining the Axes

Regardless of the layout method, the axes are shown in the drawing in the colour of the layer that was liquid at the time of placement. At the beginning of the command line there is a circle symbol which colour indicates the selected fluid layer, and its change can be made using the **"Layer"** command line:



The layout of dialog for selecting the current layer

When **"No layer"** is selected, the axes are not set on any layers, but are drawn in colour assigned to them in the "Settings" dialog. Of course, the visibility of the lens will not affect the visibility of this set of axes.



The horizontal axes 5 and 8, as well as the two vertical axes, are drawn with the colour of the layer to which they are assigned, while the horizontal axes 6 and 7 are not placed on one layer

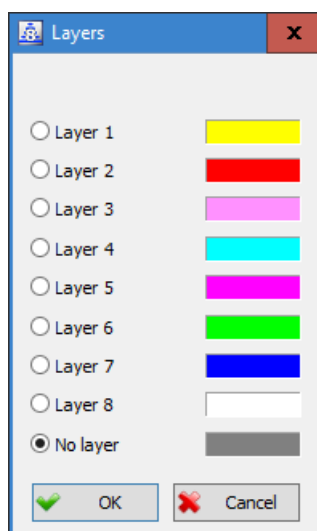
This dialogue can also be used to lock the layers, preventing accidental deletion or changing of the auxiliary axes, which is described in detail in Chapter 3.4.2.

### 3.4.1.7 Current layer

The affiliation of the axis to the layer can also be changed after installation. For this purpose, the "**Current layer**" option is provided, which is located on the command line while the command for setting the auxiliary axis is started. Also, the axes in which the "No layer" option was active when setup, can join a layer. Selecting this option on the command line a message is displays asking the program to select all the auxiliary axes you want to change their layers.

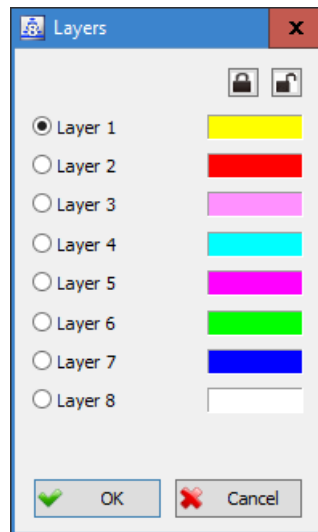
<0 sel.> Change of layer for vertical axes (<End>):

We will note that only currently active axes (vertical or horizontal) can be selected, as can be seen from the text on the command line. After selecting the desired axes, clicking on the right mouse button or choosing the option "End" from the command line, the dialog box appears as follows:



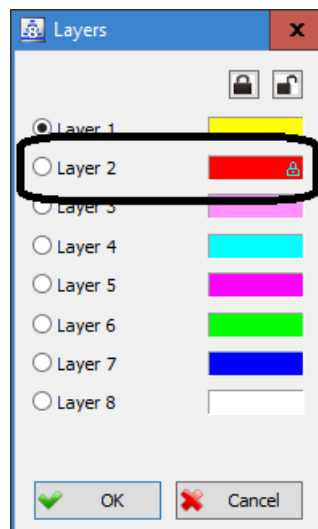
Now it is necessary to select a layer by clicking the mouse on one button where the selected axes will be associated after the exit from the dialog. In addition to one of the offered layers, the option "No layer" can be selected.

### 3.4.2 Selecting the Layer for Drawing



The layout of the dialog for selecting current layer

In this dialog, locking of layer can be done, thus preventing accidental deletion or change of entities that belong to them (auxiliary lines, texts, angles). Locking the layer does not apply to auxiliary axes, because they have a different way of selecting, changing, and deleting from other auxiliary entities. Single locking/unlocking of the layers is done by clicking the mouse on the rectangle with the colour. If the layer is locked, in the right side of the rectangle the lock pad icon will be displayed.



Clicking the mouse over rectangles with colour  
Layer 2 is locked

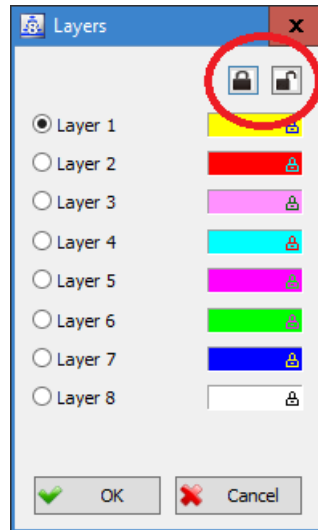
This dialog also allows you to lock/unlock all the layers simultaneously. For this purpose, buttons are provided that are above the part of the dialog with the colour of the layers:




clicking the mouse over the button with the closed padlock icon, all layers are locked.



clicking the mouse over the button with the open padlock icon, unlocking all locked layers is done.



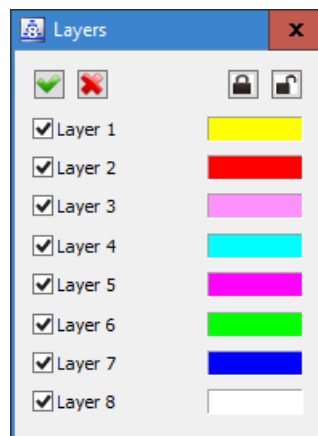
Clicking mouse over the button   
all layers are lock

We will note that beside the command for setting the auxiliary entitets, locking the layers is possible to provide by the adjusting layout command

### 3.4.6 Layers visibility



Using the "**Layers visibility**" command, the visibility of the layers can be turned off and the locking of unlocked or unlocked layers can be locked. By selecting the command from the drop-down menu "Auxiliary level" or by left-clicking on the icon above, the icon will open the following dialog box:



In front of the name of all the layers there are check boxes for adjusting their visibility. The switched on check box indicates that the set layer is visible and vice versa, the excluded check box indicates that the layer is invisible. The visibility of the layers can be changed individually - by left-clicking the mouse over its check box and at the same time for all the layers. Turning on/off the visibility of all the layers is done using the buttons located above the name of the layer:



The left click of the mouse over this button turns on the visibility of all layers.



The left click of the mouse over this button turns off the visibility of all layers.

In this dialog, the locking/unlocking of the layers can be done in exactly the same way as in the dialog that opens by choosing the "Layer" option from the command line, while commands for setting up auxiliary lines, angles and texts are started (see section 3.4.2).

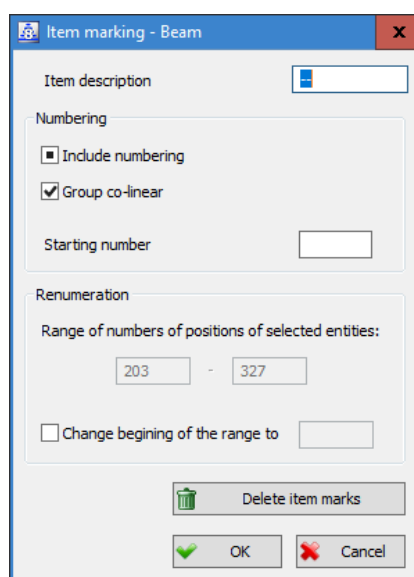
Note that any command can be started while the "Layers visibility" command is active, which means that its display on the screen does not affect the operation of the program.

## 3.4.7 Item mark

### 3.4.7.1 Item marking

#### Renumeration

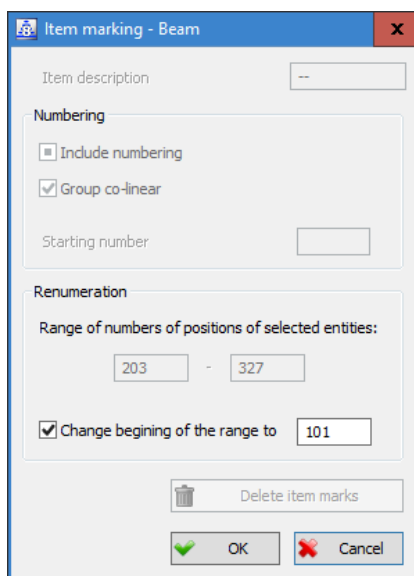
With this command, you can also perform the renumeration of position numbers to entities whose position mark has been previously assigned and that have a number in the position mark. This means that the check box "Change beginning of the range to" would be available, among the selected entities there must be at least one that satisfied the specified conditions.



Entities which satisfied the condition for renumeration are selected

**Range of numbers of positions of selected entities** - Fields showing the start and end of the range to which the numbers of the positions of the selected entities are assigned. The data in these fields can not be changed, they are informational.

**Change beginning of the range to** - When this check box is set to on, the change status becomes available and edit field in its extension. In this edit field is set a new ordinal number with the position mark which has the smallest ordinal number. On the basis of it, the ordinal number will be changed by all the other position marks of the selected entities, with the same relationship between the new and the old number. We will note that other data, such as position description, will not be changed. Also, the command will not affect selected entities which position mark does not have an ordinal number, nor those that do not have a position mark.

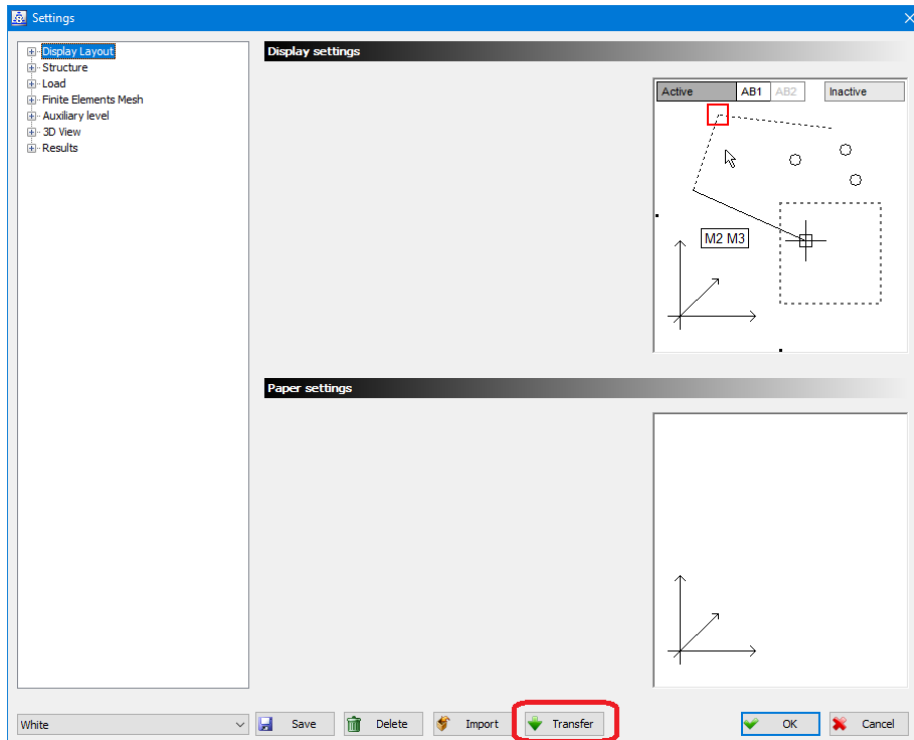


It is assumed that the new ordinal numbers of the positions of the selected entities begin from 101 and end with 225, i.e. each selected entity if the regular position number is reduced by 102 ( $203-101 = 102$ )

## 3.5 Adjusting the Parameters Used by the Program in its Operating ("Setup" menu)

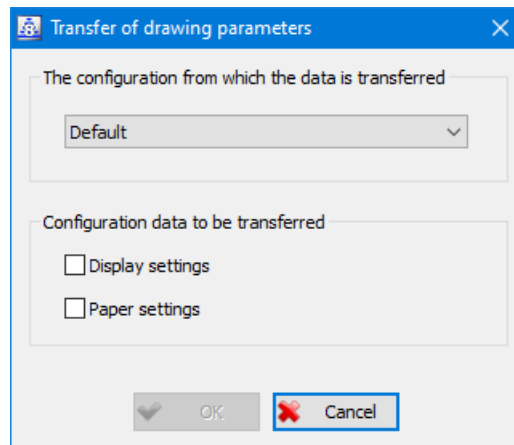
### 3.5.1 Settings

- A new configuration of drawing parameters – "Standard" was made and included in the installation.
- Several new configurations of drawing parameters for color printing have been created and included in the installation: "Default-Colour Print", "Black-Colour Print" and "Standard-Colour Print".



Button that allows you to copy the display parameters from the saved configuration to the current configuration

**Transfer** The program allows you to copy the display parameters from a saved configuration to the current configuration, whereby the screen parameters and the paper parameters can be transferred independently. Selecting this button opens a dialogue of the following appearance:



### The configuration from which the data is transferred

List of all saved configurations from which display parameters can be copied. Using the two check boxes "Display settings" and "Paper settings", the user determines which parameters will be transferred from the selected configuration to the current one.

### Display settings

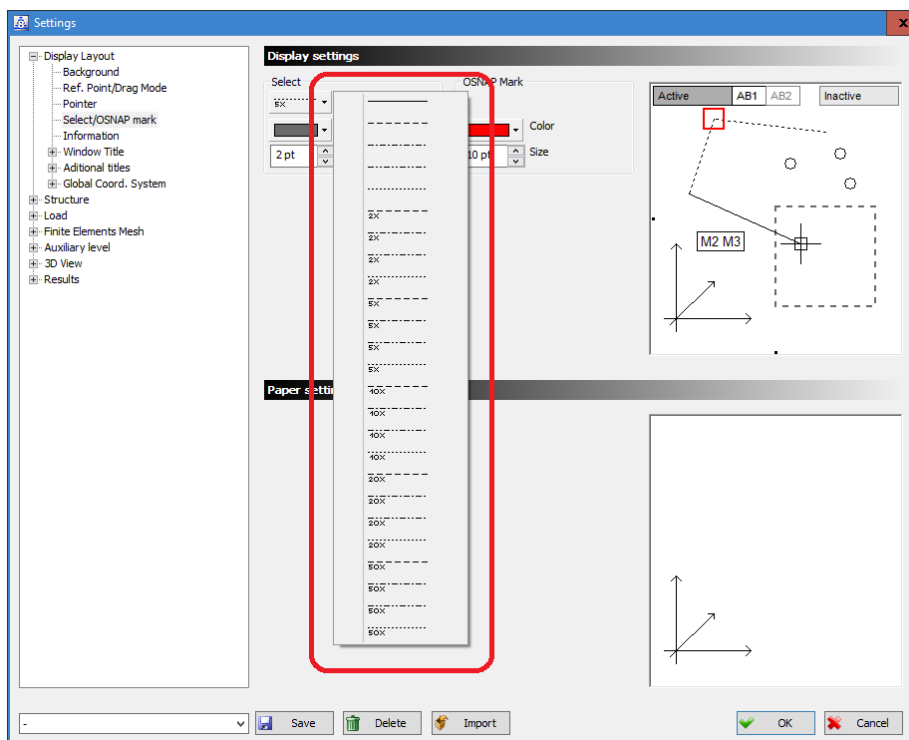
By setting the given check box to on, the display settings will be copied from the selected configuration to the current configuration.

## Paper settings

By setting the given check box to on, the paper settings will be copied from the selected configuration to the current configuration.

## Appearance of the screen

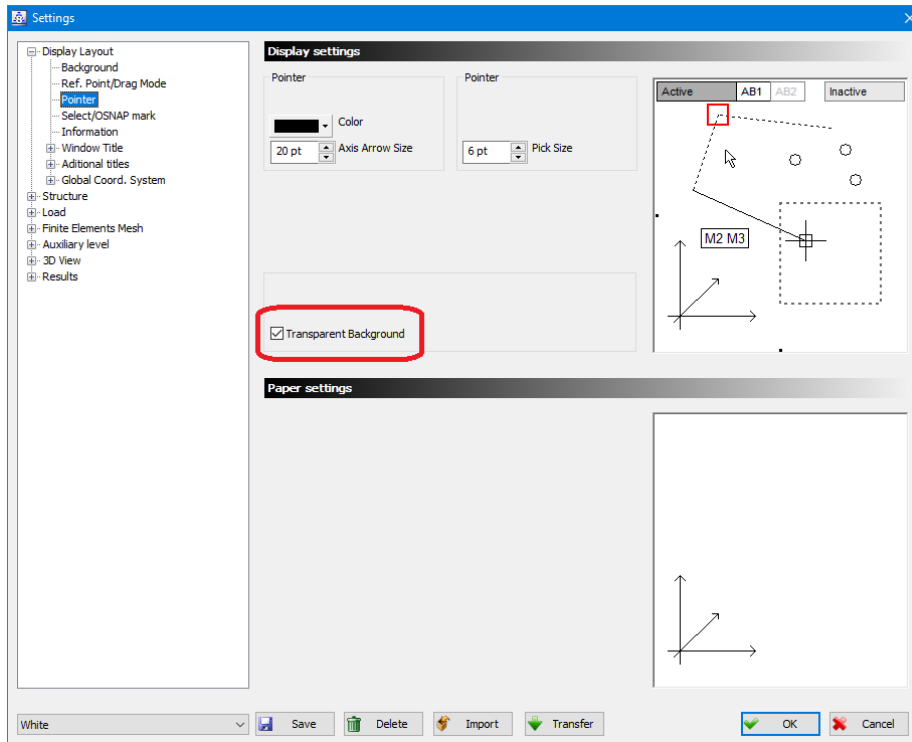
In earlier versions of the program, when printing discontinuous lines, the line type was not clearly visible on paper, due to the unadjusted size of the dashes and the spaces from which the line was drawn up. In order to avoid this, all types of interrupted lines with dashes and spaces of 10, 20 or 50 times bigger than the standard ones are enabled. These lines are in the lists for the type selection marked with the text "10x", "20x" and "50x".



Drop-down menu for selecting the type of the line

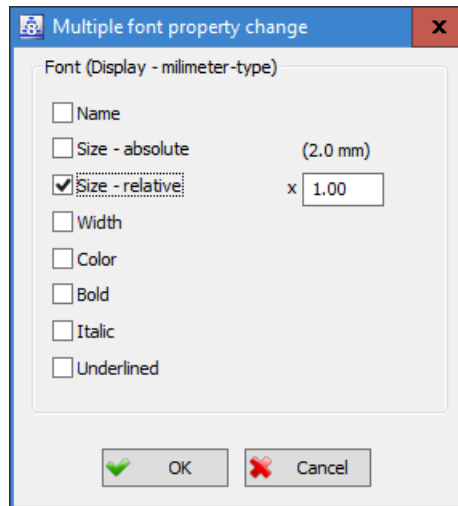
## Transparent Background

Check box that determines whether the out of domain cursor (arrow) will have a transparent background or will be drawn filled for enhanced visibility.



“Transparent Background” check box

### Quick change of font parameters



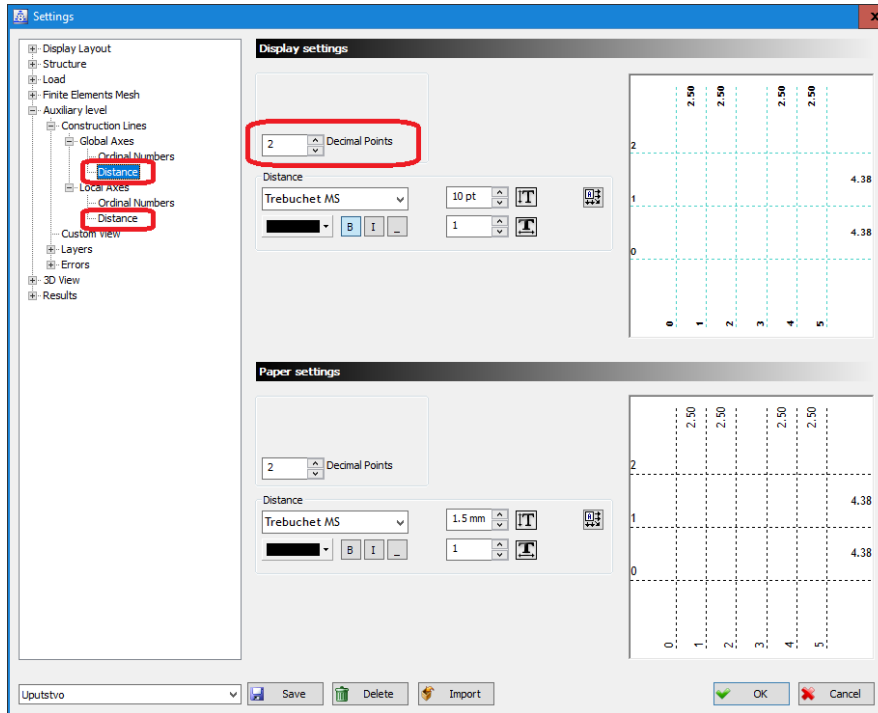
**Size - absolute** - when this check is turned on, all other texts are assigned the default font size in millimeters or pixels.

**Size - relative** - when this check box is enabled, all other texts are assigned the default font size in millimeters or pixels multiplied by the value entered in the edit field.

These two check boxes can not be set ON status at the same time, since they both are to change the font size.

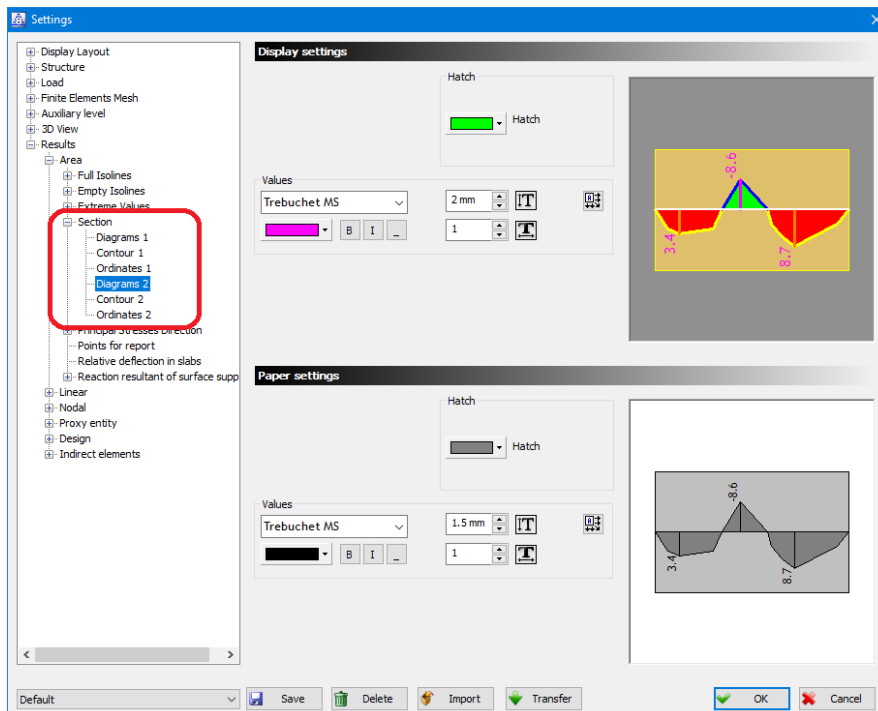
## Auxiliary level - Auxiliary axes

For both types of auxiliary axes, "Global axes" and "Local axes", in the branch of the "Distance" trunk in the edit field "**Decimal Points**", you can specify the number of decimals with which to print the distance between the axes on the screen as well as the number of decimals with which will print the spacing between the axes on paper when exporting the blocks.



Edit field to enter the number of decimal places of spaces between global onscreen auxiliaries

## Section



Parameters for defining the appearance of influence diagrams in arbitrary sections

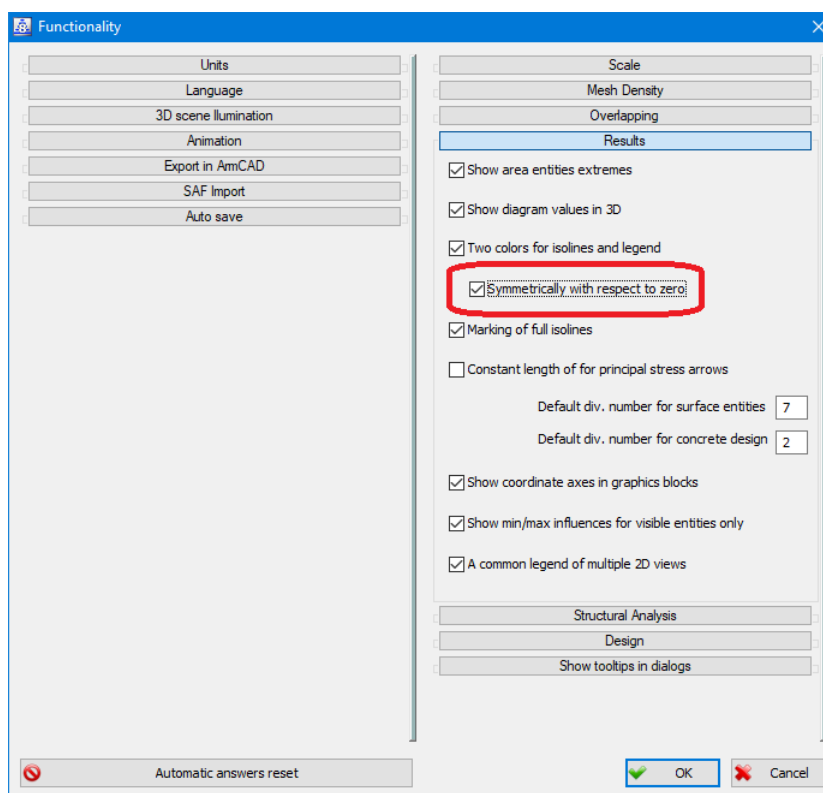
It is possible to set different parameters for one side of the diagram and different parameters for the other side of the diagram to draw the influence diagram in arbitrary sections of the plate. Now, in the "Section" branch of the dialog, there are special branches for defining how to draw one side of the diagram, and special branches for drawing the other side of the diagram.

## 3.5.2 Functionality

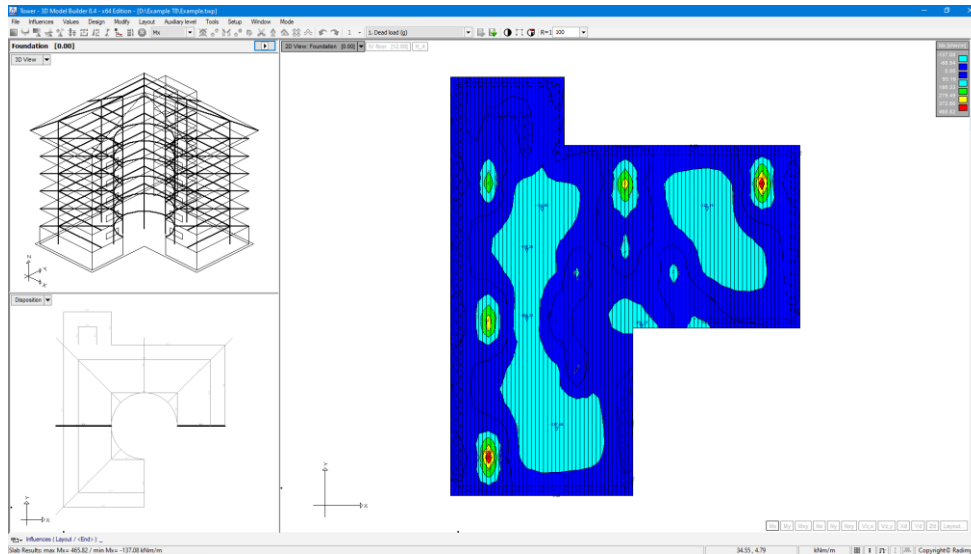
### Results

**Show area entities extremes** - this check box defines whether influence values and corresponding symbols will be shown when displaying influences in surface entities, at the points where the extremes are located. For the same purpose, the command "Show area entities extremes" is provided, which is called from the drop-down menu "Values" (chapter "8.27").

**Show diagram values in 3D** -this check box is used to define whether numerical values will be shown in the "3D view" window on the diagrams of the displayed influences, in both line and surface entities. For the same purpose, the command "Show diagram values in 3D" is provided, which is called from the drop-down menu "Values" (chapter "8.28").



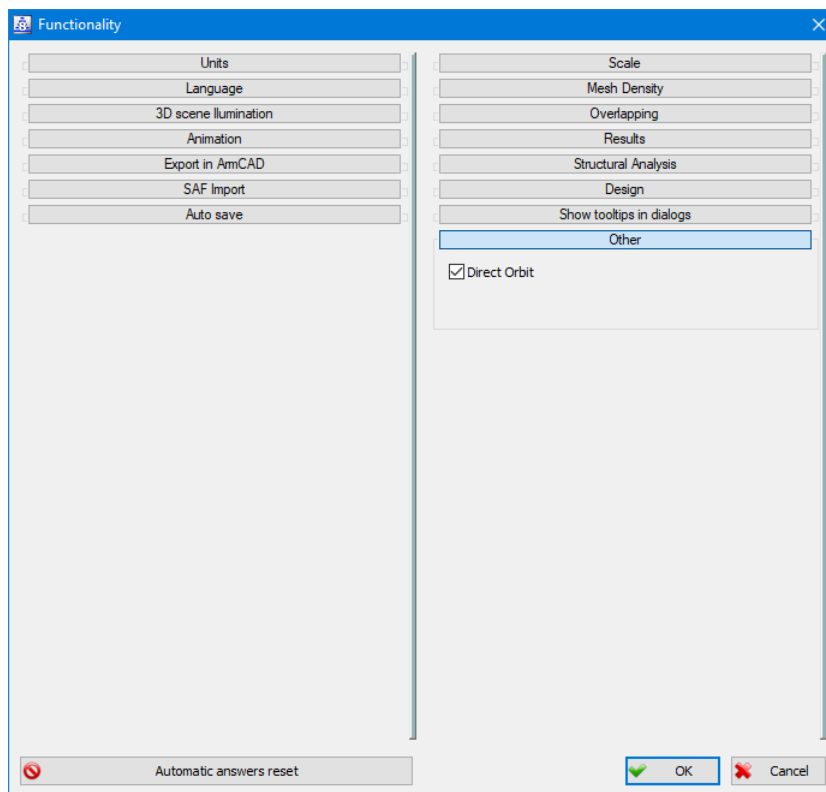
**Symmetrically with respect to zero** - using this check box may require the display of full isolines of surface entities, so that the same color refers to the same range of both positive and negative values. The check box is only active if the "**Two colors for isolines and legend**" check box is on.



Influences in the slab shown with the check box status on  
 "Symmetrically with respect to zero "

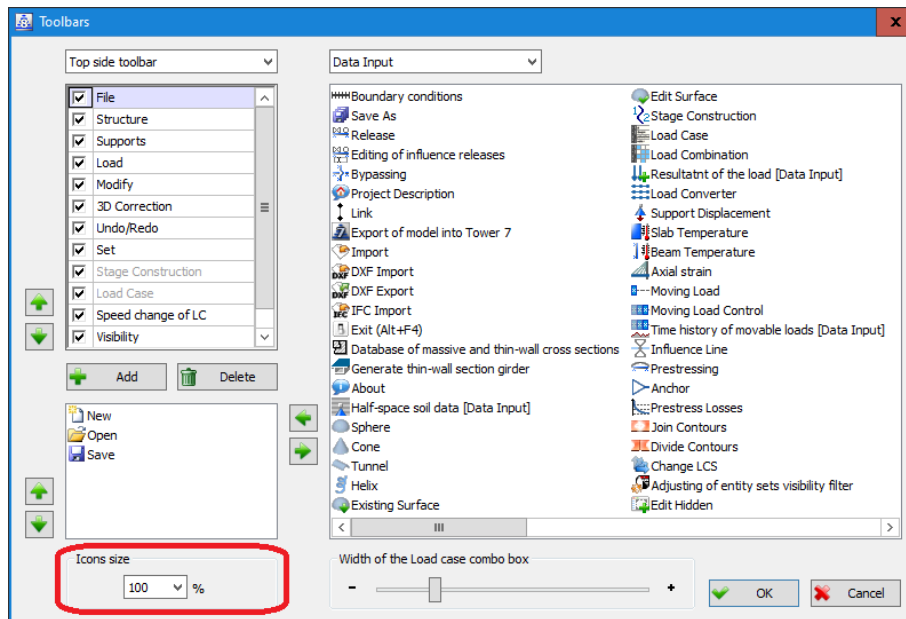
The colors to be used to display the values are selected in the "Parameters" command dialog, in the "Results ► Surface ► Full isolines" branch (see chapter "3.5.1").

### Other

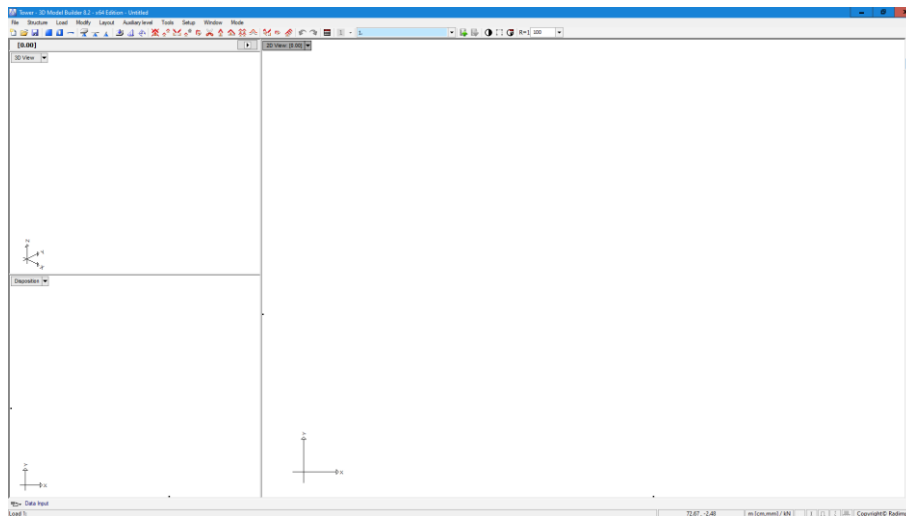


**Direct Orbit** - When this check box is set to the checked state, the "Orbit" command (chapter 2.13.1) switches to direct operation mode, which means that it is practically constantly active.

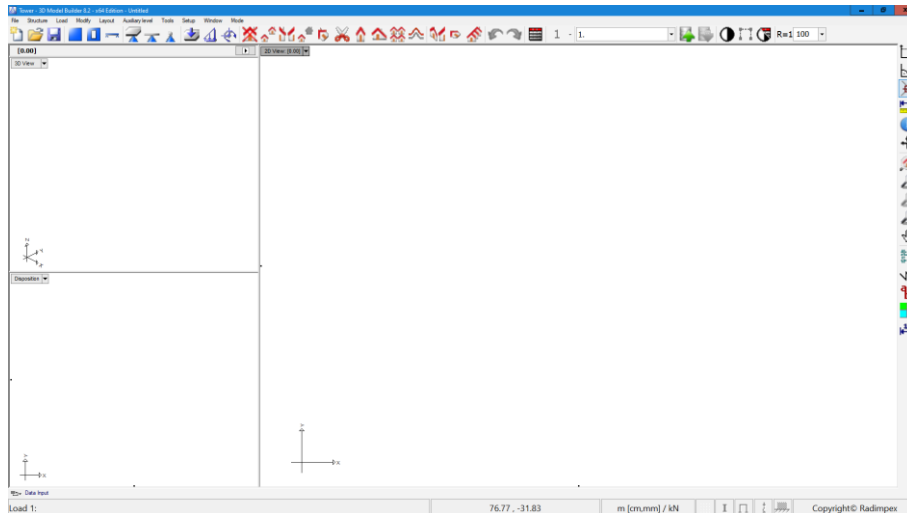
### 3.5.3 Toolbars



**Icons size** - A closed list used to increase the size of the icons and the status bar of the program so that all the data on it is visible on high resolution monitors. Three values can be selected, 100%, 150% and 200%. After the value changes, the program will issue a notification that it is necessary to restart it in order for the new size to be applied.



The default icon size is 100%



Icon size resized to 200%

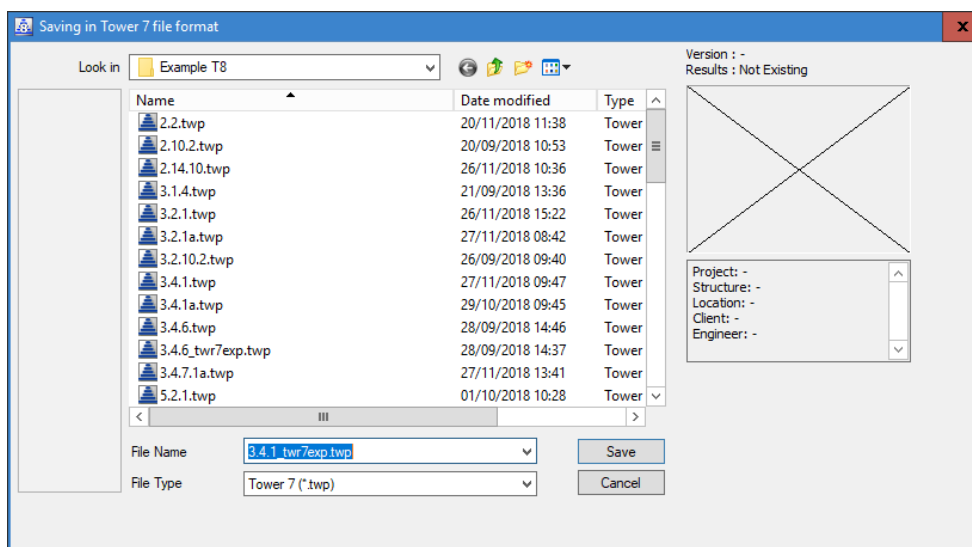
Load Case Selection Field Width - A slider that adjusts the closed list width to select the current load case.

## 3.6 Commands for Operating with Data Files

### 3.6.5 Export of model into Tower 7

With this command, the current state of the model can be recorded in a new file that has the format of Tower 7. It is important to note that only a file with the extension “\*.twp” is recorded. It stores data of construction and load, more precisely data that is created in the “Data Input” module. In addition to the structure itself, a finite element network is also stored if it is generated. This practically means that the results and project documentation can not be transmitted in Tower 7, but only the input model.

By choosing the “**Export of model into Tower 7**” command from the drop-down menu of the “File” menu, the following dialog opens:

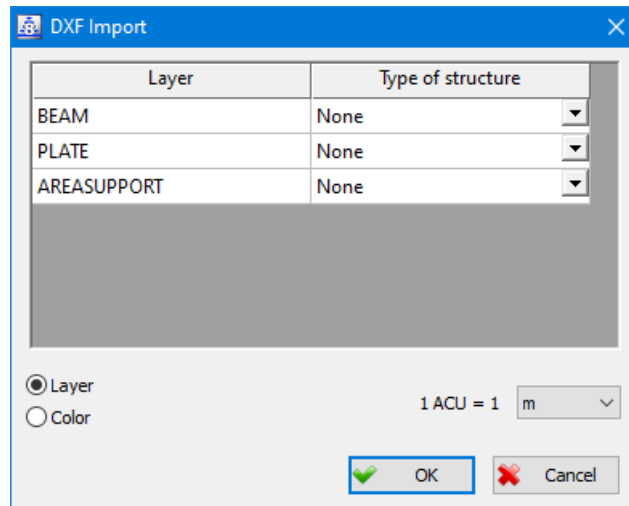


**File Type** - a field that shows that the model is exported to the Tower 7 file format.

**File Name** – edit field in which the name of the new file is assigned where the current state of the input file of the model will be recorded. The program automatically creates the name of the new file by adding the “\_twr7exp” extension to the current file name and offering it to this field. Of course, a completely arbitrary name can also be entered in the edit field.

**Save** - clicking this button, the dialog is closed and a completely new file with the given name is created.

### 3.6.7 Importing the Drawings from AutoCAD (DXF Import)

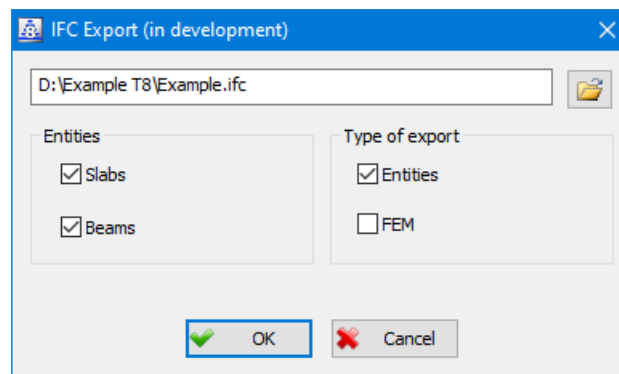


There is a closed list for picking a measurement units used when importing drawings from DXF format. It defines the unit of measure for length in the Tower which corresponds to one AutoCAD unit (ACU) from the DXF drawing.


### 3.6.10 IFC Export

Using the “**IFC Export**” command, the geometry of the model created in Tower can be exported to IFC format. Also, a finite element mesh generated for a given model in the Tower program can be exported. This command is still in the development phase, so perfect results cannot be expected in all situations.

Selecting the command from the drop-down menu “File ► Export” opens the following dialogue:



IFC Export Settings Dialogue

The edit box specifies the file name as well as its position on the disk. This information can also be specified using the standard "Windows" file capture dialog box, which is opened by activating the command field "  ".

### Entities

In this part of the dialogue, the entities are selected to be reported in IFC format. Only beams, only slabs or both beams and slabs can be exported.

**Slabs** - when this check box is turned on in IFC format all beams from the current model will be exported.

**Beams** - when this check box is turned on in IFC format all beams from the current model will be exported.

Structural elements (slabs, walls, beams and columns) exported in IFC format are associated with data related to volumes, surfaces, materials and cross-sections.

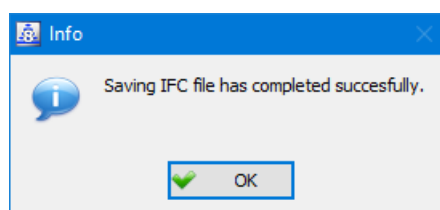
### Type of export

Part of the dialog in which it is chosen whether to export entities (slabs, beams) or to export a finite element mesh. The check boxes shown work as switches, so only one of them can be set to the on state.

**Entities** - when this check box is included in the IFC format, the entities (slabs, beams) selected in the left part of the dialogue are exported.

**FEM** -when this check box is included in the IFC format the finite element mesh is exported. If the network is not generated a check box will be unavailable.

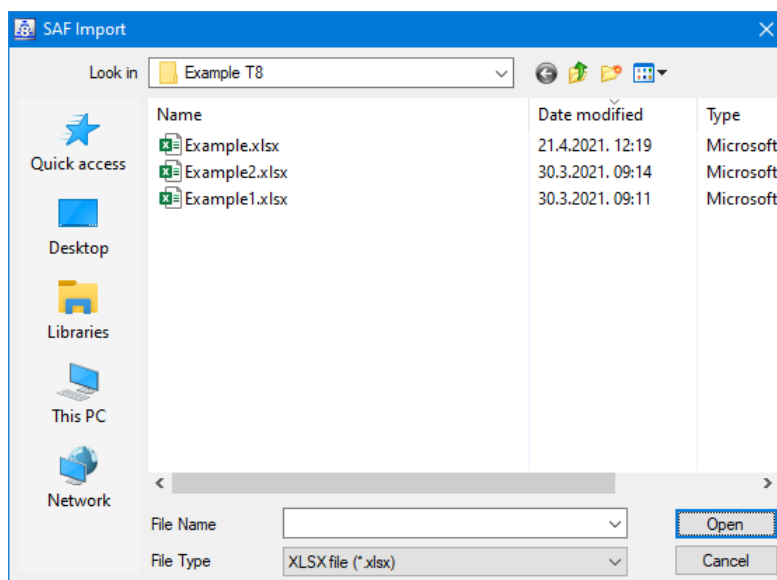
After exiting the dialogue, the program exports the selected entities and displays the corresponding message:



## 3.6.11 SAF Import

By importing and exporting models in SAF format (Structural Analysis Format), the program "Tower" allows communication with other programs that are used for the calculation of structures. SAF was created on the initiative of "Nemetschek Group" and currently SCIA, Graphisoft, Allplan, RISA, FRILO, Strusoft, AxisVM, Dlubal, Sofistik, SCAD, LIRA land, Radimpex and IDEA StatiCa participate in the initiative. Exporting a model in SAF format from one program, allows its import into another program, which easily exchanges models between different programs for the calculation of structures.

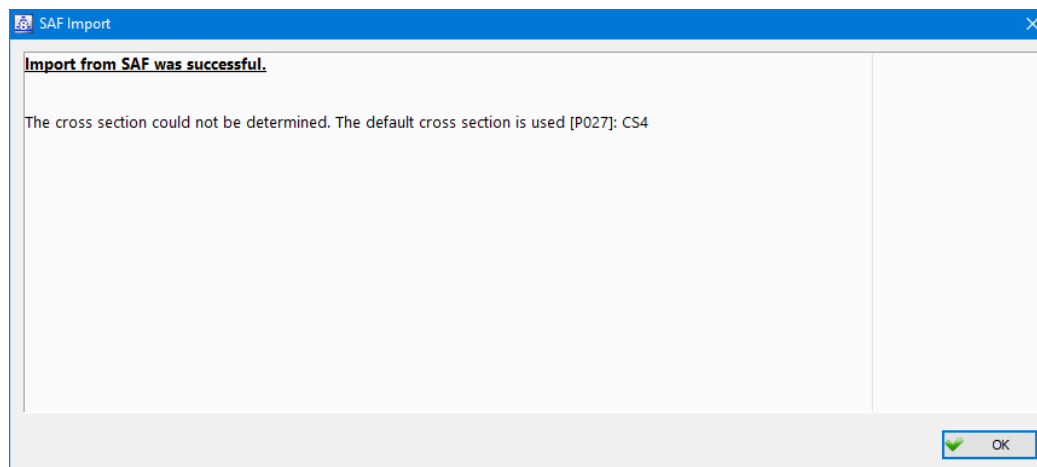
Selecting the "**SAF Import**" command, located in the "File ► Import" drop-down menu, opens a dialogue identical to the dialogue that opens when selecting the "Open" command (see chapter "3.6.2"):



Dialogue layout for importing SAF format models

The only difference is that only files with the extension “\* .xlsx” or “\* .xls” are now displayed in the dialogue.

Since SAF must have a minimum of capabilities from all programs that use it, and as modeling in one program may have some capabilities that other programs do not, by activating the “**Open**” button, the program will control all elements and loads in model and in the dialogue that then opens issue the appropriate notices:



Notification after SAF import

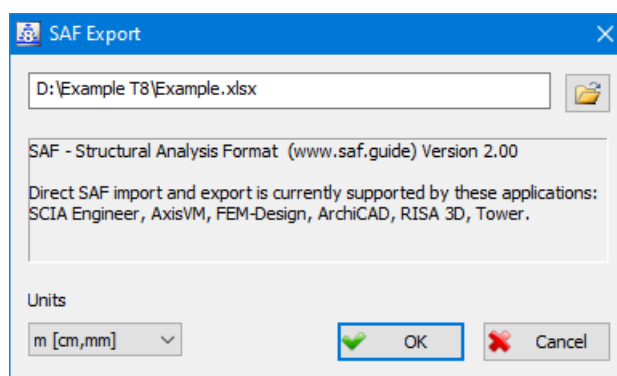
In this dialogue, the program will display notifications if some structural element or load from a model recorded in SAF format from another program could not be imported into the Tower program. At the end of each message line, the program prints an entity ID or label, which will be displayed in the note, so that the user can easily find the desired entity and modify it as needed.

As SAF is still in development, its improvement will reduce the number of possible differences between the programs that use it, and thus provide better mutual communication.


### 3.6.12 SAF Export

By importing and exporting models in SAF format (Structural Analysis Format), the program "Tower" allows communication with other programs that are used for the analysis of structures. SAF was created on the initiative of "Nemetschek Group" and currently SCIA, Graphisoft, Allplan, RISA, FRILO, Strusoft, AxisVM, Dlubal, Sofistik, SCAD, LIRA land, Radimpex and IDEA StatiCa participate in the initiative. Exporting a model in SAF format from one program, allows its import into another program, which easily exchanges models between different programs for the calculation of structures.

Selecting the "**SAF Export**" command, located in the "File ► Export" drop-down menu, opens the following dialogue:

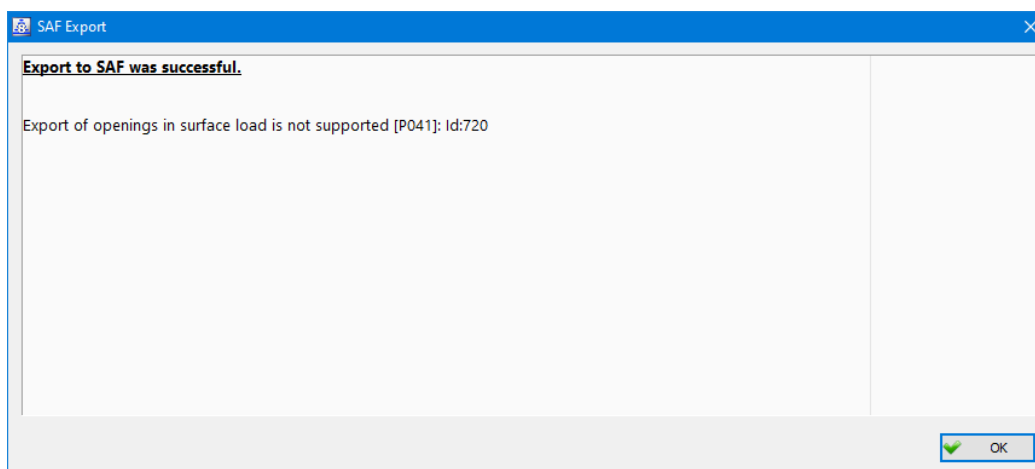


Dialogue for exporting models to SAF format

The edit field specifies the file name as well as its location on disk. This information can also be specified using the standard "Windows" file capture dialog, which is opened by activating the command field "  ".

From the closed list, the measurement units that will be used when exporting the model are selected.

Since SAF must have a minimum of capabilities from all programs that use it, and as modeling in one program may have some capabilities that other programs do not, by activating the "**OK**" button, the program will control all elements and loads in the model and in the dialog that then opens issue the appropriate notices:



Notice that SAF does not support the export of openings in surface loads

In this dialog, the program will display notifications if any structural element or load from the model created in the "Tower" program could not be saved in SAF.

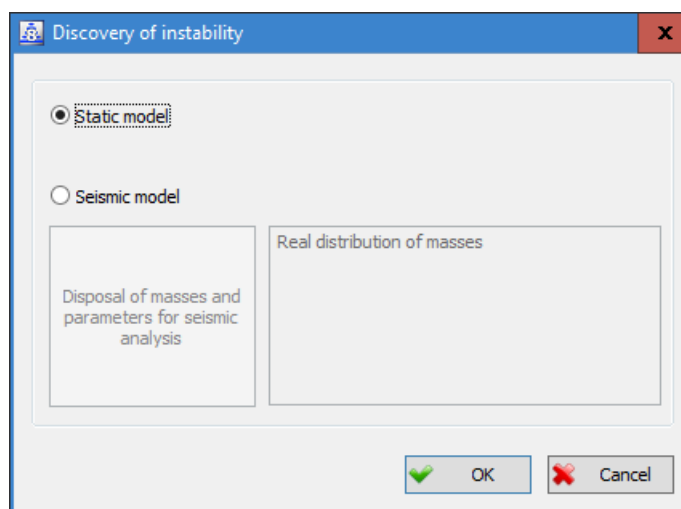
As SAF is still in development, its improvement will reduce the number of possible differences between the programs that use it, and thus provide better mutual communication.

# 5. Stability Design

## 5.2 Discovery of instability of the model

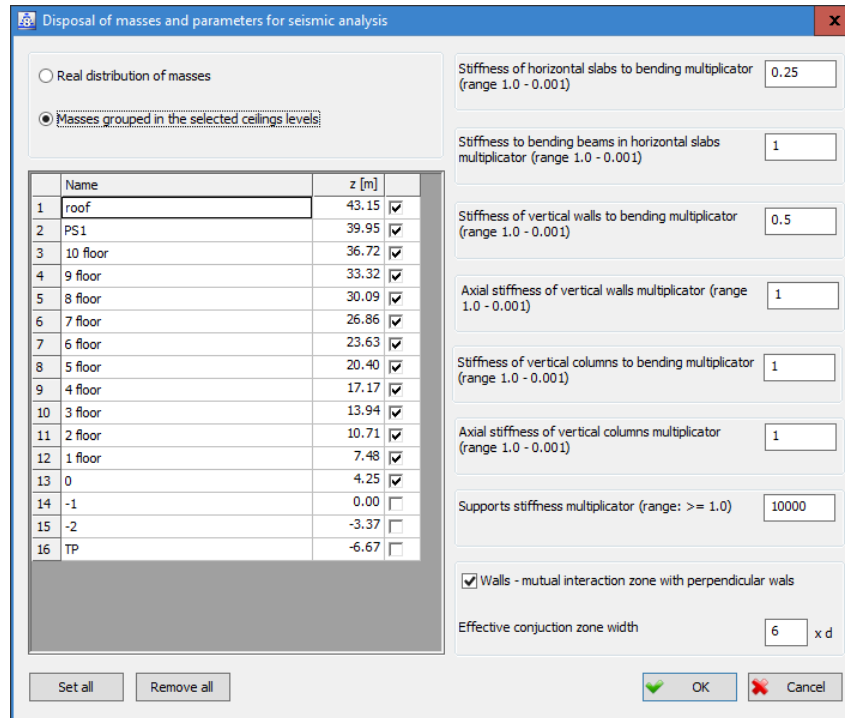
Using the “**Discovery of instability**” command, the analysis of the model with one load case is carried out by loading each node of the structure with the forces and moments of a small and randomly selected intensity and checking whether there are large deformations in it or is it labile.

Selecting this command from the pull-down menu “Stability” opens a dialog of the following layout:



**Static model** – if this switch is selected, the check will be performed on a static model.

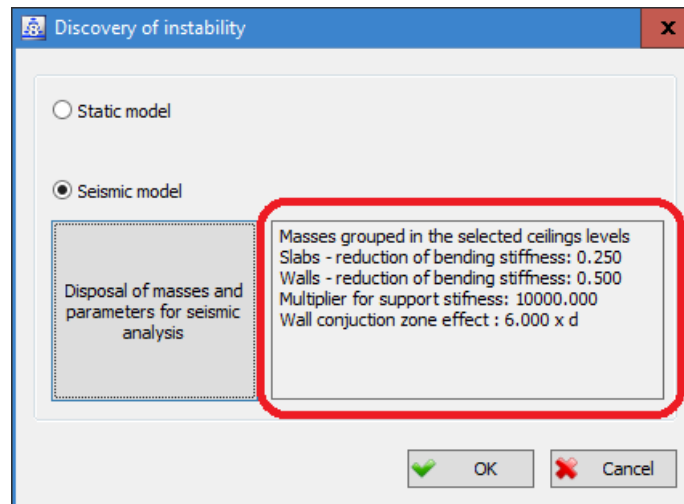
**Seismic model** - the selection of this switch indicates that the check will be performed on a seismic model. Also, with its selection, the “**Disposal of masses and parameters for seismic analysis**” button becomes available, by which the parameters for defining the seismic model can be changed. Clicking on this button opens the dialog box of the following layout:



The layout of the dialog where advanced options are defined seismic analysis

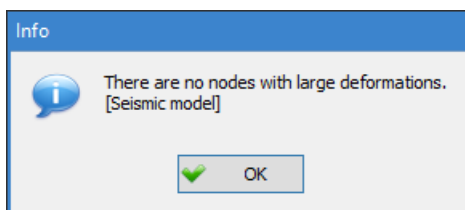
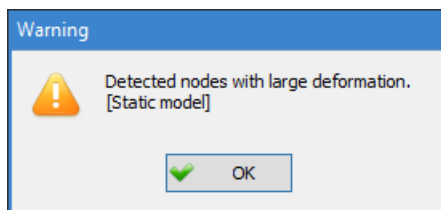
Operating with this dialog is described in detail in the command for calculating the modal analysis (see Chapter 6.1 for Tower 7), so we will not explain it here again.

All the seismic parameters analysis that are set in this dialog are displayed to the right of the button it calls.



Part of the dialog where the parameters are displayed seismic analysis

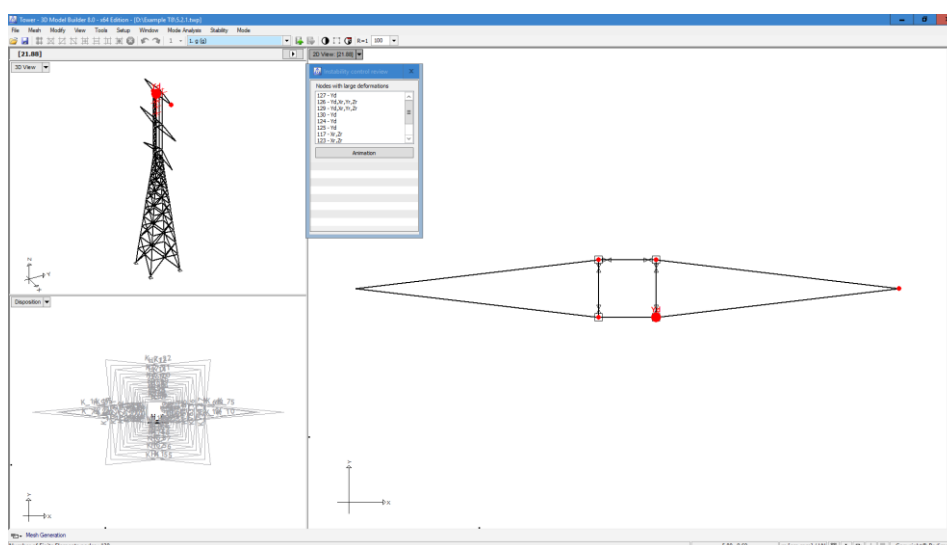
By clicking the "OK" button, the dialog box closes and the calculation of the instability calculation according to the given parameters is performed. Upon completion of the analysis, the program will display a notification whether it has found nodes with large deformations or not:



These messages also indicate the fact whether the analysis was done with a static or a seismic model.

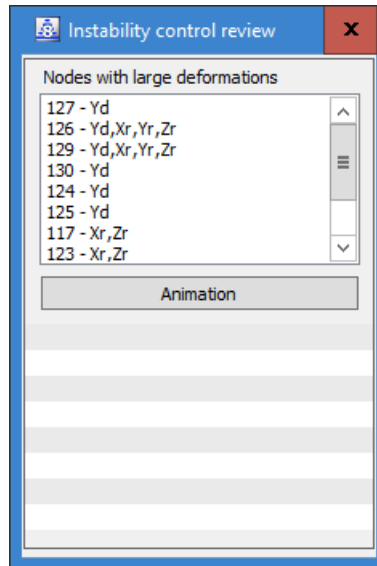
### 5.2.1 The review of instability of the model

A review of nodes with large deformations is done using the “**Instability control review**” command. By selecting it from the drop-down menu “Stability”, all nodes in which large deformations were found are marked with a red circle.

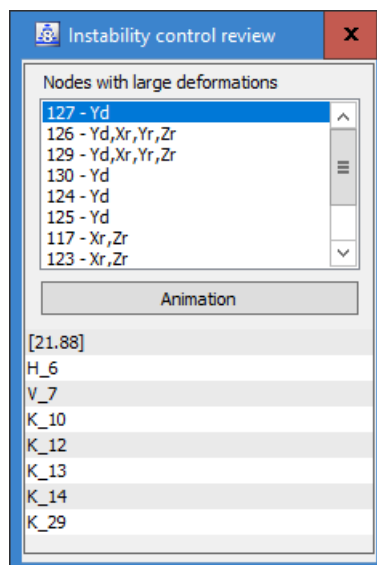


Marking nodes with large deformations in the model

At the same time the following dialog is opened:

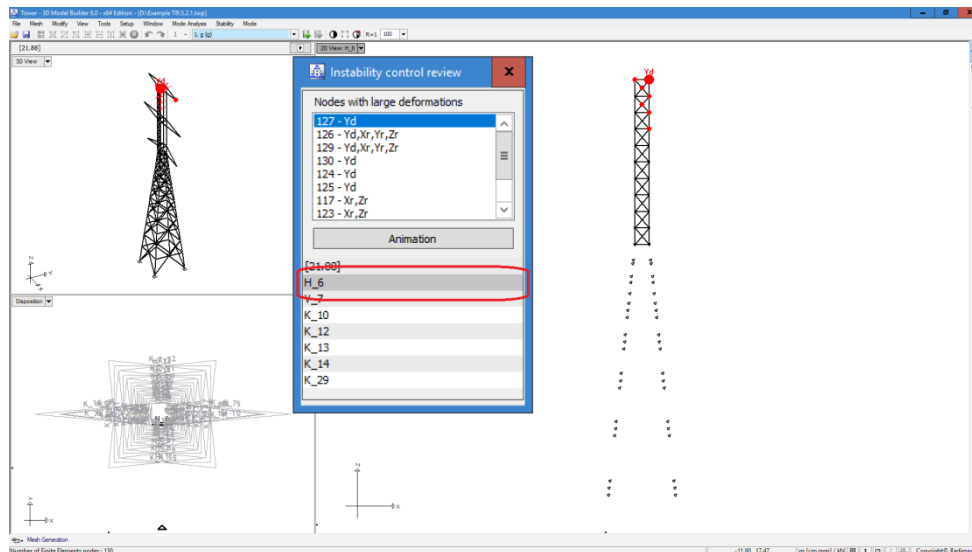


This dialog is modal, which means it will remain on the screen while other commands from the network generation module are started. It can be turned off by re-selecting the command or by clicking the "x" button. In the upper part of the dialog, the numbers of all nodes in which large deformations were found are shown. Behind each number, the signs of the found deformations ( $X_p$ ,  $Y_p$ ,  $Z_p$ ,  $X_r$ ,  $Y_r$  and  $Z_r$ ) are also printed. By clicking the mouse over one of them in the lower part of the dialog will appear a list with the names of all layouts in the model to which the selected node belongs.



In the lower part of dialog are shown all layouts to which the chosen node 127 belongs.

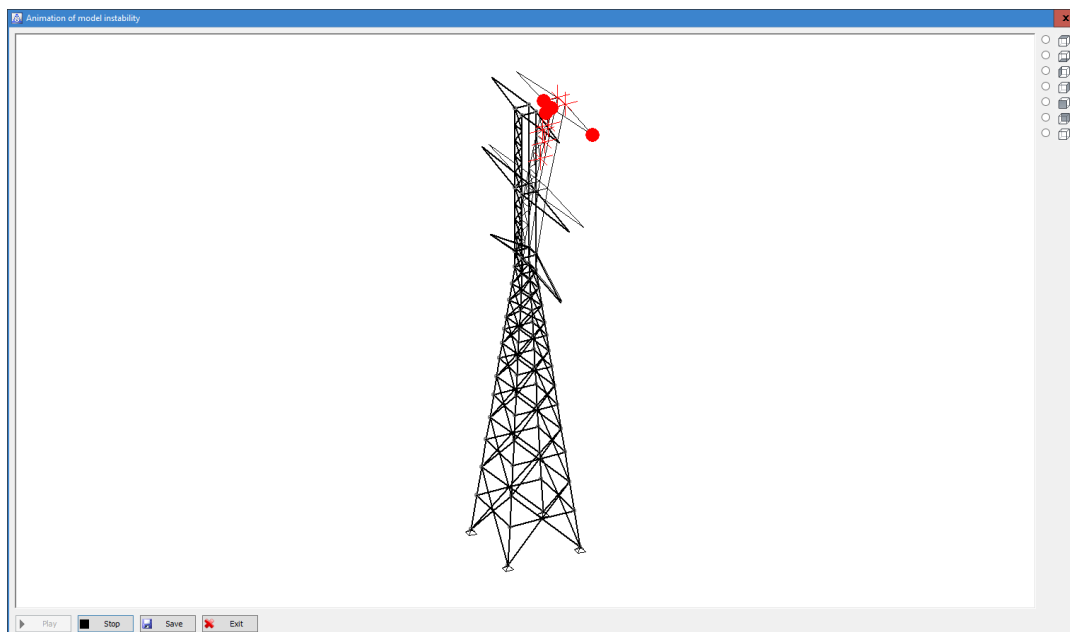
Clicking the mouse over the name of the eager look it will be shown in the 2D window.



Clicking the mouse over the name in the list, the frame H\_6 is selected for view in the 2D window

Note that the chosen node is layouted on the drawing with the symbol which is bigger than others.

**Animation** - selecting this command box opens a dialog where the animation of detected deformations is displayed in the model:



Dialog for animation layout

Operating with this dialog is exactly the same as with the animation dialog for the current tone of the modal analysis (see section 6.5).

The command for displaying large-deformed nodes can also be called from the data entry module by selecting from the "Tools" menu. In this way, very quickly in the model can find nodes with large deformations, in order to detect and remove the cause of their appearance. Note that when invoking the command in the data entry module, the button for displaying animation will be unavailable.

# 6. MODAL ANALYSIS

## 6.1 Calculation

### Masses grouped in the selected ceiling levels

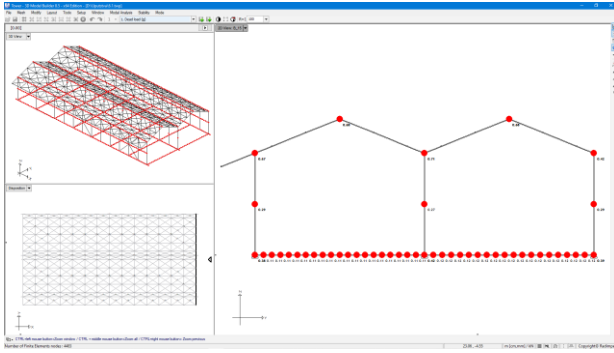
With this method of grouping, the program automatically assigns masses from vertical elements of the construction (walls, columns) to the selected ceilings. The program enables limiting the horizontal displacement of masses, so that it does not exceed the set value:

	Name	z [m]		$\varphi$
1		18.00	<input checked="" type="checkbox"/>	1.000
2		15.00	<input checked="" type="checkbox"/>	1.000
3		12.00	<input checked="" type="checkbox"/>	1.000
4		9.00	<input checked="" type="checkbox"/>	1.000
5		6.00	<input checked="" type="checkbox"/>	1.000
6		3.00	<input checked="" type="checkbox"/>	1.000
7		0.00	<input checked="" type="checkbox"/>	1.000

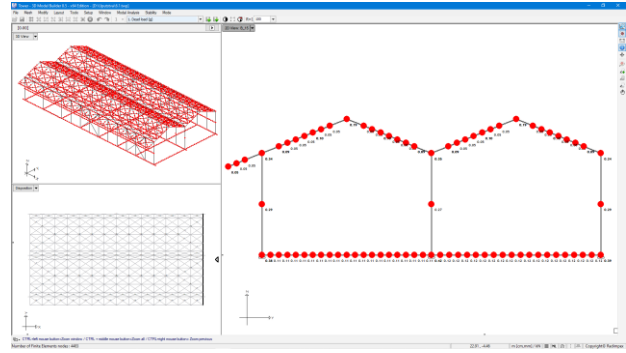
The part of the dialog where the horizontal movement of masses is limited

### **Horizontal mass displacement is limited**

By turning on the check box, an edit field becomes available, in which the length of the maximum horizontal displacement of the mass can be set. A mass that does not belong to the selected level on which the masses are grouped cannot be moved more than the specified horizontal displacement of the mass. Masses that should move more than the permitted displacement are not moved, which makes it possible to group masses even on sloping ceilings (roofs).



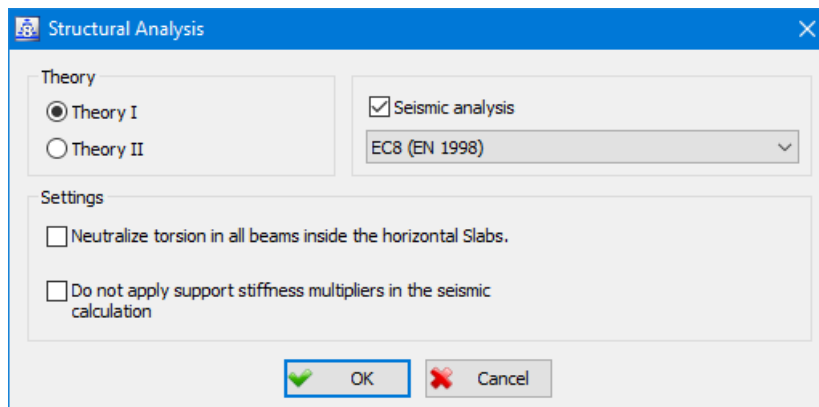
Display of masses when the horizontal movement of the masses is not limited



Display of masses when limiting the horizontal displacement of masses is set

# 7. CALCULATING THE MODEL

## 7.3 Calculation Methods

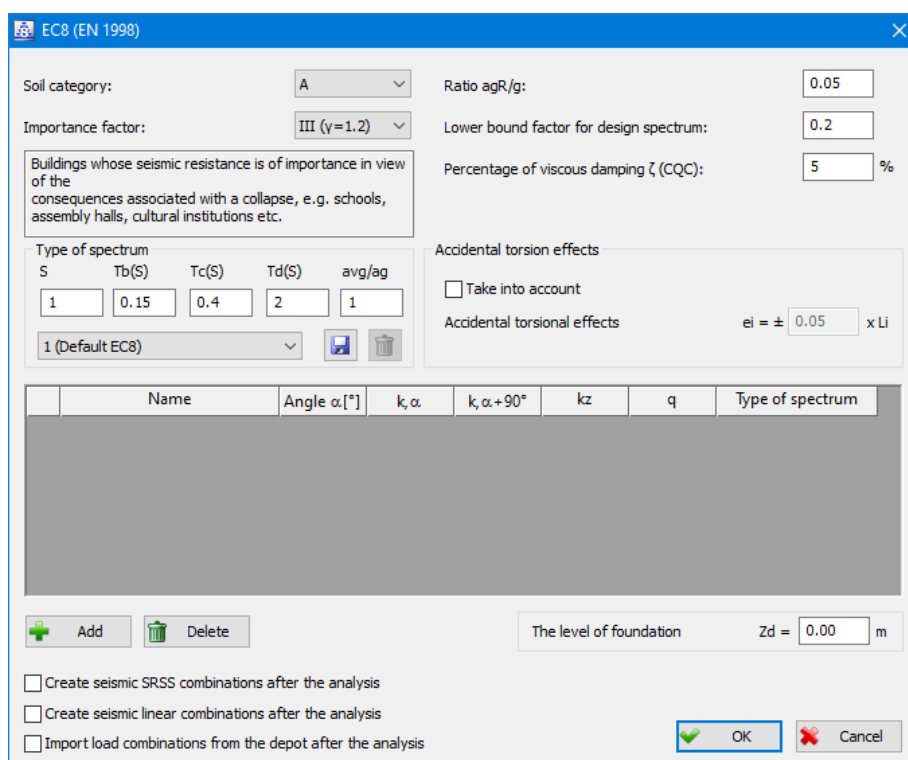


Layout of the dialog box within the model calculation command

**Do not apply support stiffness multipliers in the seismic calculation** - as the name suggests, when this check box is enabled, the support stiffness multipliers specified for the seismic model are not used in the seismic calculation. In this way, their use is limited only to the calculation of modal analysis.

### 7.3.4 Seismic Analysis

#### 7.3.4.1 EUROCODE



Appearance of the dialog box for seismic analysis in accordance with EC8 (EN 1998)

## Type of spectrum

**avg/ag** A parameter to define the type of spectrum, where:  
 avg - design ground acceleration for vertical seismic analysis  
 ag - engineered ground acceleration

### Create SRSS combination after the analysis

When this check box is turned on, after budget the program will automatically make an SRSS combination of seismic load cases.

### Create seismic linear combinations after the analysis

When this check box is on, after the calculation the program will automatically make two seismic linear combinations according to EC8 EN 4.3.5.5.1, point 3:

$$E = E_{edx} + 0.3E_{edy}$$

$$E = 0.3E_{edx} + E_{edy}$$

### Import load combinations from the depot after the analysis –

This check box is only available if there are load combinations in the repository. When set to On, after budgeting, the program will load combinations from storage. In this way, combinations containing SRSS seismic load cases that would otherwise be deleted after re-seismic calculation can be automatically saved. All you need to do is save them to storage before budgeting.

### The level of foundation Zd

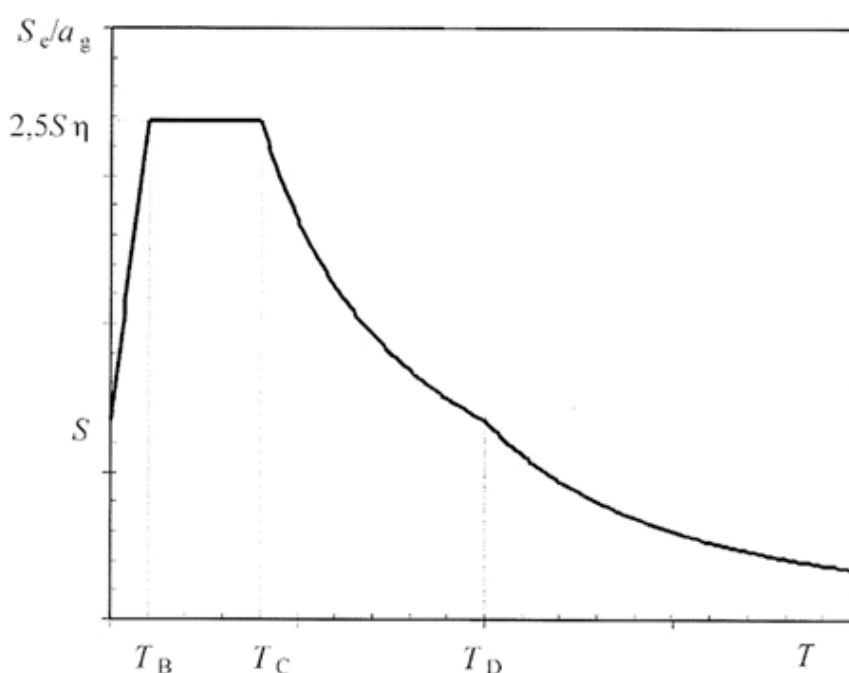
Edit field for setting the level of foundation. This data is used when generating a report for the approximate value of the base shear forces at a given level.

### 7.3.4.2 EUROCODE (Lateral forces method)

This type of analysis can be applied to buildings which response does not depend essentially on higher inherent oscillations. This requirement will certainly be fulfilled in buildings that meet the regularity requirement in height. Also, in order for this type of analysis to be used, it is necessary that the building has a basic period of oscillation  $T_1$  in two main directions that is smaller than:

$$T_1 \leq \begin{cases} 4 \cdot T_C \\ 2,0 \text{ s} \end{cases}$$

where  $T_C$  is set by the definition of an elastic spectrum.



The seismic vibration force  $F_b$  for each main direction is determined by:

$$F_b = S_d(T_1) \cdot m \cdot \lambda$$

where is:

$S_d(T_1)$  - ordinate of the project spectrum.

$T_1$  - the basic period of oscillation of the building for the translational inherent form in the considered direction.

$m$  - the total mass of the building above the foundation or above the rigid base.

$\lambda$  - a correction factor of 0.85 if the building has more than two floors and if  $T_1 < 2T_C$  or 1.0 in other cases.

Horizontal forces along the ceilings are calculated with an approximation that the horizontal displacements linearly increase in height:

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\sum z_j \cdot m_j}$$

where  $z_i$  and  $z_j$  are masses height  $m_i$ ,  $m_j$  above the level of seismic force (ie above the foundation).

This regulation is based on the method of equivalent static forces, so it is not necessary to carry out a modal analysis. Selecting this seismic analysis option opens a dialog of the following layout:

The dialog box is titled "EC8 (EN 1998) (Lateral forces method)". It contains the following fields and controls:

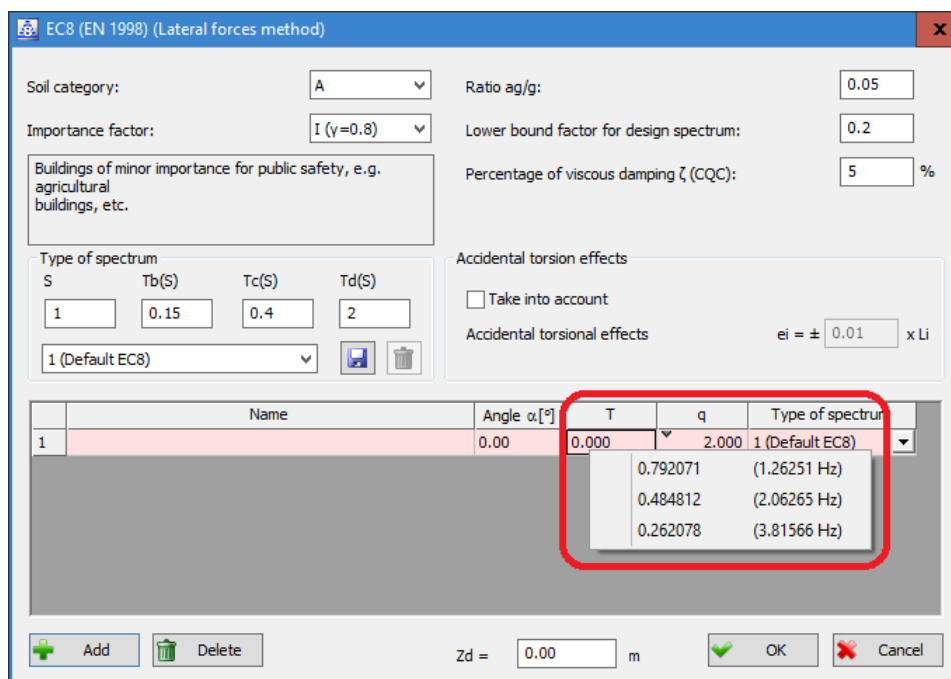
- Soil category: A (dropdown)
- Importance factor: I (γ=0.8) (dropdown)
- Buildings of minor importance for public safety, e.g. agricultural buildings, etc. (text area)
- Ratio ag/g: 0.05 (input field)
- Lower bound factor for design spectrum: 0.2 (input field)
- Percentage of viscous damping ζ (CQC): 5 % (input field)
- Type of spectrum: 1 (input field), Tb(S): 0.15 (input field), Tc(S): 0.4 (input field), Td(S): 2 (input field)
- Accidental torsion effects:  Take into account
- Accidental torsional effects: ei = ± 0.01 x Li (input field)
- Type of spectrum: 1 (Default EC8) (dropdown)
- Buttons: Add, Delete, OK, Cancel
- Zd = 0.00 m (input field)

Name	Angle α[°]	T	q	Type of spectrum

The layout of the dialog for seismic analysis by Eurocode (Method of lateral forces)

In the upper part of the dialog are parameters which define the type of soil, the type of construction, the type of spectrum, etc., while the list in the lower part of the dialog sets the direction of earthquake activity.

**T** – column where the oscillation period of the structure is set for the direction of the earthquake. An arbitrary value can be entered. If a previous modal analysis has been performed, the right mouse click over the cells in this column will open a list from which one of the calculated periods of oscillation of the structure can be selected:



Clicking on the right mouse button within the column "T" opens the drop-down menu for selecting calculated periods of oscillation

The meaning of all other parameters in the dialog is the same as for the Eurocod seismic analysis based on a multimodal analysis of the structure (see chapter 7.3.4.1).

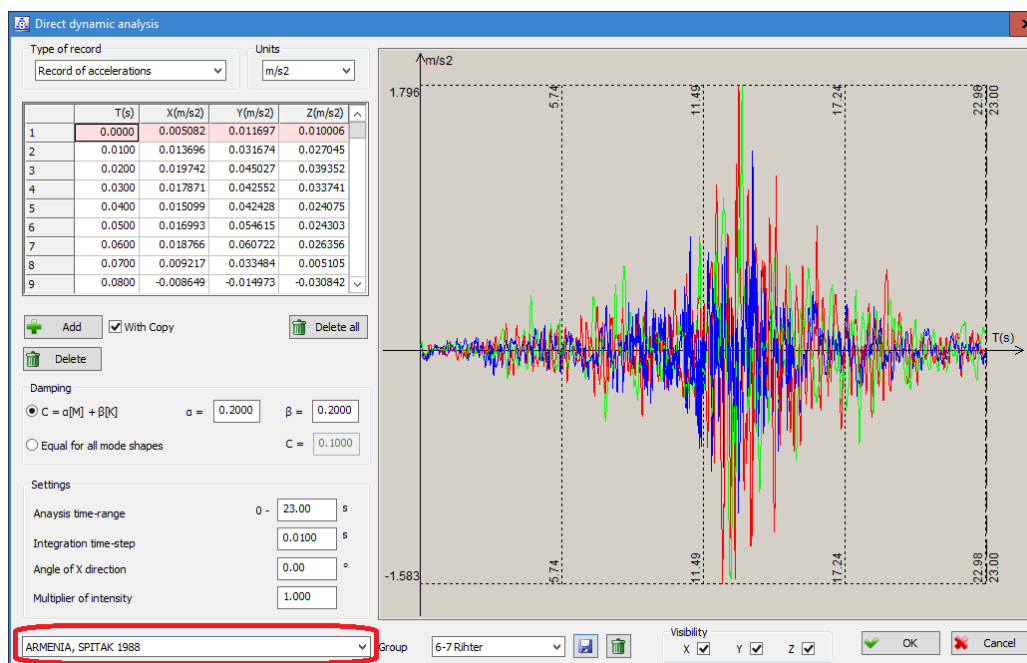
#### 7.3.4.4 Swiss Regulations (SIA)

The Swiss seismic regulation is harmonized with the changes of SIA 261 - 2020.

#### 7.3.4.6 Direct dynamic analysis

##### Recording and grouping of earthquake records

In the lower left part of the dialog, there is a list of all previously recorded earthquake records. Any of them can be selected from the list and used to analyse the direct dynamic analysis of the current model.



The list with the records of earthquakes

## Group

Since a large number of earthquake records are used for direct dynamic analysis, they can be assigned to different groups at the time of recording, all for the sake of later easier search. By choosing from this list for the current one you can set any of the previously created groups, as well as the "<All Groups>" item. The list of earthquake records shows only the earthquakes associated with the current group. When "<All Groups>" is selected, in the list of earthquake all earthquakes are recorded, regardless which groups are associated from. Then, behind their name, the group name is also written between the characters "<" and ">".



The button is used to record all the parameters that define the earthquake record. Its choice opens a dialog of the following layout:

In the top edit field, the name of the earthquake record is set, while in the lower task the group to which it will be associated is assigned. A completely new name of the earthquake record or group can be set, or one of the names in the list that can be opened by clicking the mouse above the arrows located at the right ends of these edit fields can be selected.



A button which activation from the list deletes the current earthquake saved.

### 7.3.4.7 Italian Regulations (NTC 2018)

The calculation of seismic forces according to Italian standards is also based on multi-modal analysis. Selecting this seismic calculation option opens the dialog box of the following layout.

NTC 2018

Soil category: A

Importance factor: I (Cu=0.7)

Buildings of minor importance for public safety, e.g. agricultural buildings, etc.

Topographic category: T1

Lower bound factor for design spectrum: 0.2

Percentage of viscous damping  $\zeta$  (CQC): 5 %

Type of spectrum

Ratio ag/g: 0.05

F0: 2.5

Tc\*: 0.25

Limit state: SLO

Accidental torsion effects

Take into account

Accidental torsional effects ei = ± 0.05 x Li

Name	Angle $\alpha$ [°]	$k, \alpha$	$k, \alpha + 90^\circ$	kz	q	Type of spectrum

+ Add   - Delete

Create seismic SRSS combinations after the analysis

Create seismic linear combinations after the analysis

Import load combinations from the depot after the analysis

OK   Cancel

Layout of the dialog box for seismic calculation according to Italian regulations (NTC 2018)

#### Soil category

There are 5 basic soil categories (A, B, C D and E) and based on the selected category, values of parameters F0 and Tc \*, as well as the ratio of design acceleration and ground gravity acceleration, the coefficients Ss and Cc are calculated.

#### Importance factor

There are also four classes of significance of objects that define the significance coefficient Cu - class I (0.7), class II (1.0), class III (1.5) and class IV (2.0).

The minimum nominal service life of a structure (Vn) depends on the type of structure. Vn = 10 years for temporary structures, Vn = 50 years for structures of ordinary significance, and Vn = 100 years for structures of greater significance.

The reference period is the product of these two values:

$$V_r = V_n * C_u$$

#### Topographic category

There are 4 topographic categories (T1, T2, T3 and T4) and based on that, the topographic coefficient of increase of St.

### Type of spectrum

The parameters  $F_0$  and  $T_c^*$  depend on the geographical location of the object and on the probability of exceeding in the reference time period ( $P_{vr}$ ) and are taken from public sources. In the Tower program, they are entered as input data.

In Italy there are 4 limit states - 2 are limit states of usability (SLO, SLD), and 2 are limit states of bearing capacity (SLV, SLC).

Each of the states defines the value of  $P_{vr}$  - the probability of exceeding in the reference return period.

Pvr - probability of exceeding in the reference return period	
SLO	81%
SLD	63%
SLV	10%
SLC	5%

Depending on the class of significance (class I, class II, class III and class IV), it is prescribed for how many limit states the calculation must be performed. Each of the spectra of these limit states has different parameters  $a_g$ ,  $F_0$  and  $T_c^*$ , because these values depend on both the location and the  $P_{vr}$ .

Based on previously determined data, spectrum functions are also determined.

The function of the elastic spectrum for horizontal actions is given as:

$$0 \leq T < T_B \quad S_e(T) = a_g \cdot S \cdot \eta \cdot F_0 \cdot \left[ \frac{T}{T_B} + \frac{1}{\eta \cdot F_0} \left( 1 - \frac{T}{T_B} \right) \right]$$

$$T_B \leq T < T_C \quad S_e(T) = a_g \cdot S \cdot \eta \cdot F_0$$

$$T_C \leq T < T_D \quad S_e(T) = a_g \cdot S \cdot \eta \cdot F_0 \cdot \left( \frac{T_C}{T} \right)$$

$$T_D \leq T \quad S_e(T) = a_g \cdot S \cdot \eta \cdot F_0 \cdot \left( \frac{T_C \cdot T_D}{T^2} \right)$$

The function of the elastic spectrum for vertical actions is given as:

$$0 \leq T < T_B \quad S_{ve}(T) = a_g \cdot S \cdot \eta \cdot F_v \cdot \left[ \frac{T}{T_B} + \frac{1}{\eta \cdot F_v} \left( 1 - \frac{T}{T_B} \right) \right]$$

$$T_B \leq T < T_C \quad S_{ve}(T) = a_g \cdot S \cdot \eta \cdot F_v$$

$$T_C \leq T < T_D \quad S_{ve}(T) = a_g \cdot S \cdot \eta \cdot F_v \cdot \left( \frac{T_C}{T} \right)$$

$$T_D \leq T \quad S_{ve}(T) = a_g \cdot S \cdot \eta \cdot F_v \cdot \left( \frac{T_C \cdot T_D}{T^2} \right)$$

Where:

$$S = S_S \cdot S_T$$

$$\eta = \sqrt{10 / (5 + \xi)} \geq 0,55$$

$$T_C = C_C \cdot T_C^*$$

$$T_B = T_C / 3$$

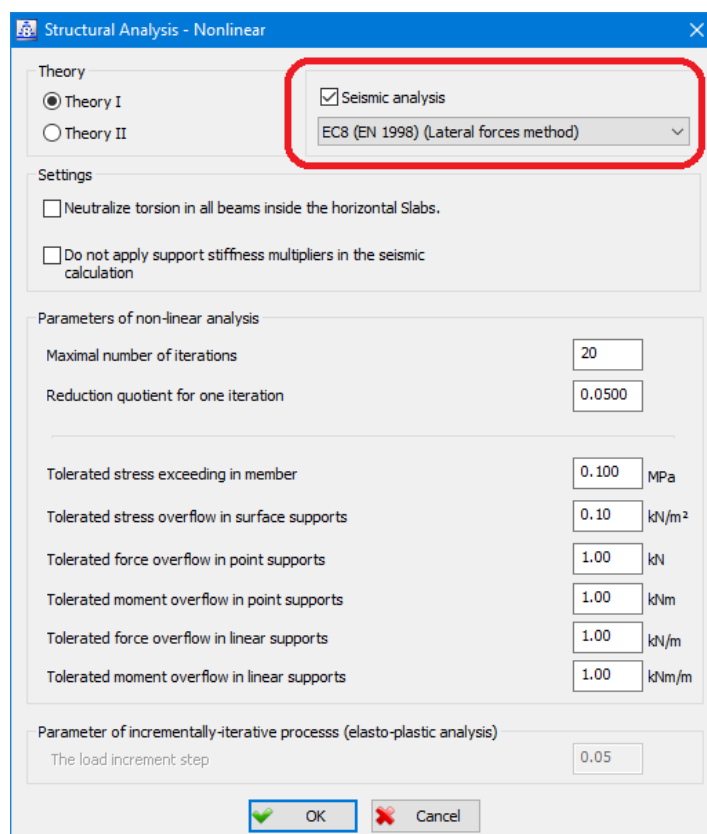
$$T_D = 4,0 \cdot \frac{a_g}{g} + 1,6$$

$$F_v = 1,35 \cdot F_o \cdot \left( \frac{a_{og}}{g} \right)^{0,5}$$

The meaning of all other parameters in the dialogue is the same as in the seismic calculation according to EUROCOD (Chapter 7.3.4.1).

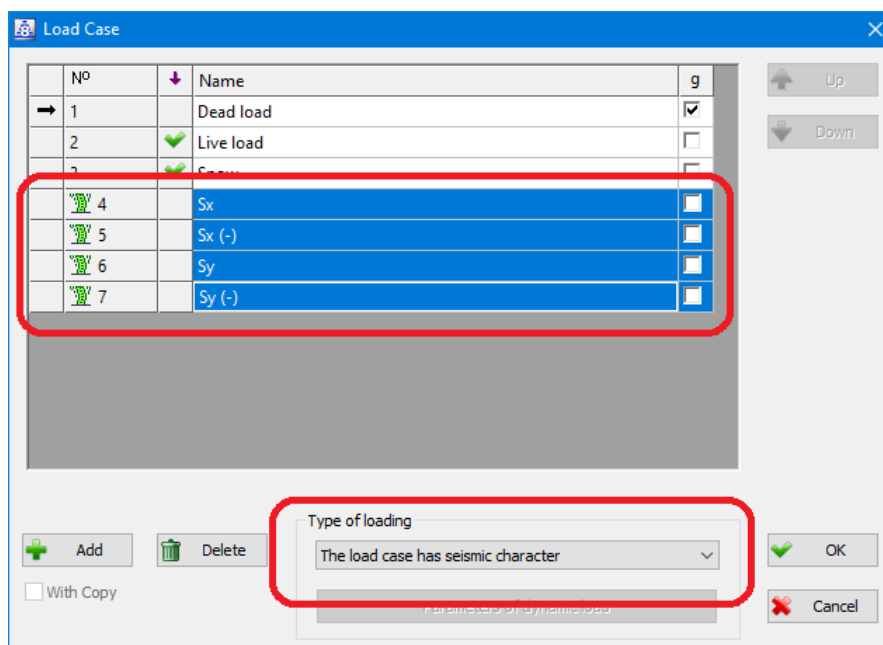
### 7.3.4.8 Seismic calculation of models with nonlinear elements

Seismic calculation is enabled for models that contain elements with nonlinear behavior.

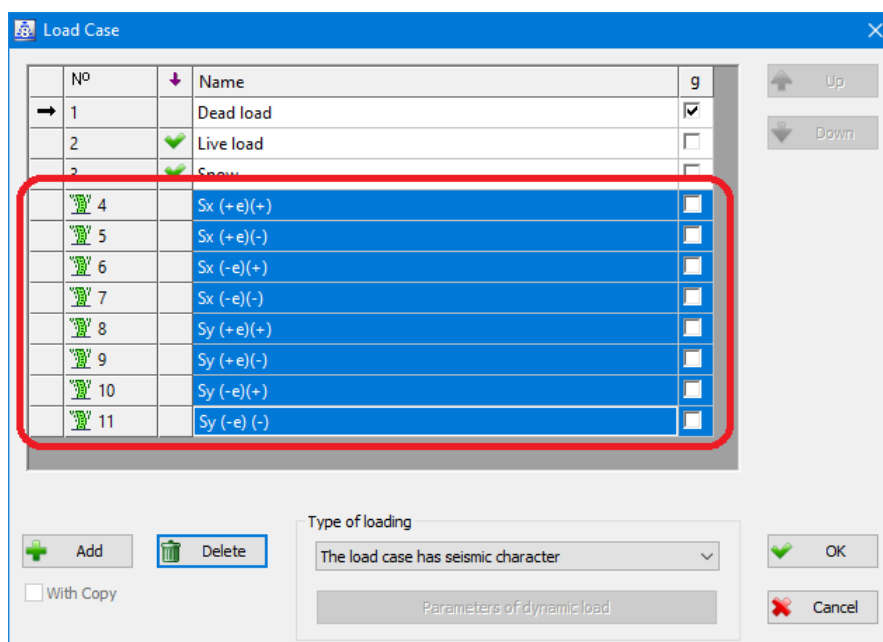


Check box for inclusion of seismic calculation

**Seismic calculation** - In order for this check box to be available, it is necessary to add two empty load cases for each of the directions of the earthquake for which the seismic calculation will be performed in the dialog of the "Load Case" command (chapter 3.2.1), at the very end of the list of loads. If accidental torsional effects are also taken into account during the seismic calculation, instead of two, four empty load cases must be specified for each of the directions of the earthquake. Also, to all of these load cases must be assigned a seismic character.



Four load cases are given, 2 for x and 2 for y the direction of the earthquake and all are assigned a seismic character



When random torsional effects are taken into account, four are given blank seismic load cases for each earthquake direction

Non-linear seismic calculation can only be carried out for regulations based on the equivalent static load method or the lateral forces method.

## Example 1:

Let's say that the seismic calculation is performed for two earthquake directions and that we called them **Sx** and **Sy** in the dialog for defining the seismic calculation data. If accidental torsional effects are not taken into account, it is necessary to add four empty load cases, two for each of the directions of the earthquake, to the "Load cases" command dialog (chapter 3.2.1) at the very end of the list, and to assign a seismic character. Regardless of what names you have assigned to these load cases, the program will map and rename them as follows during the calculation:

- |                         |  |
|-------------------------|--|
| 1 case to <b>Sx</b>     | - seismic influences with a positive sign for the x direction of earthquake action     |
| 2 case to <b>Sx (-)</b> | - seismic influences with a negative sign for the x direction of the earthquake action |
| 3 case to <b>Sy</b>     | - seismic influences with a positive sign for the y direction of earthquake action     |
| 4 case to <b>Sy (-)</b> | - seismic influences with a negative sign for the y direction of the earthquake action |

## Example 2:

If random torsional effects are taken into account, it is necessary to add eight empty load cases for the same example in the "Load cases" command dialog, four for each of the directions of the earthquake. The program will map and rename them as follows during calculation:

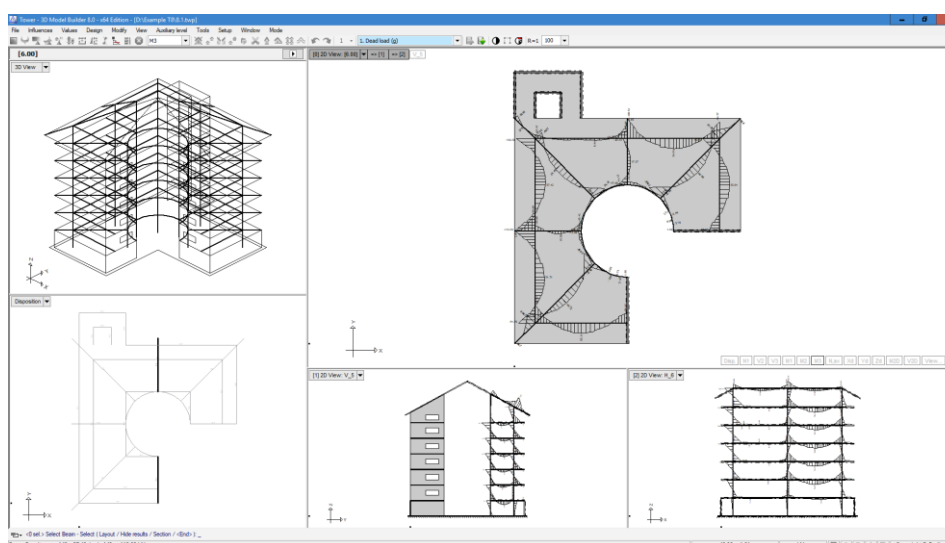
- |                              |  |
|------------------------------|--|
| 1 case to <b>Sx (+e) (+)</b> | - seismic influences with a positive sign for the x direction of the earthquake action and a positive ("left") eccentricity  |
| 2 case to <b>Sx (+e) (-)</b> | - seismic influences with a negative sign for the x direction of the earthquake action and a positive ("left") eccentricity  |
| 3 case to <b>Sx (-e) (+)</b> | - seismic influences with a positive sign for the x direction of the earthquake action and a negative ("right") eccentricity |
| 4 case to <b>Sx (-e) (-)</b> | - seismic influences with a negative sign for the x direction of the earthquake action and a negative ("right") eccentricity |
| 5 case to <b>Sy (+e) (+)</b> | - seismic influences with a positive sign for the y direction of the earthquake action and a positive ("left") eccentricity  |
| 6 case to <b>Sy (+e) (-)</b> | - seismic influences with a negative sign for the y direction of the earthquake action and a positive ("left") eccentricity  |
| 7 case to <b>Sy (-e) (+)</b> | - seismic influences with a positive sign for the y direction of the earthquake action and a negative ("right") eccentricity |
| 8 case to <b>Sy (-e) (-)</b> | - seismic influences with a negative sign for the y direction of the earthquake action and a negative ("right") eccentricity |

# 8. POST-PROCESSING

## 8.1 Purpose and Description of Module Operation

The review of the results

The analysis results are displayed in the "2D view" window. Since the program can have more 2D windows, and in each of them can be installed arbitrarily selected parts of the structure (frames, levels, auxiliary views), it means that at the same time, the results can be displayed in different layouts.

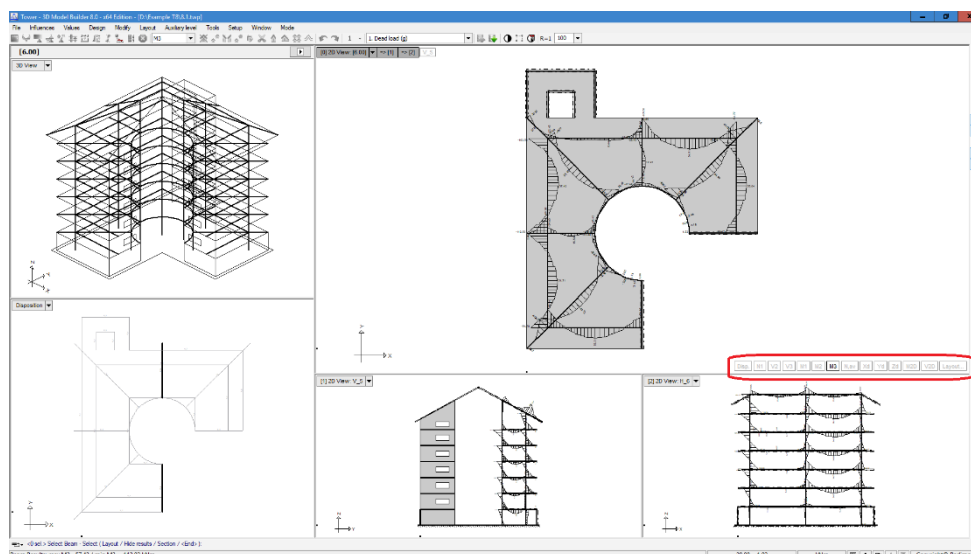


Simultaneous display of beam influences in three different layouts

Quick choice of influences

The influences that occur when starting the command can be subsequently changed by using command line options, a quick selection list, or through the field of the influence names appearing in the lower right corner of the main 2D window.

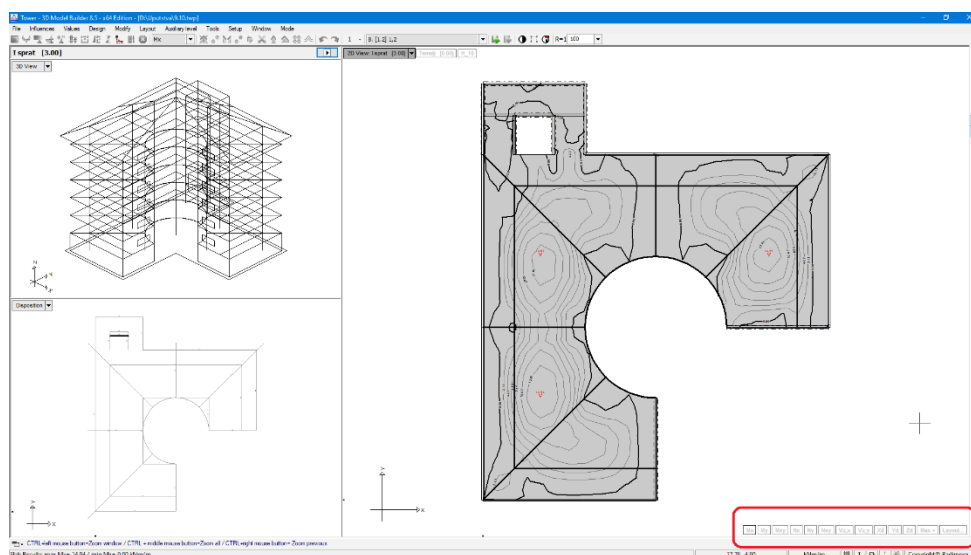
At the moment of launching the appropriate command for displaying the influence, the lower right of the main 2D window displays the fields in which the influences can be changed quickly. On each of these fields, the name of the influence is printed, and by clicking the mouse through it, it will be displayed on the screen.



The fields for quick choice of influences

As in the case of a closed list, the fields for a quick selection of influences can be used in the presentation of all types of influences: static influence in all elements of the structure, influence in the gearbox, the results of the dimensioning of concrete slabs and concrete beams. Regarding that, a number of influences can be displayed for some entities, due to limited space for quick selection, only the names of the most commonly used names will be available. Other influences can be selected from the dialog that opens with the **"Layout ..."** field. This way of choosing the influence is the most transparent and fastest to use.

In case that influences in the slab or surface support are displayed, for the envelope load case, the fields **"Max +"** and **"Max -"** will also be available, the selection of which defines whether isolines will be drawn for maximum positive or maximum negative influences.



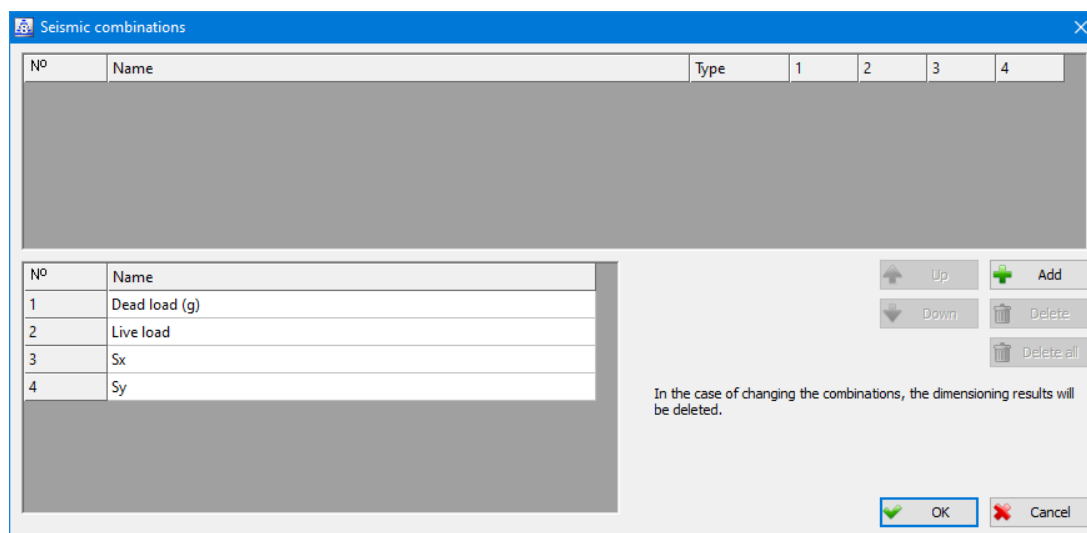
Influence quick selection fields

In the "Tower" it is possible to print numerical values of the influences on the diagrams of line entities, as well as the values of isolines of surface entities that are displayed in the window "3D view". The display of these values is defined by the check box **"Show diagram values in 3D"**, which is located in the dialogue box of the command **"Functionality"**, in the file "Results" (chapter "3.5.2 Functionality"). For the same purpose, the command **"Show diagram values in 3D"** is provided, which is called from the drop-down menu "Values" (chapter "8.28").

## 8.2 Selecting the Current Load Case

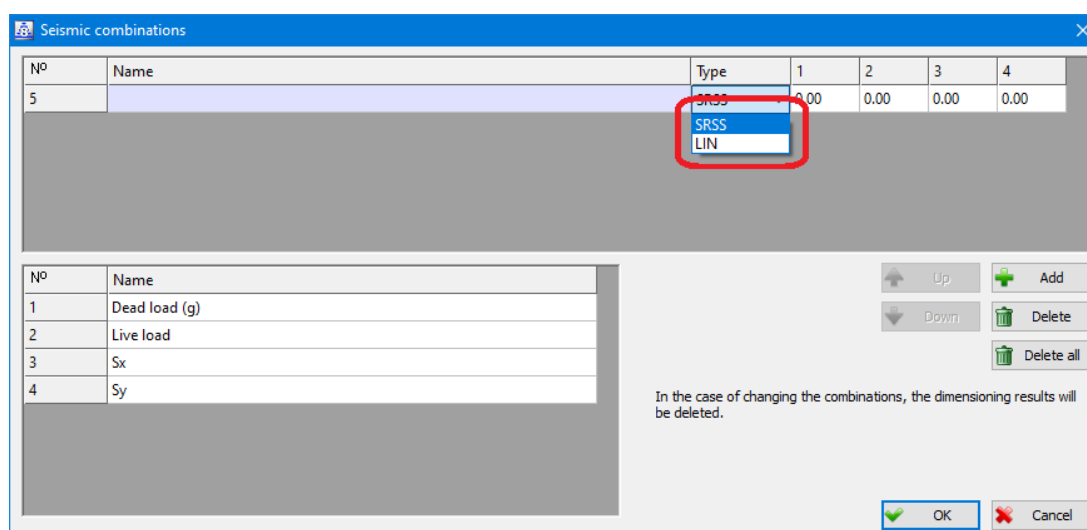
### 8.2.1 Creating seismic combinations

Within the "Load cases" command, it is possible to create seismic load combinations. By selecting the "**Seismic combinations**" command field, a dialog with the following appearance opens:



Layout of the dialog for creating seismic combinations

**Type** Column in which from a closed list for each of the seismic combinations you can choose the method of calculating the value of the combination of seismic actions: "**SRSS**" or "**LIN**".



List for choosing the method of calculating the value of combining seismic actions

### Creation of "linear" seismic combinations

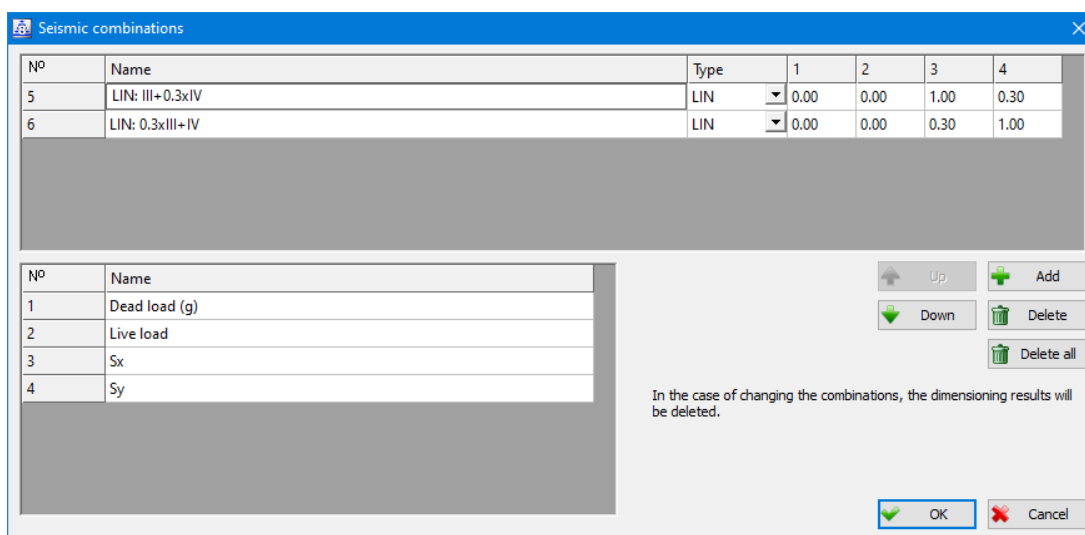
This kind of the combination of seismic actions allows the creation of seismic combinations in accordance with EC8 EN 4.3.5.5.1, point 3:

$$E = E_{edx} + 0.3E_{edy}$$

$$E = 0.3E_{edx} + E_{edy}$$

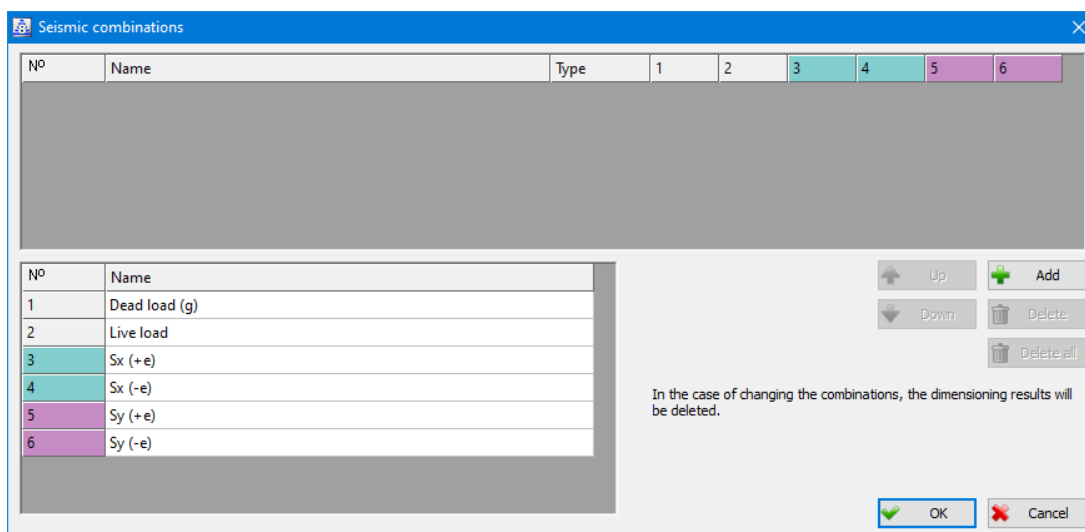
This method of combining seeks the highest possible value when adding up the absolute values of the influence of the ingredients, with one of them having a multiplier of 1.0, and all others 0.3. It is necessary to create two linear seismic combinations.

The linear seismic combination is defined by selecting the method of calculating the value of the combination of seismic actions "**LIN**" in the "Type" column, and then in the columns that have the same serial number as the load cases that enter the combination, assign a multiplier.



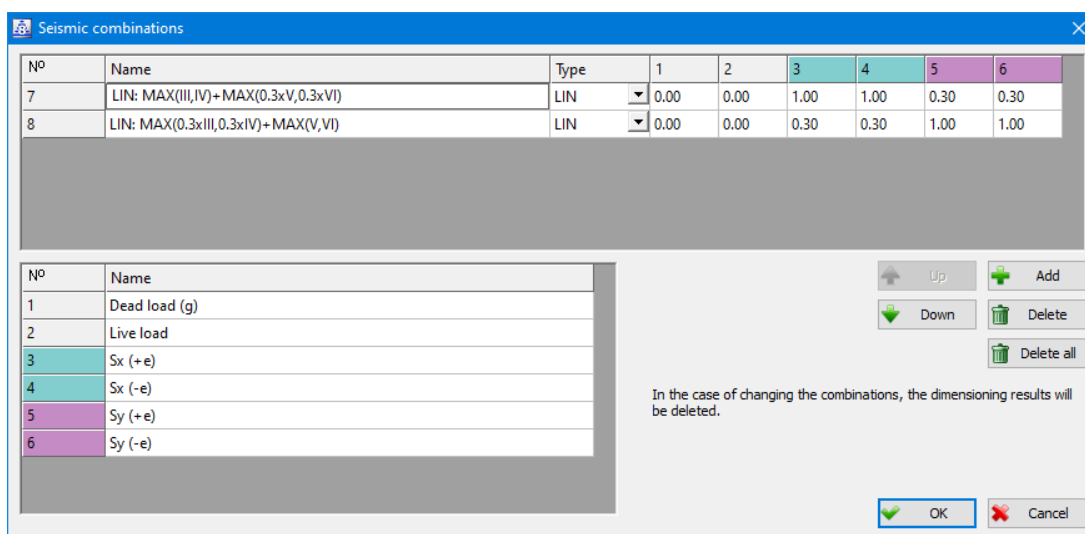
By assigning the multiplier to the third and fourth load case, two linear seismic combinations were created

A special case is the creation of linear seismic combinations from pairs of multimodal seismic load cases created by including the random torsion effect. Then, in each node of the every element, the influence values of these two load cases are compared and the greater of them is adopted for the combination. In the dialog for creating seismic combinations, these load pairs are marked with special colors.



Load pairs created by including the effect of random torsion in multimodal seismic calculation

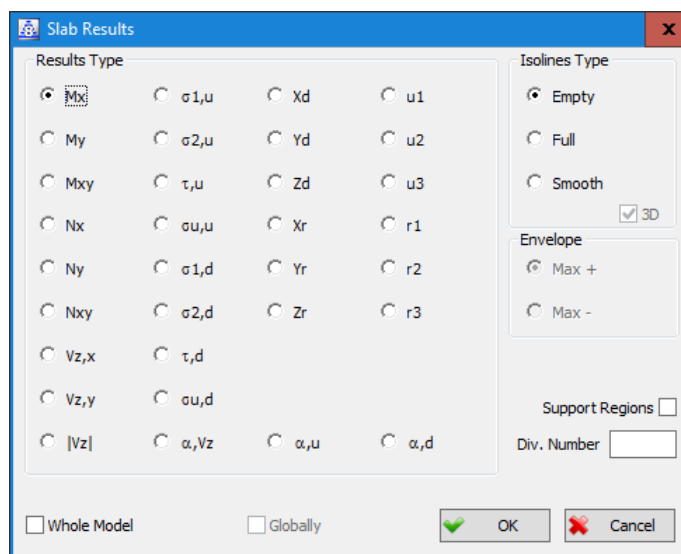
The creation of linear seismic combinations from these pairs of loads is done in the same way as for ordinary load cases - by choosing the method of calculating the value of the combination of seismic actions "LIN" in the column "Type" and entering the multiplier in the corresponding columns.



Two seismic linear combinations of two load pairs were created

In the seismic linear combination "LIN: MAX(III,IV)+MAX(0.3xV,0.3xVI)", the influence is calculated by adding the maximum absolute value of 0.3xV and 0.3 xVI to the maximum absolute value of load cases III and IV, while the influence in the second seismic combination "LIN: MAX(0.3xIII,0.3xIV)+MAX(V,VI)", is calculated by adding the maximum absolute value of 0.3xIII and 0.3xIV to the maximum absolute value of load cases V and VI.

## 8.5 Influences in Slab/Wall



Appearance of the dialog box for selecting the influences in plates and walls

### Results Type

- ◆  $\alpha, T_z$  Representation of the trajectory of transverse forces in the slab, representing the angle of the resultant vectors  $T_{z, x}$  and  $T_{z, y}$ . This influence is not shown via isolines, but with arrows that clearly illustrate the direction of its action. An arrow will be placed in the middle of each finite element in the direction of the trajectory of the transverse forces. Arrows will not be able to be displayed if the program is required to display an enveloping or seismic load case.

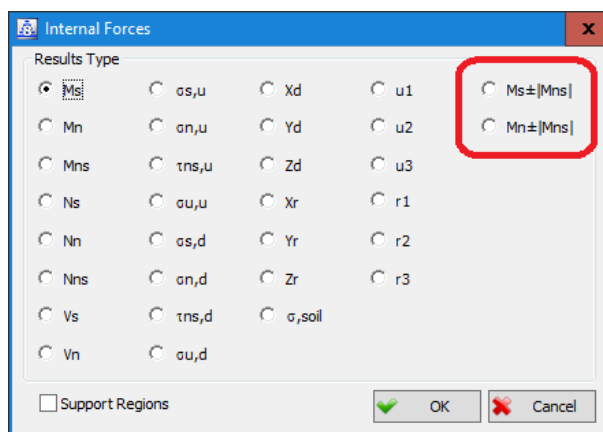
### Envelope

Switches, located in the "Envelope" part of dialog box, will be available if the current load case is selected:

- a load case in which a moving load is given,
- envelope of extreme influences,
- multimodal seismic load case
- SRSS combination.

### 8.5.2 Influences in Arbitrary Cross-section (Section)

When there are no isolines on the screen, there are two new influences in the dialog box that opens before the intersection position is set, for which cross-sectional force diagrams can be drawn:



Two new influences for the vector cross section in the slab

**$M_s \pm |M_{ns}|$**  The bending moment acting in the direction of a given cross section plus the absolute value of the twisting moment.

**$M_n \pm |M_{ns}|$**  The bending moment acting in the normal direction on a given cross-section line plus the absolute value of the twisting moment.

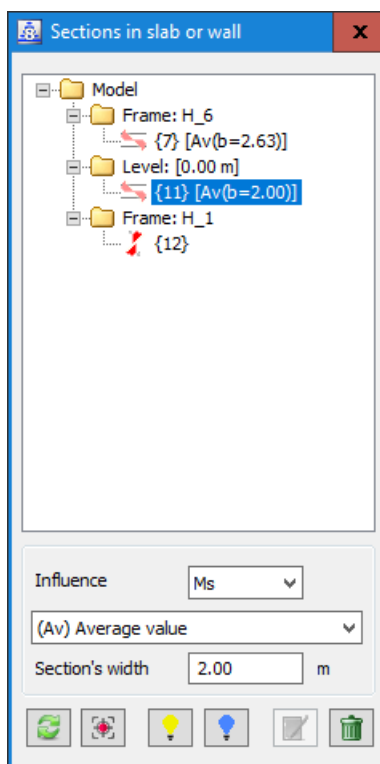
The program enables remembering the position of the line sections, as well as the specified width of the strips of set sections when recording a model, so that they remain permanently remembered for later use.

If necessary, the user can subsequently delete the previously defined sections in a quick and easy way by activating the right mouse button, the "Reset" command (see chapter 8.20).

We note that in the "Visibility" dialog (see chapter 2.10.1) it is possible to turn on or off the sectional visibility in the slab.

### 8.5.6 Review of influences in slab sections

Choosing the "**Review of influences in slab sections**" command, which is located in the analysis module in the "Values" drop-down menu, the following dialog opens:





The dialog displays a list of all sections in the slab that are in the model grouped by assemblies. Depending on whether or not the isolines are plotted on the screen or not, two cases can occur. If the isolines are present on the screen, the dialog will show all the sections set at the moment of display of the isolines of the set influence, so that the sectional diagram will also refer to these influences. Otherwise, when there are no isolines on the screen, all the sections that were set until the isolines were shown will appear in the list, while in the dialog you can select the influence you need to plot the sectional diagrams.

**Influence** A closed list is in the dialog only if the isolines are not present on the screen and from it is selected the influence for which it is necessary to draw the diagrams of the sectional forces. It is necessary to select the cross section from the sections list first and then the list will become available for the choice of influences. The following influences are available: sectional forces, stresses and deformations. Note that if the "Nn", "Ns" or "Nns" is selected and the section has no width for the type of influence, a closed list with options "**Section's influences**" and "**Lever and force**" will appear below this list.

### Section's width

If you want the integral influences along the set section line at a specified width, then you need to specify the desired width of the selected section in the list by entering the values in the given edit field.

 - icon indicating that the width of the tape is set for the set section, that is, that the influences will be integral along the specified line of the section at the desired width.

 - icon indicating that the width of the tape "0" is set for the set section, that is, a simple diagram will be obtained without integrating.

If the width of the tape is not zero, the closing list for selecting the type of diagram you want to display is displayed above the edit field: "**M (Lever)**", "**T (Area)**", "**Av (Average value)**". Regardless of which option you choose, the program will print the specified width of the "**b =**" bar and mark the selected diagram type in the

dialog along with the set section, as well as at the beginning of the cut line. The average value designation is "Sr", for lever is "M" and for area is "T".



Adding and deleting a sectional layout may cause the sectional layout in the dialog to no longer correspond to the condition in the model. By clicking this button, the list is refreshed, its alignment with the current state of the section in the model.



The activation button in the 2D view is set for the current assembly containing the selected section. The currently selected section in the list will be particularly highlighted in the drawing. Setting in the 2D view of the assembly to which the section belongs you can do by the left double click of the mouse over its name in the list, or selecting the "Go to assembly" command from the drop-down menu that opens with the right mouse click over the section name.



Clicking on this button turns on the visibility of the selected sections, if they were previously turned off. If assemblies are selected in the trunk, the visibility of all sections belonging to them will be turned on. Cross-section visibility can also be activated by selecting the option "Visible" from the drop-down menu, which is opened by right-clicking on the section name or assembly.



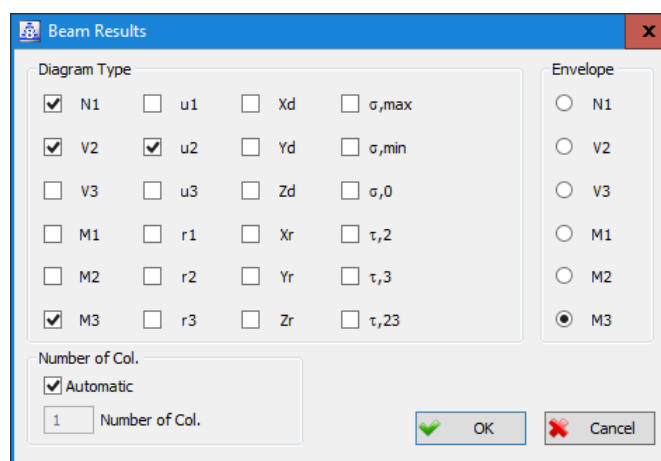
Clicking on this button turns off the visibility of the selected sections. If assemblies are selected in the trunk, the visibility of all sections belonging to them will be excluded. Cross-section visibility can also be turned off by selecting the "Not visible" option from the drop-down menu that opens by right-clicking on the section name or assembly.



The activated button is currently removes the chosen section from the drawing. The section deletion command is also in the menu that opens with the right mouse click over the name of the section in the list. Since the sections in the list are grouped by circles, the right mouse click over the name of the assemblies opens a menu with the option "Delete all", which deletes all sections from the set assembly.

## 8.6 Influences in Beams

### 8.6.1 Individual Diagrams

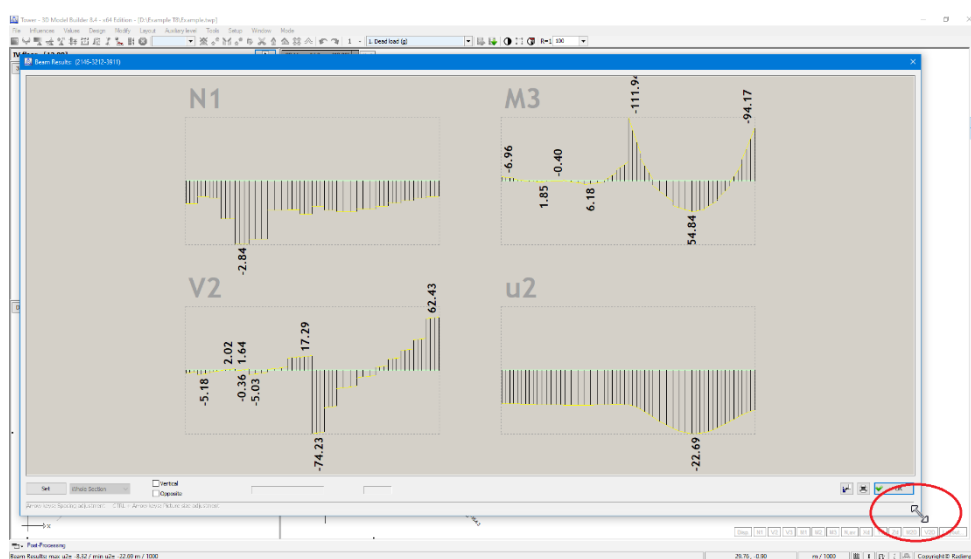


Switches, located in the "Envelope" part of dialog box, will be available if the current load case is selected:

- a load case in which a moving load is given,
- envelope of extreme influences,
- multimodal seismic load case
- SRSS combination.

For the selected influence in the "Envelope" part, extreme value envelope will be constructed, and for other diagrams corresponding values.

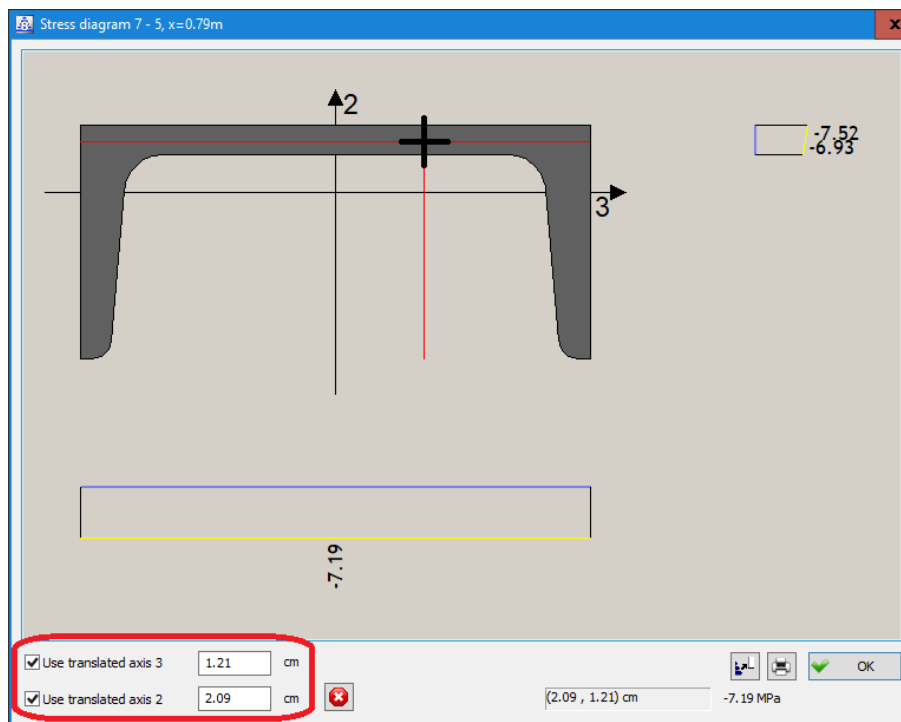
The size of the dialogue in which the effects in the selected beams are displayed can be changed by stretching, in order to ensure the best possible visibility. It is necessary to first place the mouse pointer on one of the edges of the dialog or on one of its corners, where it will change its shape into an arrow that shows possible directions of stretching. When this happens, you should press the left mouse button and stretch the dialog to the desired position without releasing it.



Stretching the lower right corner changes the size of the dialogue

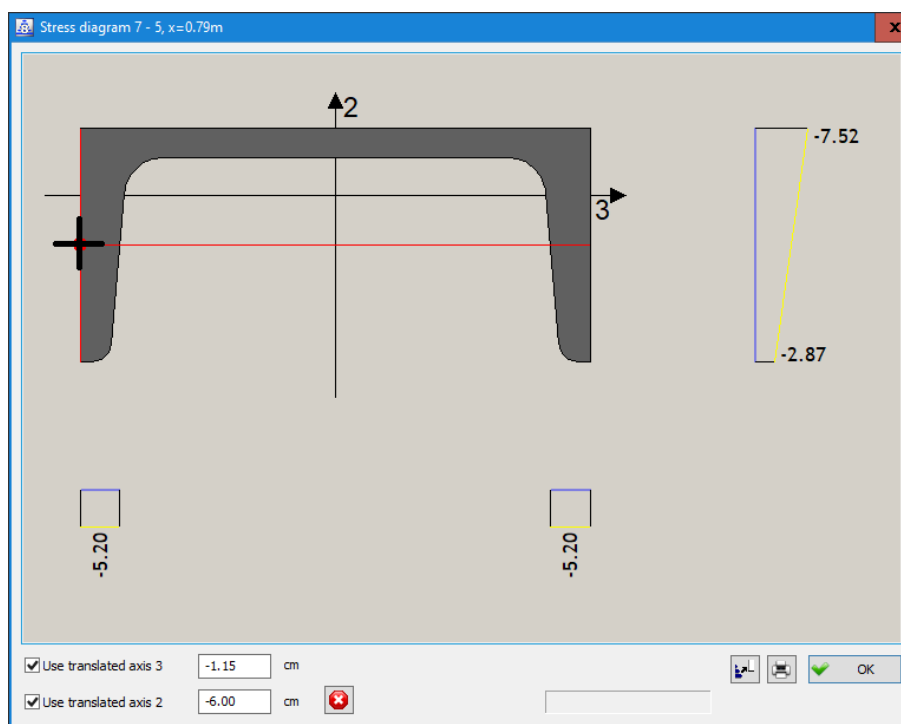
### 8.6.3 Diagrams in Cross-section

The position of the cross section in relation to which the tension diagram is drawn can also be set with the mouse by clicking above any point of cross section of the beam. The displacement of the given intersection lines with respect to the local axes of the beam is printed in the edit boxes "Use translated axis 3" and "Use translated axis 2".



The new position of the cross-section lines is given by clicking the mouse above the cross-section of the beam

When the mouse pointer is positioned above one of the characteristic points from the cross-section of the beam, the program will display a circle of red, indicating that the mouse click will select that point as a reference for the placement of the cross-sections with respect to which the tension distribution diagram is drawn. The characteristic points are: the intersection points of the contour lines of the cross section, the points from the middle of the contour lines, the intersection points of the contour lines with the local axes of the beam, and the intersection points of the contour lines with the given intersection lines.



New intersection lines are defined using the characteristic point from the middle of the contour segment of the cross section of the beam



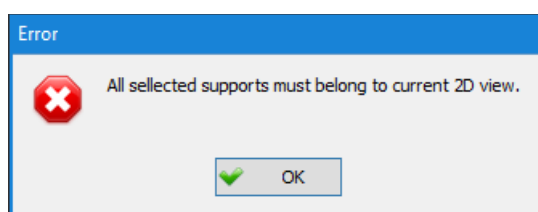
By clicking this button, the position of the intersection lines can be canceled, so that they again coincide with the local axes of the beam.

## 8.10 Resultant of the reactions

Using the command "**Resultant of the reactions**" the sum of the reactive load of the selected supports can be obtained in the direction of the global coordinate axes, eccentricity, as well as a graphical representation of the resultant reaction. The command is most commonly used to control the correctness of a given load and other structural elements. Selecting a command from the "**Influences**" drop-down menu enters the procedure for selecting the supports whose reaction sum is requested, and the command line takes the following form:

<0 sel.> Supports - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End>):

At the same time, surface, line and point supports, which belong to the same 2D view, can be selected. If the selection is made from a 3D view, care must be taken that all selected supports belong to the current 2D view, otherwise the program will throw out the appropriate message:



After the selection is completed, the program opens a dialog showing the values of the results of the reactions of the selected supports and its position:

Resultant	Value [kN]
Rx	0.00
Ry	0.00
Rz	31319.63

Eccentricity	Value [m]
ex	-2.39
ey	0.70

Report

Current load case

1. Dead load (g)

Treat envelope as a single loadcase

All basic load cases

All load case combinations

All loads

Selected load cases and combinations

Choice

Report

Display labels

OK

The graphical representation shows a cross-section of a beam with a resultant reaction force. The resultant is located at a distance of 23.00 m from the left edge and 0.70 m from the bottom edge. The eccentricity values are ex = -2.39 m and ey = 0.70 m. The beam dimensions are 27.50 m in height and 23.00 m in width.

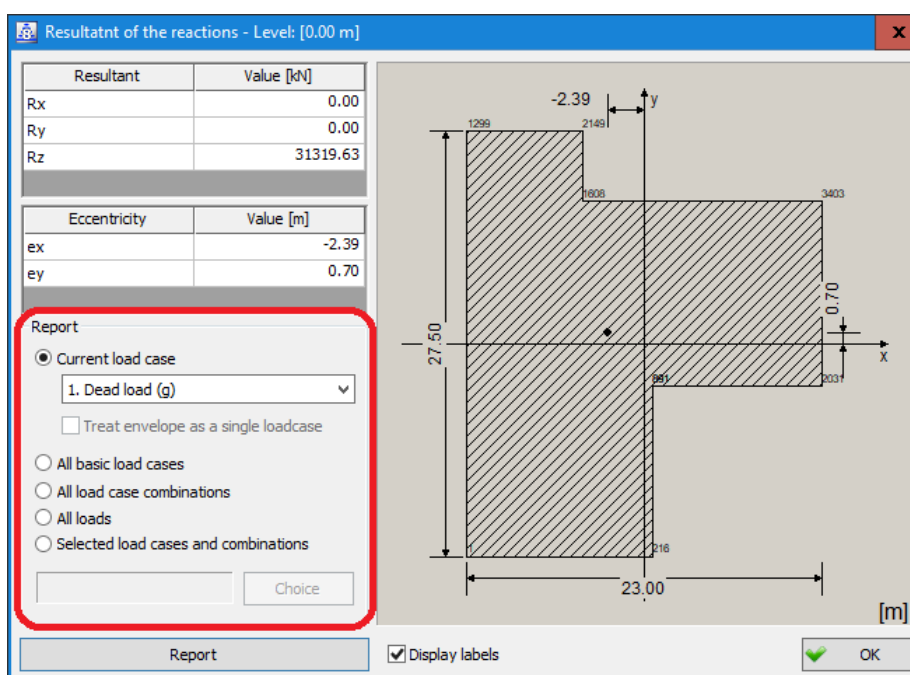
The layout of the dialogue to show the result of the reactions of the selected supports

In the left part of the dialogue, the tables show the values of resultant and eccentricity, while in the right part of the dialog there is a graphical representation.

## Display labels

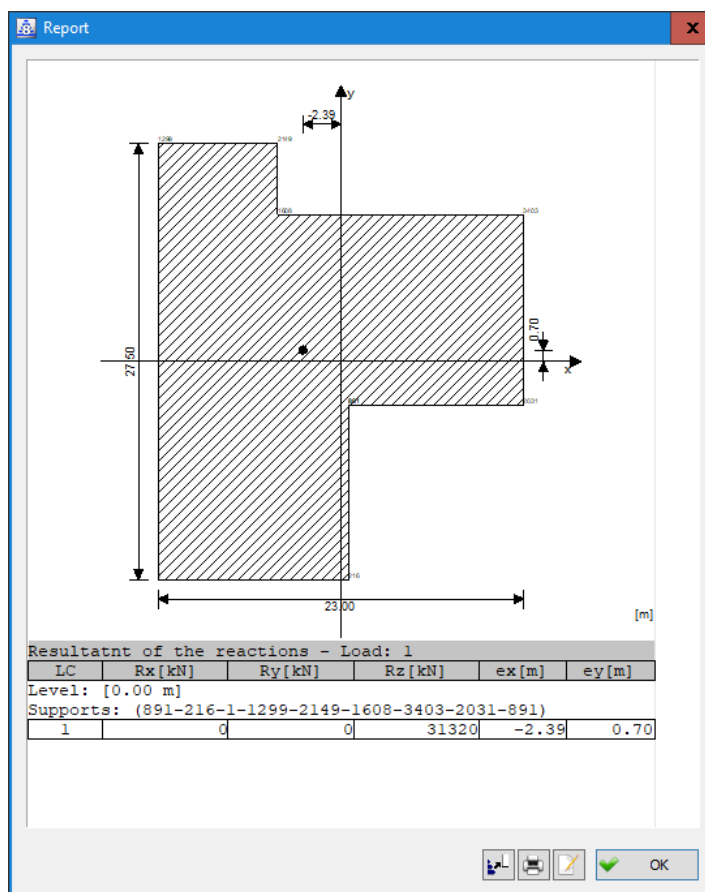
The status of the check box determines whether the drawings in the dialog and the report will show the numbers of the support nodes, or the markings of the positions of the supports, if any. If the check box is on, the drawings will simultaneously show position labels and node numbers for dots and line supports, while surface mounts will show node numbers if position labels are not specified, and if only the position labels are displayed.

The program also allows you to create a text report for the resulting reactions. In the lower left part of the dialog ("Report"), the load for which a text report will be created is made. In addition to the individual load case selected from the list, a report can also be generated for: "All basic load cases", "All load case combinations", "All loads" or "Selected load cases and combinations". If a report is created for a load case that is not currently set up in the result processing module, the reaction and eccentricity result tables in the dialog will be blank.



The part of the dialog where the load selection is made

**Report** A button that opens a window reserved for displaying a text report of the reaction:



Appearance dialog for text report display

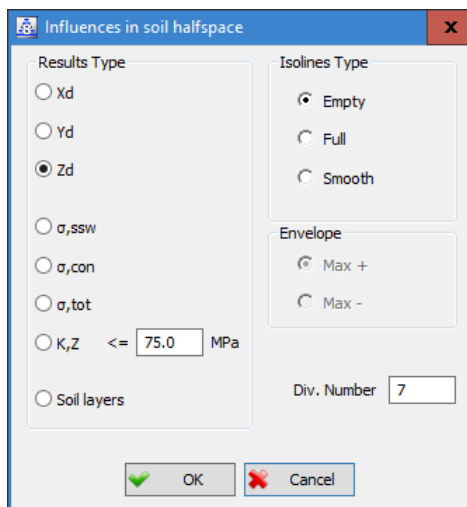
Working with this window is exactly the same as in other commands in the program, in which text reports are created.

## 8.12 Proxy entity

### 8.12.2 Influences in the proxy entity

Influence in the proxy entity can be displayed for basic cases of load, combination and envelope load.

## 8.13 Influences in soil halfspace



The layout of the influence selection dialog in the half-space

**K,Z** the equivalent coefficient of the stiffness of the soil bed of the half-space, which represents the ratio of tension and deformation of the half-space node in contact with the structure. In the edit field, which is on the right from the name of the assigned influence, the boundary stiffness of the soil bed is determined by which isolines will be displayed.

## 8.17 Bill of quantities

Beams – bill of quantities

Beams - bill of quantities by sets						
Set	Section/Material	$\gamma$ [kN/m <sup>3</sup> ]	L [m]	O [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]
1	b/d=20/40 Concrete C 30	25.000	132.19	158.63	10.575	26.959
2	D=40 Concrete C 40	25.000	195.45	245.62	24.562	62.615
3	b/d=20/40 Concrete C 30	25.000	747.75	897.30	59.820	152.50
4	b/d=10/15 Timber-Conifers-Sol.	5.000	8.228	4.114	0.123	0.063
Total:			1083.6	1305.7	95.080	242.14
Beams - bill of quantities by cross sections						
Section/Material	$\gamma$ [kN/m <sup>3</sup> ]	L [m]	O [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]	
b/d=20/40 Concrete C 30	25.000	879.94	1055.9	70.395	179.46	
D=40 Concrete C 40	25.000	195.45	245.62	24.562	62.615	
b/d=10/15 Timber-Conifers-Solid Wood	5.000	8.228	4.114	0.123	0.063	
Total:			1083.6	1305.7	95.080	242.14
Recapitulation of materials quantities						
Material	$\gamma$ [kN/m <sup>3</sup> ]	O [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]		
Concrete C 40	25.000	245.62	24.562	62.615		
Concrete C 30	25.000	1055.9	70.395	179.46		
Timber-Conifers-Solid Wood	5.000	4.114	0.123	0.063		
Total:			1305.7	95.080	242.14	

The table with premeasurement for beams

- O [m<sup>2</sup>]** - the outer surface of the shells of all the beams of the assigned set/cross-section.

### Recapitulation of materials quantities

Slabs - bill of quantities						
Set	d [m]/Material	$\gamma$ [kN/m <sup>3</sup> ]	P [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]	
1	d=0.160 Concrete C 30	25.000	3367.4	538.78	1373.5	
2	d=0.600 Concrete C 30	25.000	441.75	265.05	675.69	
Total:			3809.1	803.83	2049.2	
Beams - bill of quantities by sets						
Set	Section/Material	$\gamma$ [kN/m <sup>3</sup> ]	L [m]	O [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]
1	b/d=20/40 Concrete C 30	25.000	132.19	158.63	10.575	26.959
2	D=40 Concrete C 40	25.000	195.45	245.62	24.562	62.615
3	b/d=20/40 Concrete C 30	25.000	747.75	897.30	59.820	152.50
4	b/d=10/15 Timber-Conifers-Sol.	5.000	8.228	4.114	0.123	0.063
Total:			1083.6	1305.7	95.080	242.14
Beams - bill of quantities by cross sections						
Section/Material	$\gamma$ [kN/m <sup>3</sup> ]	L [m]	O [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]	
b/d=20/40 Concrete C 30	25.000	879.94	1055.9	70.395	179.46	
D=40 Concrete C 40	25.000	195.45	245.62	24.562	62.615	
b/d=10/15 Timber-Conifers-Solid Wood	5.000	8.228	4.114	0.123	0.063	
Total:			1083.6	1305.7	95.080	242.14
Recapitulation of materials quantities						
Material	$\gamma$ [kN/m <sup>3</sup> ]	O [m <sup>2</sup> ]	P [m <sup>2</sup> ]	V [m <sup>3</sup> ]	m [T]	
Concrete C 40	25.000	245.62		24.562	62.615	
Concrete C 30	25.000	1055.9	3809.1	874.22	2228.6	
Timber-Conifers-Solid Wood	5.000	4.114		0.123	0.063	
Total:			1305.7	3809.1	898.91	2291.3

The table of the recapitulation of the quantity of material

- O [m<sup>2</sup>]** - the outer surface of the shells of all beams that are created from the given material.
- P [m<sup>2</sup>]** - the surface of all slabs that are created from the given material.

## 8.18 Structural Design Results Textual Representation

### Beams

The dialog box for generating textual reports in beams and columns - the layout

#### From the ends only

Only values in points at the ends of the selected beams / columns are shown in the report.

The program allows you to obtain textual reports as well for influences in beams grouped in nodes. Textual reports for forces grouped by nodes can be applied when analysing bonds in metal structures.

Parameters used to define the report creation for influences in beams grouped in nodes

### Forces grouped by nodes

A check box that becomes active only if the "Extreme" check box is turned off, and turning it on a closed list for displaying the text report is available:

**All forces** - A textual report for forces grouped by nodes will be a complete report, with all the cases of load and cross-sectional forces.

**Extreme and corresponding** - A textual report for forces grouped by nodes will contain the extreme values of the cut-off forces and the corresponding values.

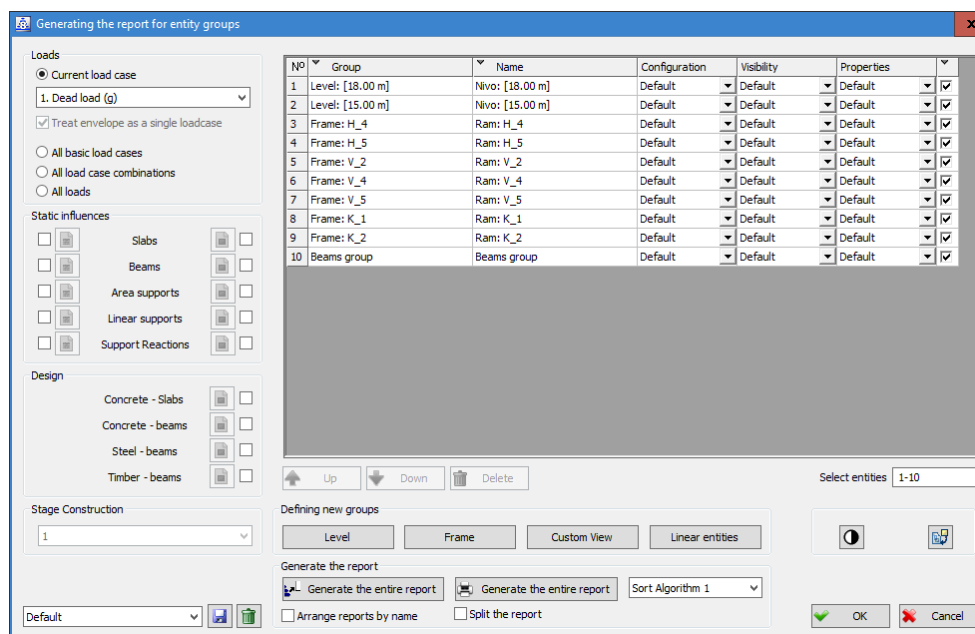
**Extremes only** - A textual report for forces grouped by nodes will only contain extreme values of cut-off forces.

A textual report of influence in beams grouped by nodes contains a sketch of the node, with all the beams that included.

Forces in the joint 1142 - Load: 1-16						
Mark	LC	N1 [kN]	V2 [kN]	M2 [kNm]	M3 [kNm]	
(1142 - 454)	1	18.472	-27.944	0.276		-31.234
b/d=20/40	2	0.740	-5.104	0.036		-5.971
	3	27.059	1.286	0.166		1.509
	4	29.698	+3.233	0.258		+4.641
	5	26.046	-45.379	0.427		-51.124
	6	19.581	-35.599	0.330		-40.192
	7	-8.365	-30.761	0.121		-34.535
	8	-11.004	-32.707	0.029		-37.667
	9	+48.391	-26.242	+0.544		-28.385
	10	45.752	-28.188	0.452		-31.517

A dialog with the content of the created text report for forces grouped by nodes

## 8.19 Generating the report for entity groups



Layout of the dialog that is opened by activating the command "Generating the report for entity groups"

For each group of entities, the "**Visibility**" column contains a closed list from which you can select any previously recorded visibility configuration that will generate reports.

If the report for all groups is generated with the same configuration, it is enough to assign it to one group, and then select the "**Apply to all**" command from the drop-down menu, which opens with the right mouse click over its name. In this way all selected groups will be associated with the selected configuration

For each group of entities, in the column "**Properties**" there is a closed list where you can choose any previously recorded configuration of block properties (size, margins, scale...) that will generate reports. operating with the configurations in this column is exactly the same work with the configurations in the column for visibility, and we will not re-explain it.

We note that the created entity groups are recorded in the file when the model is taken and thus remain permanently stored for later use.

In addition to the created groups, all the configurations assigned to them in the columns "Visibility", "Properties" and "Configuration" are recorded in the file. When starting the command, the program compares these configurations and configurations that are recorded in the database. If there is no configuration in the database assigned to a group, a sign (\*) is printed in front of its name indicating that it is loaded with the current model and can only be used in it.



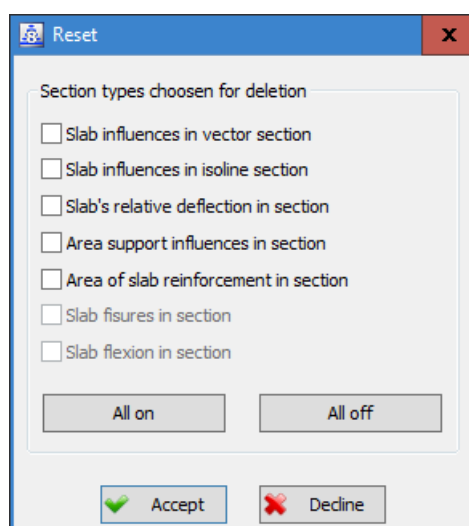
Selecting this icon opens the "Visibility" dialog where the visibility of each element of the construction and load is adjusted separately. It also records configurations with different visibility settings for the entities. The configurations that are loaded with the current model will appear at the end of the closed list with the configurations, and sign (\*) will be displayed in front of their name. If necessary, in this dialog they can be saved to the database for later use in the program. operating in this dialog is the same as the work in the Visibility dialog, located in the "Layout" drop-down menu (see Chapter 2.10), and we will not explain it again.



Selecting this icon opens a dialog where you can adjust the characteristics of the blocks that are generated. It also records configurations with different characteristics settings. Configurations that are not recorded in the database will appear at the end of the closed list with the configurations, and in addition to their name, a sign(\*) will be displayed. If necessary, in this dialog they can be saved to the database for later use in the program. The dialog is the same as the dialog that opens with the right mouse click through the "Export the block" icon in the results processing module (see chapter 8.24.1), and we will not re-explain it.

## 8.20 Removing the Presented Influences from the Screen Instantly (Reset)

Since the program allows memory of the specified sections during the recording the model, the user is enabled to delete them in a quick and easy way. Right-clicking on the "Reset" command opens a dialog of the following layout:



The right click of the mouse over the "Reset" command opens the dialog

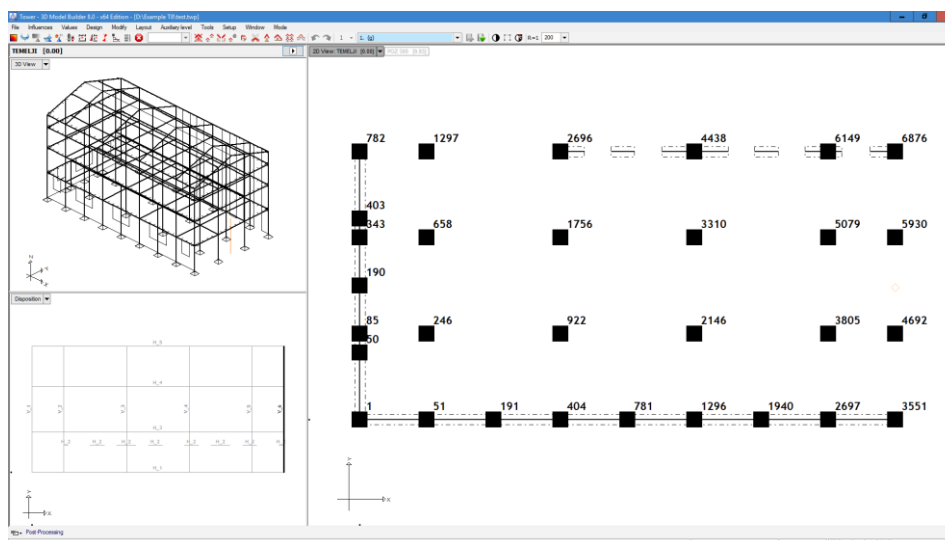
The dialog shows all types of sections that can be found in the model and can be deleted by the user. In the case that some kind of section does not exist in the model, the appropriate check box will be inactive.

The status of the check box that is located in front of the name of the section type determines whether or not to delete an assigned section. By setting the check box to the on condition and by going to the "**Accept**" button, the appropriate section type will be deleted from the model.

Selecting the "**All on**" and "**All off**" command fields, the user has the ability to put all checkboxes on or off at the same time.

## 8.21 Numbering the Finite Element Mesh Nodes

It is possible to display the numbers of the nodes of the finite element network at points of point supports when they are located outside the slab or beam.



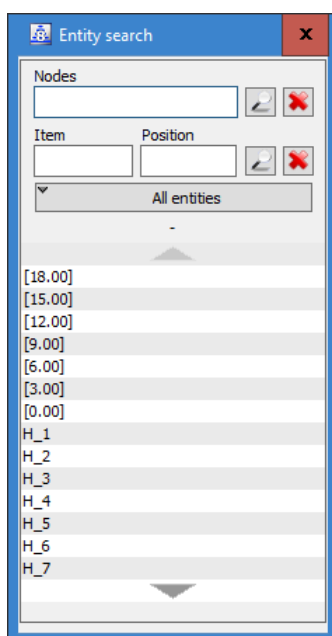
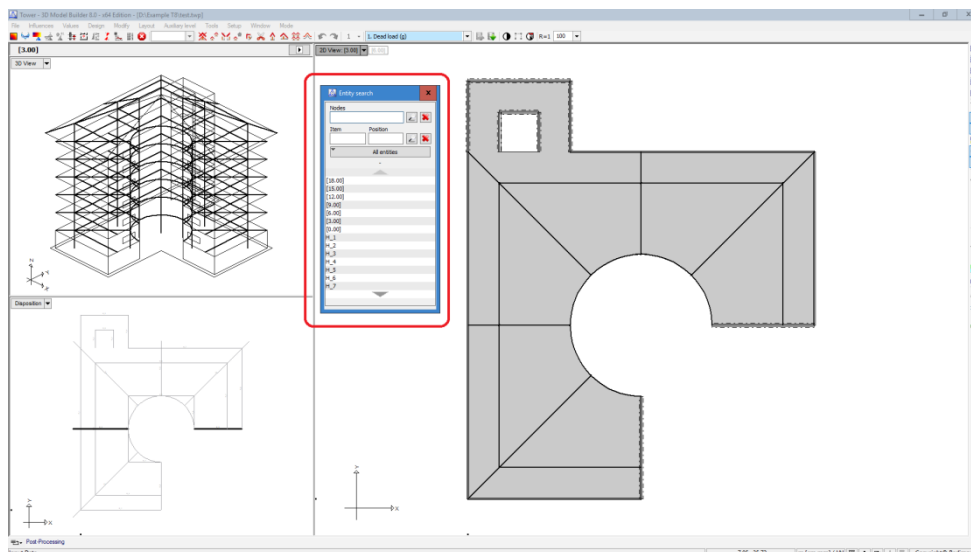
## 8.22 Entity search

Using this command, you can find any node from the finite element network in the model. Apart from the fact that the requested node is marked with a special symbol, a list of all the assemblies which it belongs to, with the possibility that with one click of the mouse on any of them will be selected for view in the 2D window. The command is most useful in controlling results and creating textual reports that refer to node numbers from the finite element network. It may be required to find one or two nodes simultaneously. As the beams in the textual reports are marked with the numbers of nodes of the starting and ending points, the search for these two nodes simultaneously finds the assemblies to which the beam belongs, and by choosing one of them it is very easy to display the required beam on the screen.

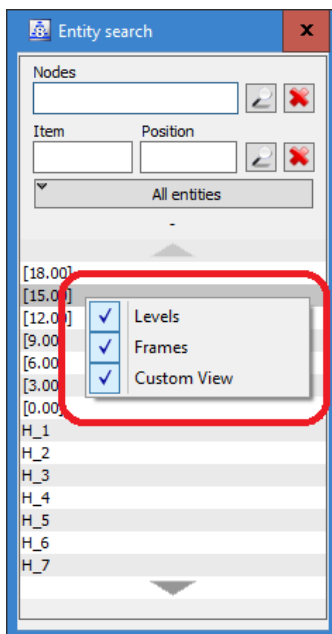
Apart from finding nodes, this command can also be used to find entities in the model, via tags and position numbers, of course, if they were previously marked.

The command is in the "Layout" menu, in the "Data Input" and "Post-Processing" modules. Finding an entity through position markings is always available, while finding nodes is only available in the results processing module if a static or multi-modal analysis is performed, because the finite element network gets its final form only when one of these two analysis is implemented.

After activating the command, the "floating" dialog opens with the following layout:



In the dialog, a list of the names of all assemblies in the model is displayed, except for the complex ones. Right-clicking the mouse over the list opens a drop-down menu that selects the types of assemblies that will be displayed in the list.



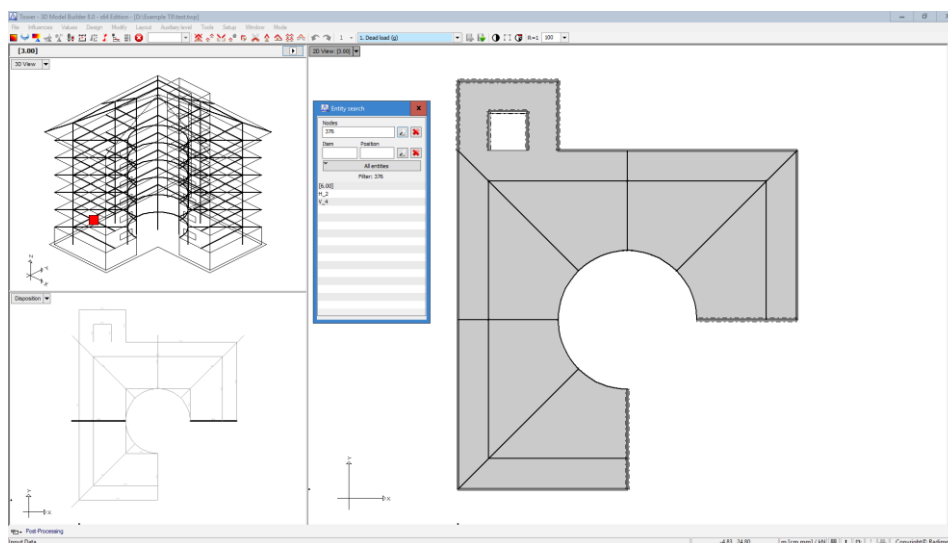
The drop-down menu for selecting the type of the assemblies

### Finding the nodes

**Nodes** Edit field to enter the number of the requested node. It may be required to find one or two nodes simultaneously. If two nodes are required, their numbers in the edit field must be divided by some character (signs of interruption, space, letters, ...).

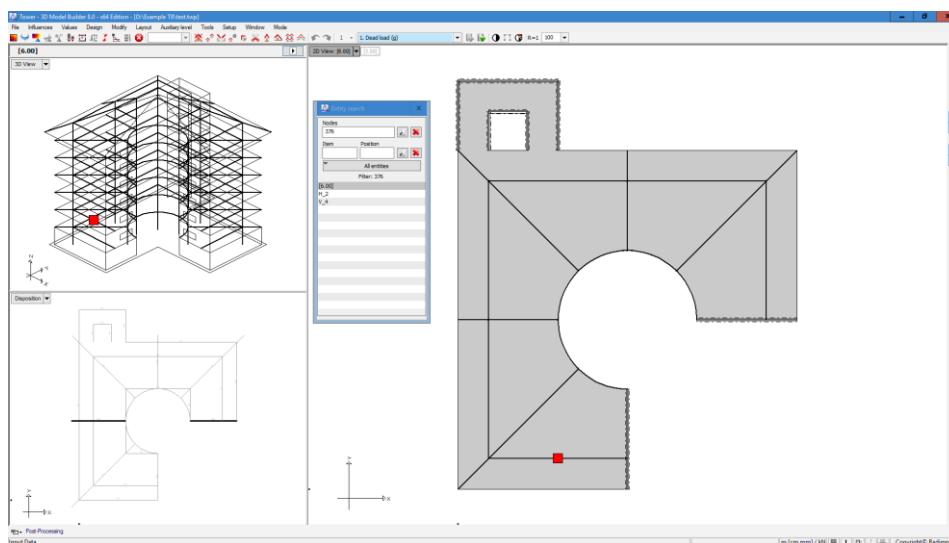


By choosing this button, the program marks the requested node with a special symbol, and the list only shows the names of the assemblies that the requested node belongs to. If the requested node does not belong to any sets, this list will be empty. In the first row, above the list of assembly names, the default filter is printed, so that at any time, you can see which nodes are specifically marked in the drawing.



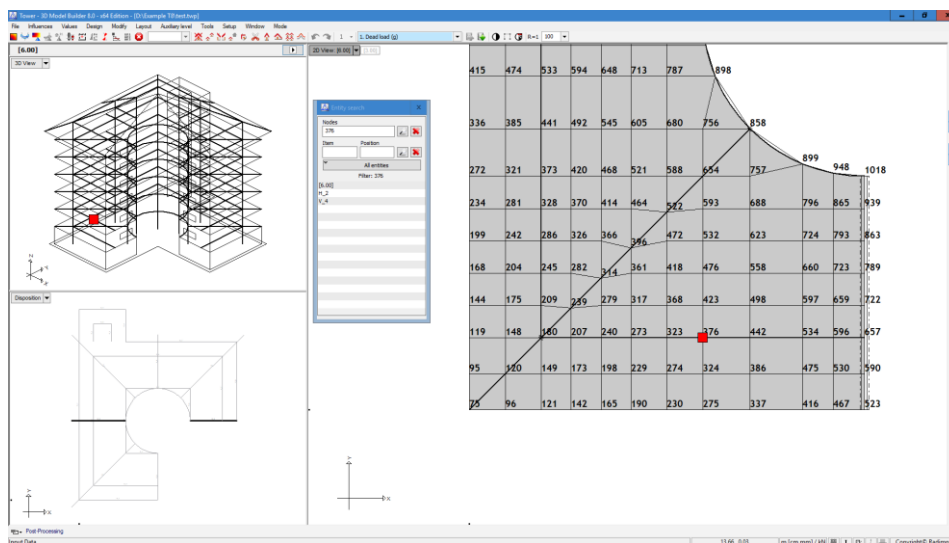
It was required to find a node with the number 376 so that it was marked in the model with a red box, and the list shows only the assemblies to which it belongs

If you click the mouse over the name of one of the assemblies in the list, it will be displayed in the 2D view window, and the requested node will be marked with the predefined symbol in this layout too.



Clicking the mouse over its name in the list, level 6.00 is selected to display in the 2D view window

In the "Visibility" dialog, the display of the finite element network and the printing of node numbers can be turned on. In this way it can be checked whether the number is marked on the drawing which search was requested.



Required node with shown net of finite elements and node numbers



Choosing this button, you can cancel the specified filter, or remove the nodes from the field edit box.

### Finding entities by the item marks

#### **Item description**

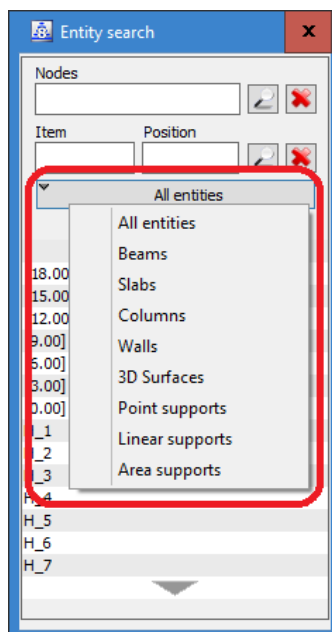
Edit the field where the description of the position of the requested entity is entered.

#### **Position number**

Edit field to enter the ordinal position number of the requested entity. If the number of the position is not assigned in this edit field, then the program will search the entities only according to the given description of the position.

## All entities

Clicking the button, which is located below the edit field for entering the description and the ordinal position number, a drop-down menu of names of all entities that can be found using position tags opens.

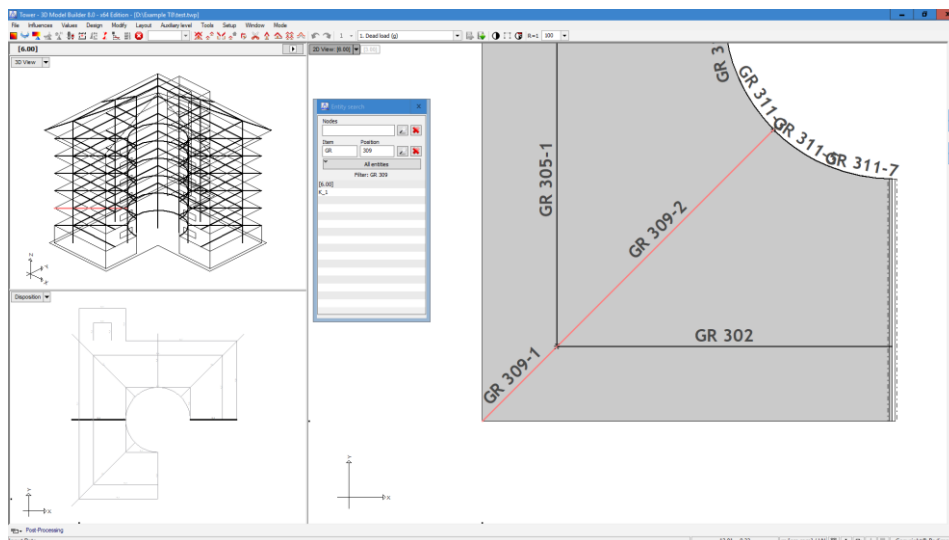


Drop-down menu for selecting the entity

If only one entity is selected from the menu, the specified search will only be performed for that entity, and its name will be printed on the button. If the "All Entities" option is selected, the search for position tags will be applied to all entities, and the name of the selected option will be printed on the button.



Selecting this button, the program of the found entities is marked with a special colour, and the list only shows the names of the assemblies to which the requested entities belong. If you click the mouse over the name of one of the assemblies in the list, it will be displayed in the 2D view window, and the found entities will be marked with the colour specified in this layout. If the found entities do not belong to any sets, this list will be empty. In the row above the list with assembly names, the filter is printed, so you can at any time see the sign and the number of entity positions that are specifically marked in the drawing.



In the model are found and marked all entities with the position sign GR and ordinal number 309



Choosing this button, the specified filter is canceled, that is, the description and position number from the edit field is removed.

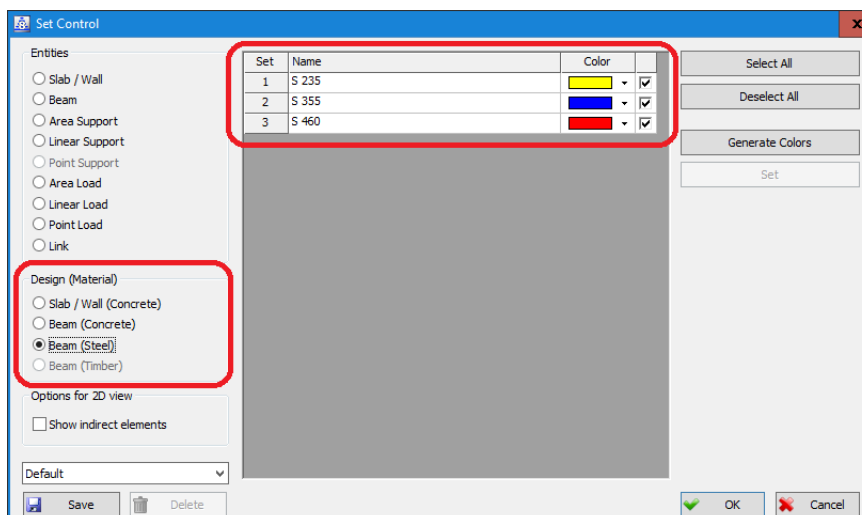
The "Entity search" command is transparent, which means it can also be triggered when another command is active. Also, any command can be started when the dialog of this command is active, which means that its display on the screen does not affect the operation of the program.

## 8.23 Set Control

When the "Set Control" command is run in the result processing module, switches in the "Design (Material)" section are available in the dialog that opens with the "Select" option, by which entities can be sorted and displayed according to the dimensioning material used. The following options are offered:

- Slab / Wall (Concrete)
- Beam (Concrete)
- Beam (Steel)
- Beam (Timber)

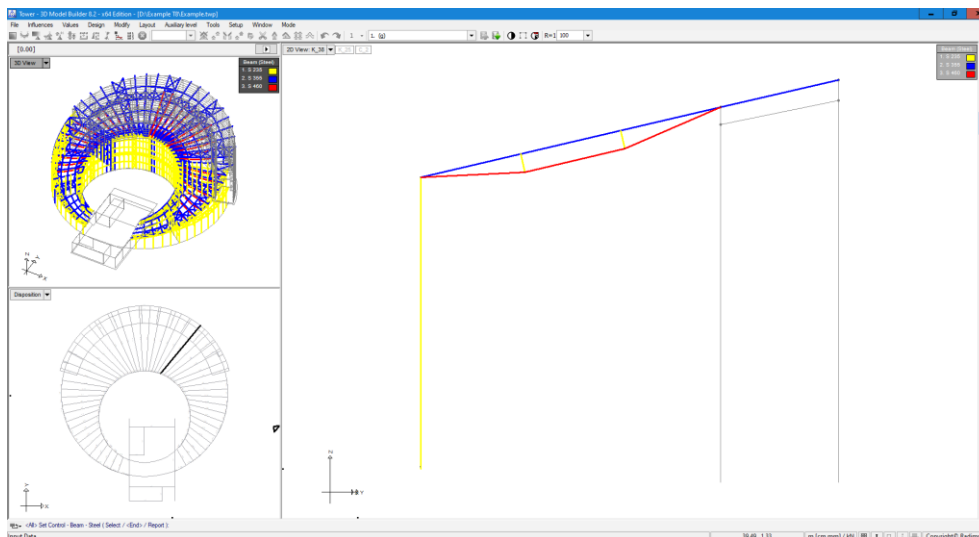
If some material is not used in a given model, the corresponding item will be unavailable.



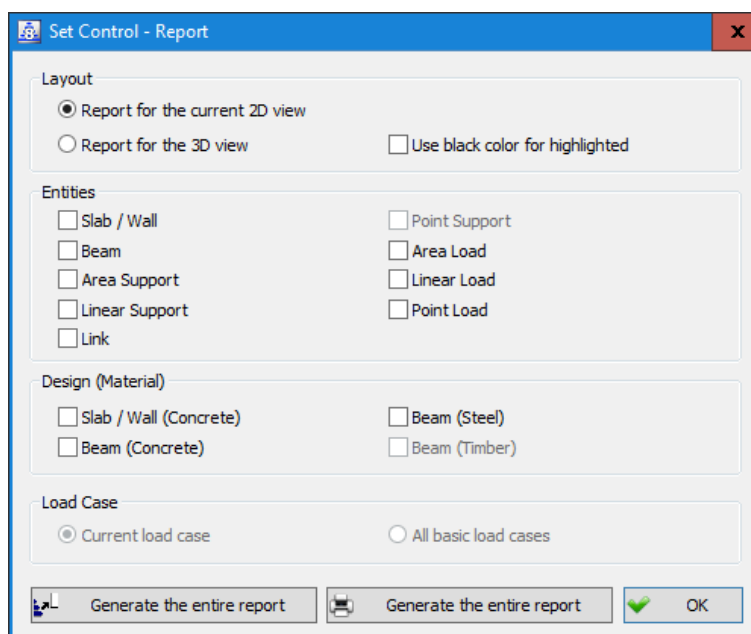
Control of sets for steel beams was selected

As the entities are grouped according to the material, the column names for concrete all used concrete grades, while for steel and wood all types of these materials are used.

After exiting the dialog, all entities that are of the selected material, colored with the given colors for each of the types used, are displayed on the screen.



Steel beams are shown with colors from the control of the sets



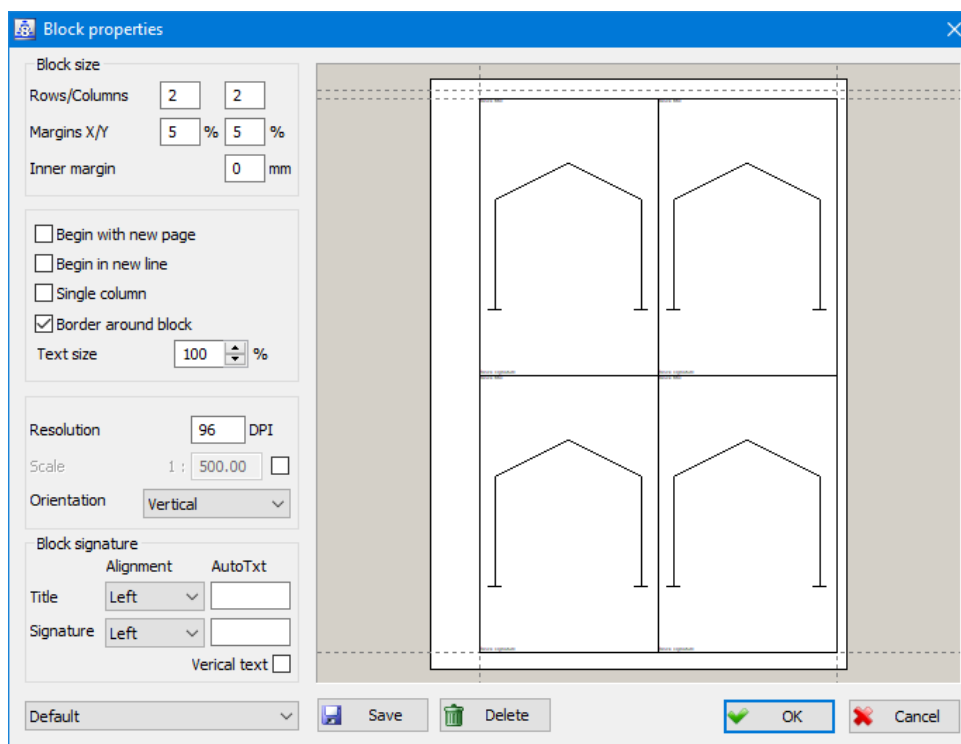
Dialogue box for generating the report of the control of the sets

The section "**Design (Material)**" contains check boxes, the inclusion of which determines for which materials the set control report is generated.

## 8.24 Creating the Design Documentation

### 8.24.1 Export of Graphic Blocks

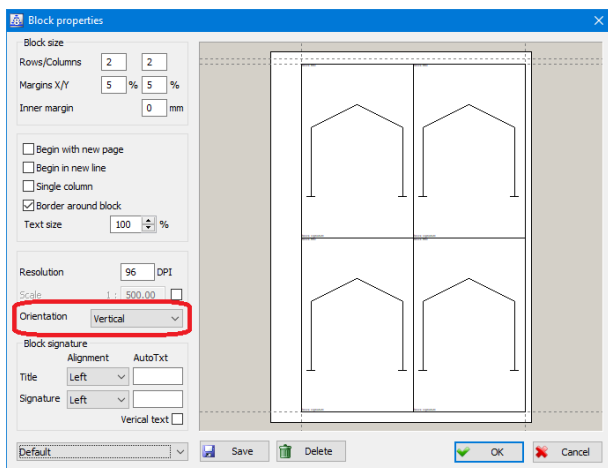
#### Default Characteristics of Blocks to be Exported



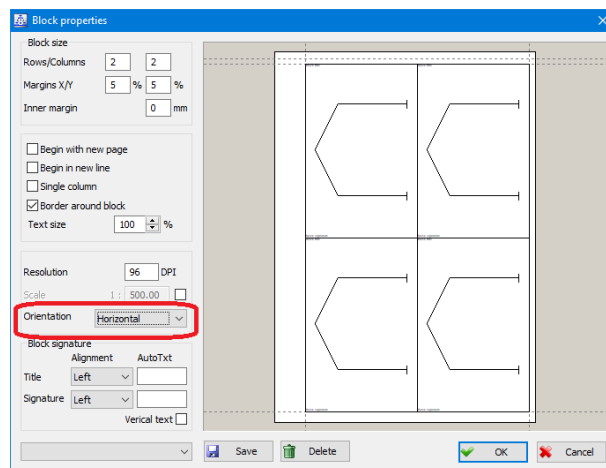
Layout of the dialog for adjusting "default" characteristic of blocks that will be exported

**Text size** Edit box for setting the scaling factor of texts in graphic blocks. In this way, the program allows additional increasing or decreasing of the font size in relation to the size that is set in the parameters.

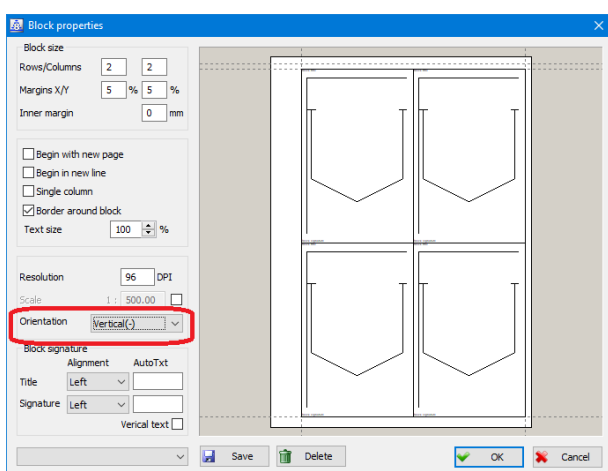
From the closed list "Orientation" one of the four offered block orientations is selected: "**Vertical**", "**Horizontal**", "**Vertical(-)**", "**Horizontal(-)**", where the block rotates by 0°, 90°, 180° and 270°.



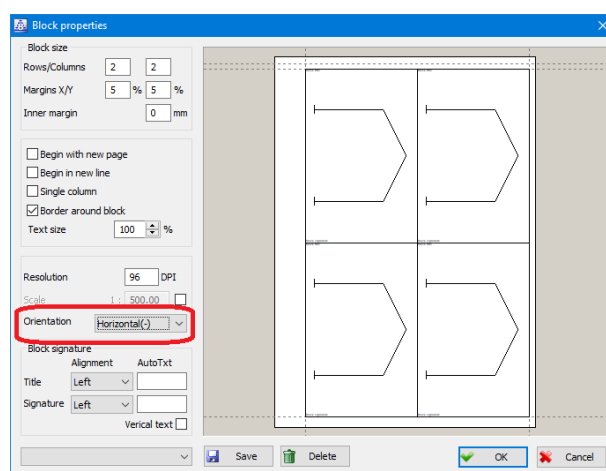
"Vertical" orientation selected



"Horizontal" orientation selected





"Vertical(-)" orientation selected



"Horizontal(-)" orientation selected

The closed list in the lower left corner of the dialog is intended for selecting some of the previously recorded configuration of block characteristics.

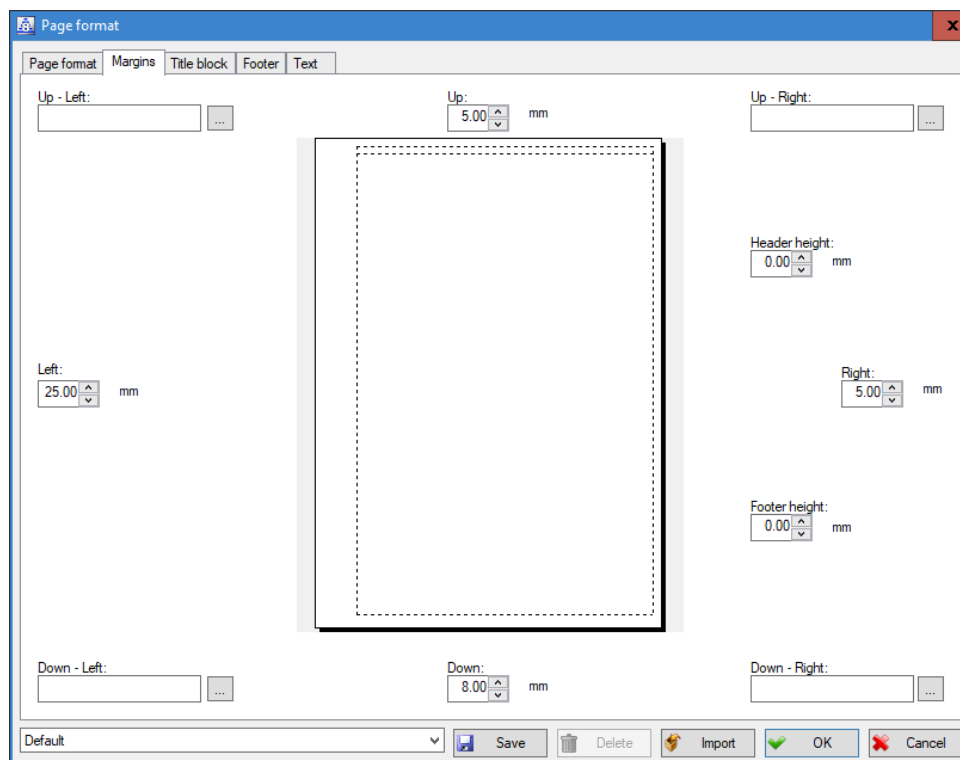
 **Save** Selecting this button, you are saving the current state of the parameters in the dialog.

 **Delete** Selecting this button, you are deleting the current configuration.

### 8.24.3 Defining the Paper Format (Page format)

- A new configuration of report parameters was created and included in the installation: "Standard".

#### Margin defining



Layout of the "Margin" file

#### Header height

In case each page of your document should contain a header, it is necessary to specify its height in the assigned edit box. If you do not want to set a header, enter the value "0" in the edit field.

#### Footer height

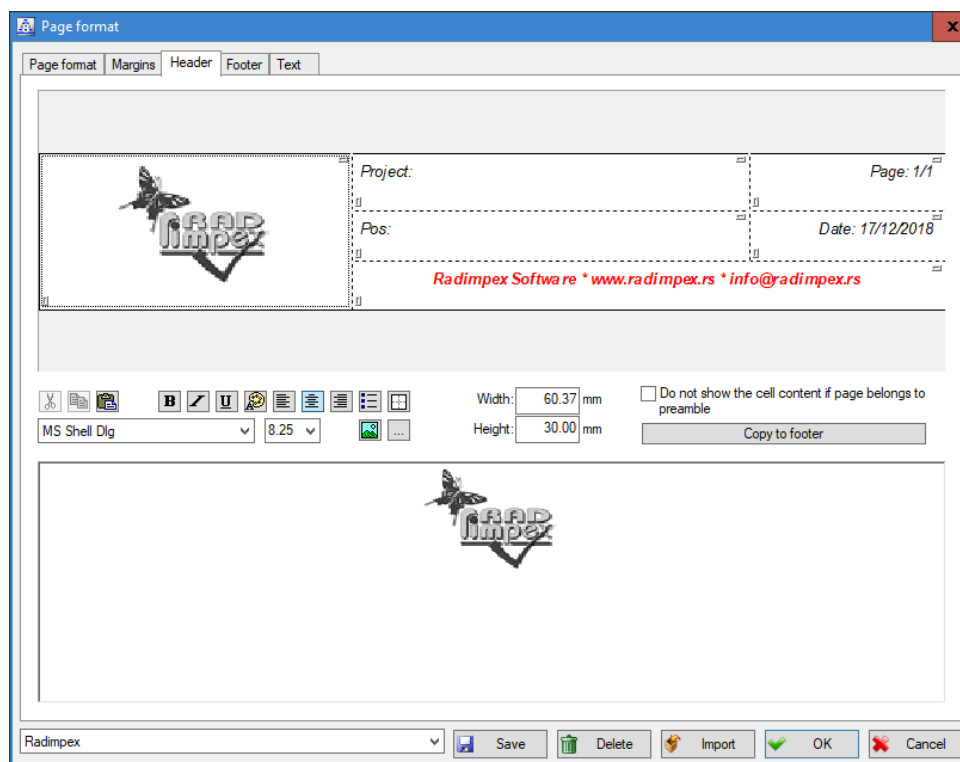
If each page of your document should contain the footer that appears at the bottom of the page, it is necessary to specify its height in the set edit box. If you do not want to set the footer, enter the value "0" in the edit field.



The program-defined texts can be automatically inserted into the edit boxes for text input, as well as the texts that are assigned in the "Project Description" dialog, located in the "File" drop-down menu. By selecting the icon, which is on the right side of each of the previously described edit boxes, the drop-down menu opens with the available texts:

- #c** - Project (Project Description)
- #b** - Structure (Project Description)
- #m** - Location (Project Description)
- #k** - Client (Project Description)
- #j** - Engineer (Project Description)

## Defining the layout of the header



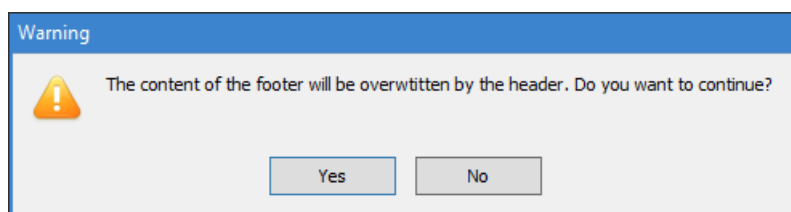
Layout of the "Header" tab

By selecting this icon, a drop-down menu opens to automatically load the program-defined texts and texts that are specified in the "Project Description" dialog box, located in the "File" drop-down menu. The following texts are available:

- #c - Project (Project Description)
- #b - Structure (Project Description)
- #m - Location (Project Description)
- #k - Client (Project Description)
- #j - Engineer (Project Description)

### Copy to footer

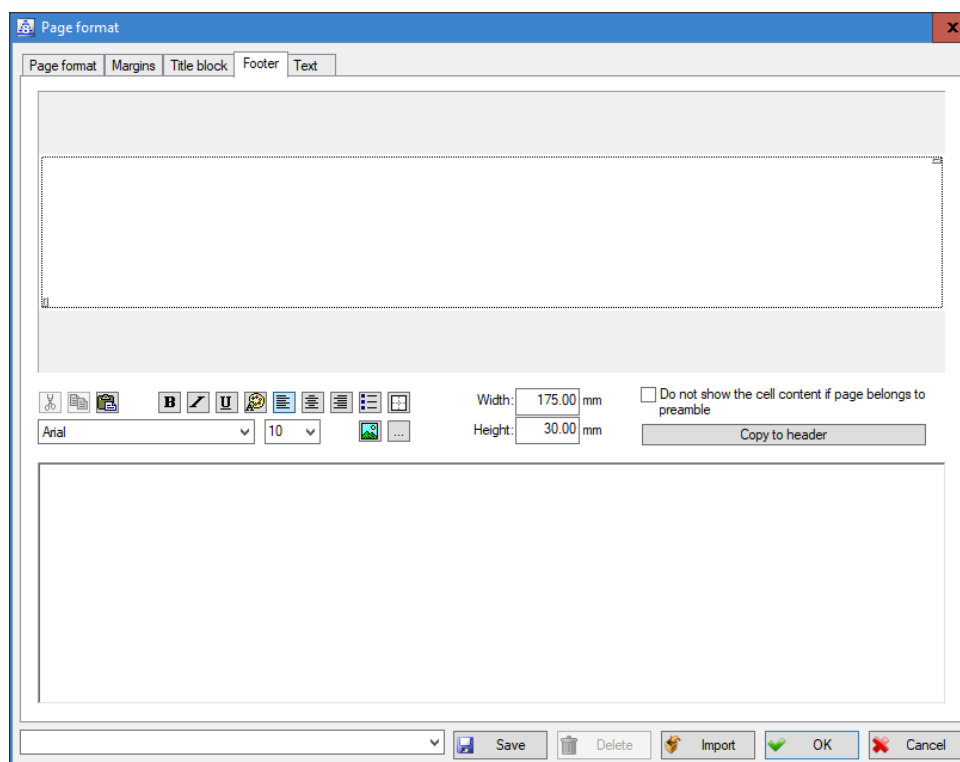
Since the document next to the header can have a footer, by selecting this button in a quick and easy way the contents of the header are copied to the footer. By activating the button, the following message appears:



Selecting the "Yes" button the content of the footer will be replaced by the current header

## Defining the footer layout

Within the "Footer" tab, the layout of the footers is defined, which will be printed at the bottom of each page of the document.



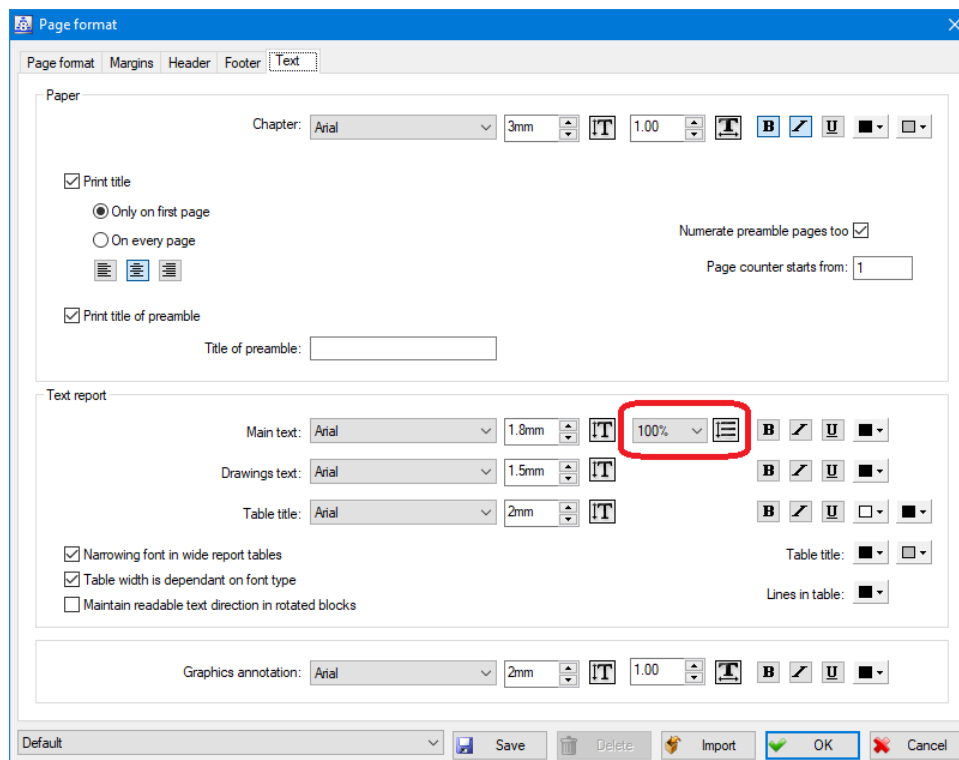
The layout of the "Footer" tab

Operating in this dialog is entirely appropriate for working in the header dialog, and we will not explain it again.

## Defining the texts on the paper



From the list in front of this icon, the size of the space between the lines in the report can be altered.



Drop List for adjustin the spacing between report lines

### Narrowing font in wide report tables

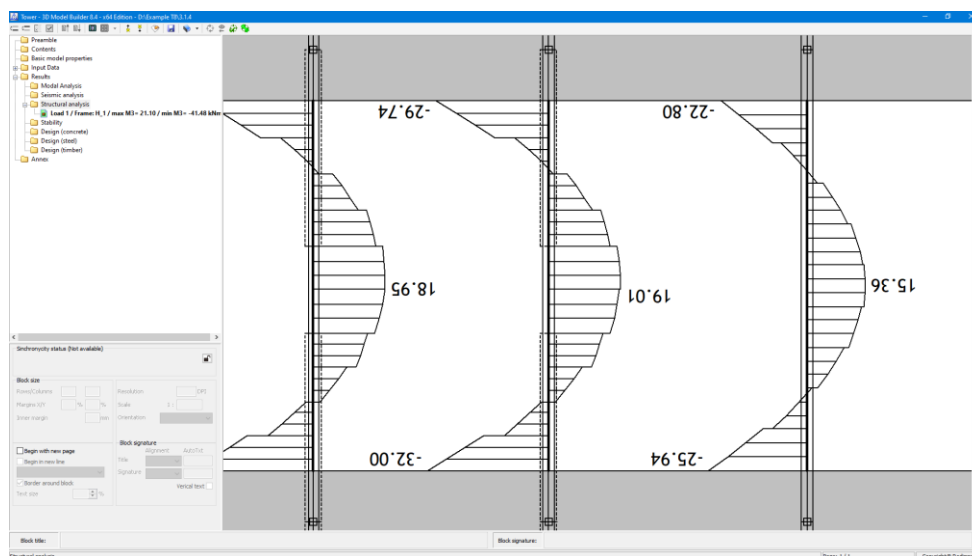
Wide textual reports limit the font size selection of the entire report, because they disable the page breaking. By placing this check box on, the program automatically compresses the fonts in wide spreadsheets to avoid the need for all reports to reduce the font size. In this way, it is possible to use fonts of a larger range of sizes to make reports than was previously the case.

### Table width is dependant on font type

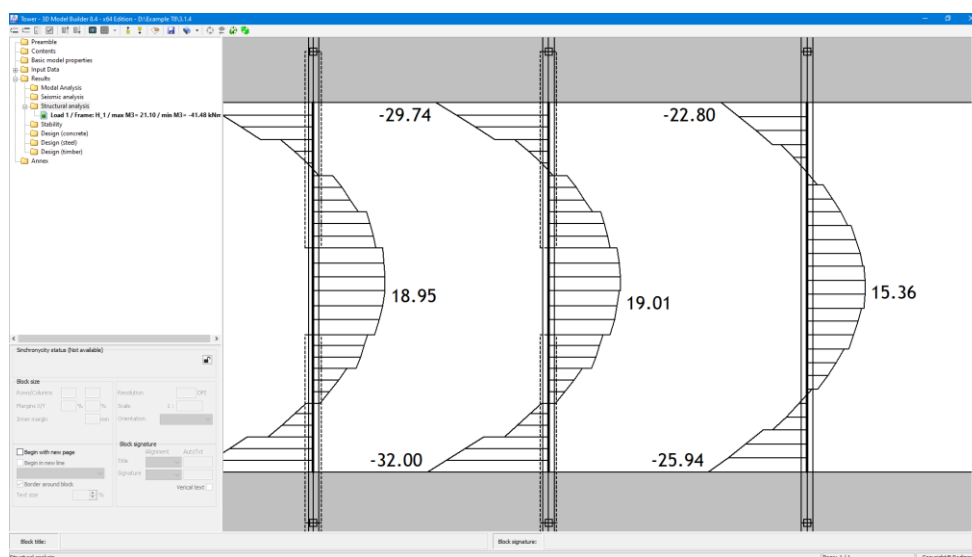
Check box with the setting on the enabled state of the program performs smarter calculation of the width of the table in the text report in the function of the selected font.

### Maintain readable text direction in rotated blocks

By rotating the blocks, the texts in them are not rotated, but the user can automatically change the orientation of the texts by turning on this check box, so that their readability is preserved.



Orientation of texts in rotated blocks when the check box is off  
 “Maintain readable text direction in rotated blocks”



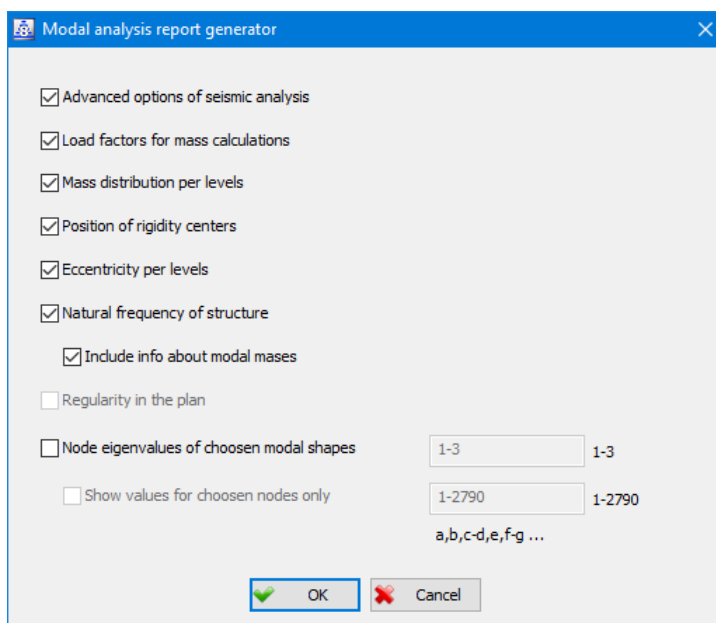
Orientation of texts in rotated blocks when the check box is on  
 “Maintain readable text direction in rotated blocks”

### Lines in table

Command field for selecting the colour of the table lines.

## 8.24.5 Generating the Textual Reports

### Generating a text report on the conducted modal analysis



Dialog layout for generating modal analysis reports

**Include info about modal masses** - if this check box is included, in the table "Natural frequency of structure" a column with the modal masses of each tone will be displayed, while in the table "Node eigenvalues of chosen modal shapes" for all selected tones the columns with normalized rotation vectors and masses will be displayed in each selected node of the structure. Therefore, this check box will be available only if at least one of these two tables is included for generation.

Modal Analysis						
Natural frequency of structure						
No	T [s]	f [Hz]				m [T]
1	0.9714	1.0295				344.2540
2	0.7742	1.2916				559.4858
3	0.4638	2.1562				569.2897
4	0.2158	4.6334				284.1910
5	0.1604	6.2340				64.8606
6	0.1472	6.7957				42.0608

Modal shape (Mode 1)							
Node	xp	yp	zp	xr	yr	zr	m [T]
100	0.000000	0.000000	0.007582	-0.002378	0.003614	0.000000	0.000000
101	-0.000000	0.000000	0.008611	-0.002064	0.003623	-0.000000	0.000000
102	-0.000000	-0.000000	0.006332	-0.002755	0.003473	-0.000000	0.000000
103	0.007183	0.002355	0.012138	-0.000308	0.003309	0.000969	0.000000

New columns in the tables when the "Include info about modal masses" check box is turned on

In discrete systems, the oscillation forms are vectors, and the modal mass corresponding to the oscillation form can be calculated by the following equation:

$$m' = \psi^T M \psi,$$

where  $\psi$  is the (normalized) form of oscillation and  $M$  is the mass matrix.

Modal mass can be understood as the total mass that is activated in a certain mode shape. A higher value of the modal mass indicates a greater importance of a certain modal shape.

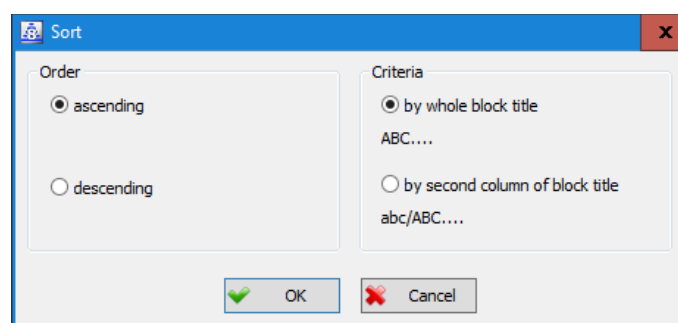
Unlike the modal mass, which is a property of the form of oscillation, the data "percentage of mass engagement" is calculated in relation to the reference direction of seismic action.

## 8.24.6 Commands for Operating with Blocks in Tree

### Sorting blocks within project documentation

The drop-down menu, which opens by right-clicking on a chapter title or selecting multiple blocks in a trunk, contains a command that sorts blocks. In case the command is activated when selecting a chapter title, all blocks within the selected title will be sorted, while in the case of multiple block selection, only the selected blocks will be sorted.

After running the command, a dialog of the following appearance opens:



The layout of the block sort dialog

To arrange the block layout in order:

**ascending**      Blocks will be sorted in ascending order.

**descending**      Blocks will be sorted in descending order.

Block Editing Criteria:

#### **by whole block title**

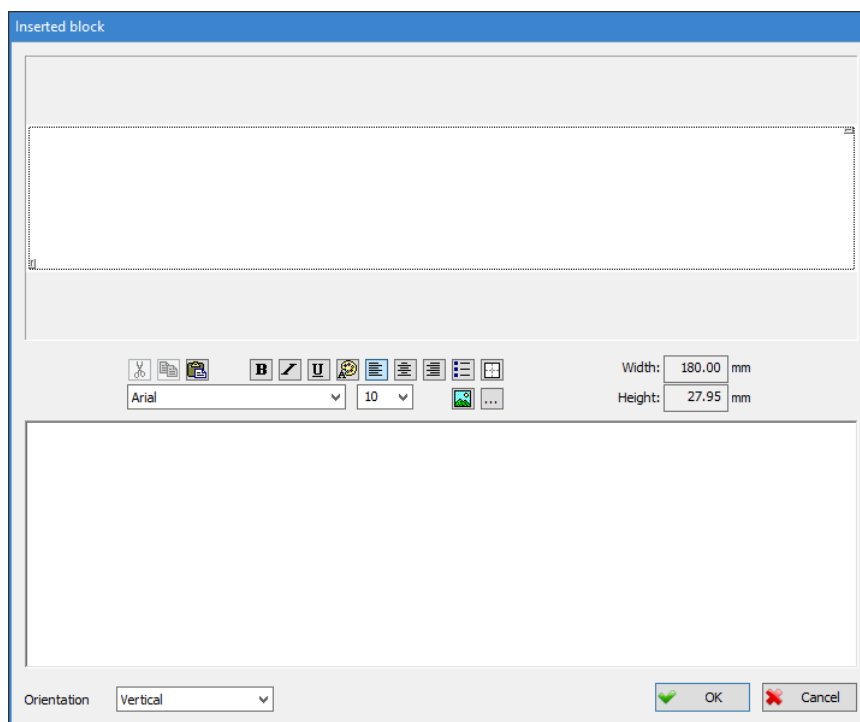
Blocks will be sorted by their name, in the order they have been selected.

#### **by second column of block title abc/ABC...**

The blocks will be sorted by another column of names, in the order selected. Columns are considered to be parts of the block name separated by the symbol "/".

As described above, blocks can only be sorted within the same chapter heading.

## TXT blocks within the project documentation



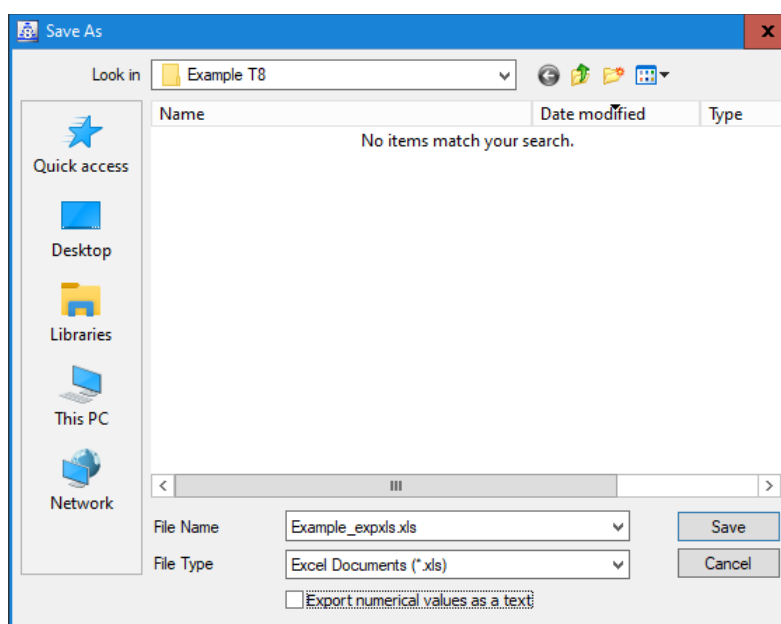
The layout of the dialog where the content of the inserted block is entered

From the closed list **"Orientation"** the default orientation of the inserted block is selected: **"Vertical"**, **"Horizontal"**, **"Vertical(-)"** or **"Horizontal(-)"**.

## Exporting block to Excel



Selecting this icon, the dialog for exporting the selected text block to the XLS format opens.



**Export numerical values as a text** - Check box that condition determines whether numeric values will be represented as text or as numeric values.

Now you need to specify the desired file name and, by activating the **"Save"** command, make it capture in the selected directory.

You can export all text messages generated in the program into a separate file with the extension **"\* .xls"**.

### 8.24.18 Exporting the block to the clipboard



Exporting a block to the **"clipboard"** can also be done from the paper editor, by activating the displayed icon. This command works in the same way as the command contained in the results processing module, except that instead of the contents of the window, the selected graphic block is exported to the clipboard. Exporting the selected block to the **"clipboard"** can be done by simultaneously pressing the **"Ctrl"** and **"C"** keys.

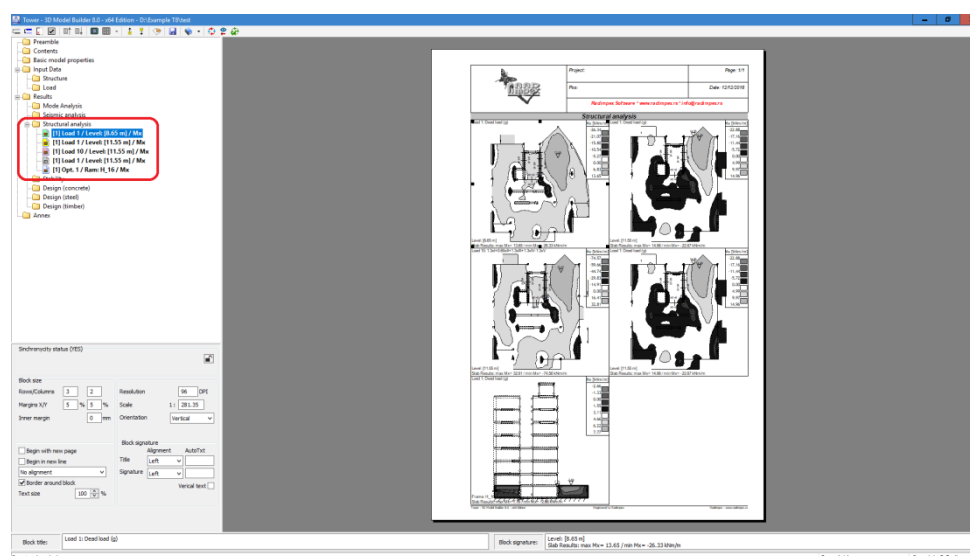
### 8.24.23 Automated update of the report

In designing, there is often a need for subsequent modifications in the model for which the project documentation has already been made partly or entirely. Modifications to the model can be different: geometric changes (deletion and addition of structural elements, contour change, adding holes,...), changes in characteristics (material change, dimension of cross sections, stiffness of supports,...), load change (intensity change, and adding load cases), etc. Each of these changes practically invalidates the validity of almost all the previously made reports where you invested a lot of time and efforts. For this reason, a command is provided enabling automatic updating of the already created report, so that all existing blocks in the report are automatically matched with the current state of the model.

The program associates each block when it is created with data that will serve for:









- monitoring the condition of the block's compliance with the current situation in the model.
- enable the refresh of the block according to the current state of the model, and in everything according to its previous content (to show the same layout, with the same visibility, the same kind of influence,...).
- interaction of reports and models, i.e. the ability to get directly from the report directly to that location and to that state in the results processing module, which corresponds to the status and display of the block.

Each block can have one of the 5 states that are clearly visible in the report tree:

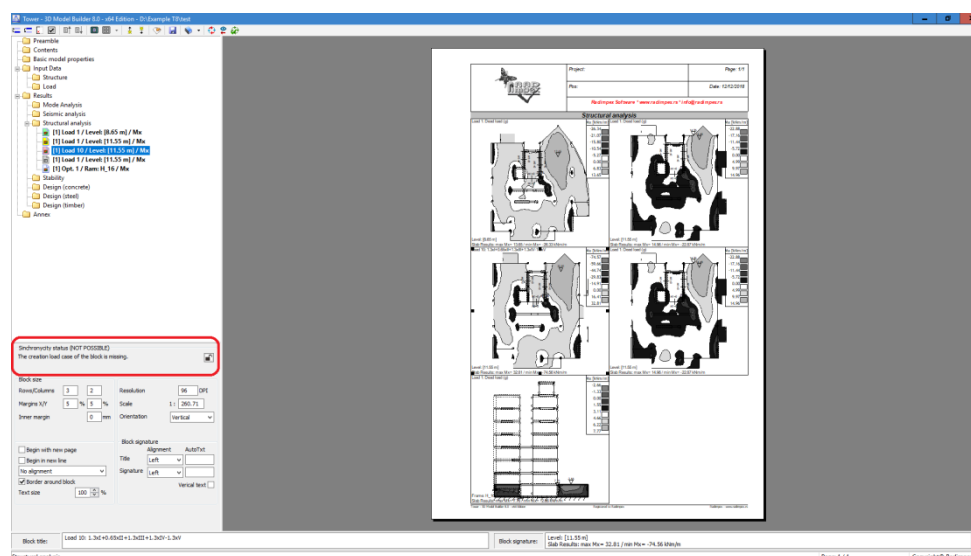


Icons in front of the block indicating the state of matching the block with the model

The colour of the icon, located in front of each block in the report, shows the status of the compliance of the block with the model:

-  The block matches the current situation in the model.
-  The block does not match the current state of the model, but can be refreshed. If the block is selected, the cause of the mismatch is also displayed in the part of the screen below the report tree.
-  The block does not match the current state of the model and can not be refreshed. The reason why refreshment is not possible is printed in the part of the screen below the report tree.
-  The block is locked, which means that the user locked the block and therefore excluded it from the synchronization process.
-  The block does not contain the data necessary for its alignment with the model. This is the case, for example, with the blocks made in one of the earlier versions of Tower or blocks imported from another model.
-  The button to lock the selected block to match the model. The button icon indicates whether the block is locked or not:
  -  - Selected block is unlocked
  -  - Selected block is locked

The information about the updating of the selected block is printed in the part of the screen below the report tree:

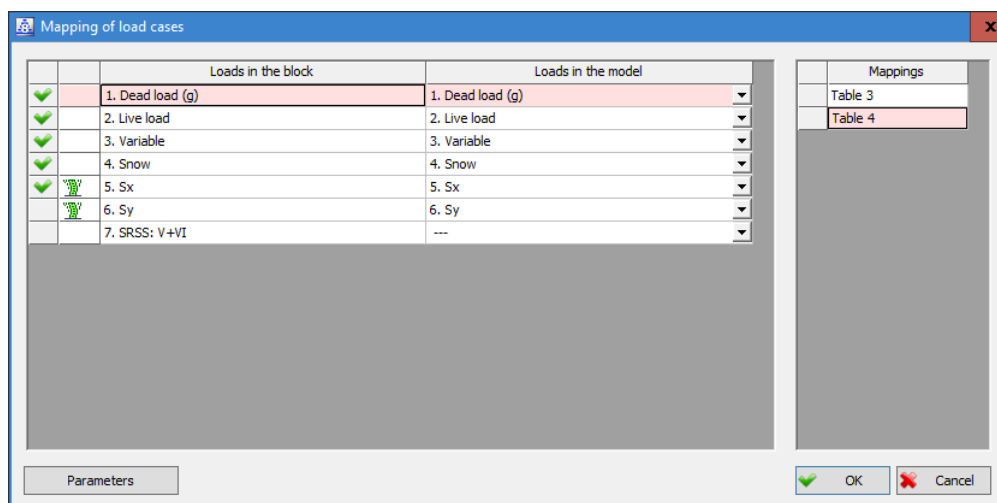


Part of the dialog where the data on the compliance of the block is printed

When entering the editor of the paper, the program executes only a quick check of the compliance of the blocks. The first consistency checking, which is a complex and slow operation, is executed by running a special command.



Selecting this icon, you run the **“Check of blocks refresh status”** command, after which the following dialog opens:

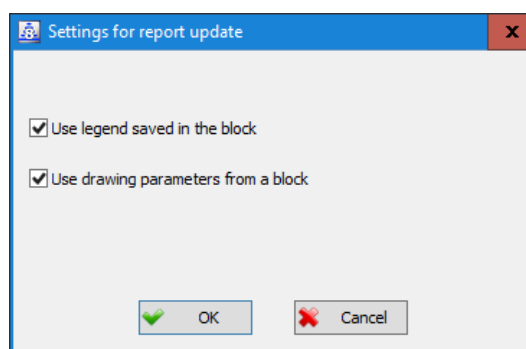


Dialog for mapping cases of loads

If in the meantime there has been a change in load cases (the number of load cases), it will be necessary to reconfigure cases of load of an unbalanced block into current load cases in the model. The procedure takes place by clicking on the arrow "▾", located at the end of each row, to open the drop-down list and select a load case on the model in which you want to place the load case from the block. The displayed list has as many lines as there were cases of load in the model at the moment of creating blocks. The "✓" symbol that can be found in front of the load case name from the block indicates that this load case is used in the block and that this case should be placed in the appropriate case of load in the model. Since the blocks in the report could be made with different load cases, it is necessary for each of these blocks to map the load cases. In the "Mappings" table, the user can edit the status in the left table by selecting the appropriate mapping table. One report can contain blocks from multiple time periods in the past with different states of number and character cases of load and the "Mappings" list refers to each different situation of load cases.

### Parameters

The activation button opens the dialog of the following layout:

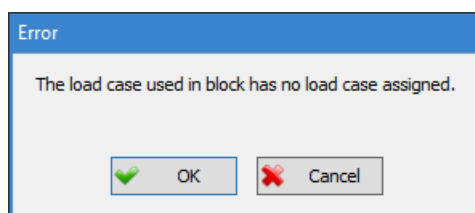


Using the check box "**Use legend saved in the block**", the user is enabled to update the block, determining whether the block will take over the number of divisions from the legend model or from the block.

**Use drawing parameters from a block** - When this check box is turned on, the blocks are refreshed with the same drawing parameters with which they were exported. When off, blocks are refreshed with the parameters currently set in the "Settings" command dialog, under "Paper Settings".


By exiting the "Mapping of load cases" dialog on the "OK" button, the program closes the load case mapping dialog and, with respect to all the specified parameters, performs a check of the consistency of the blocks with the current situation in the model.

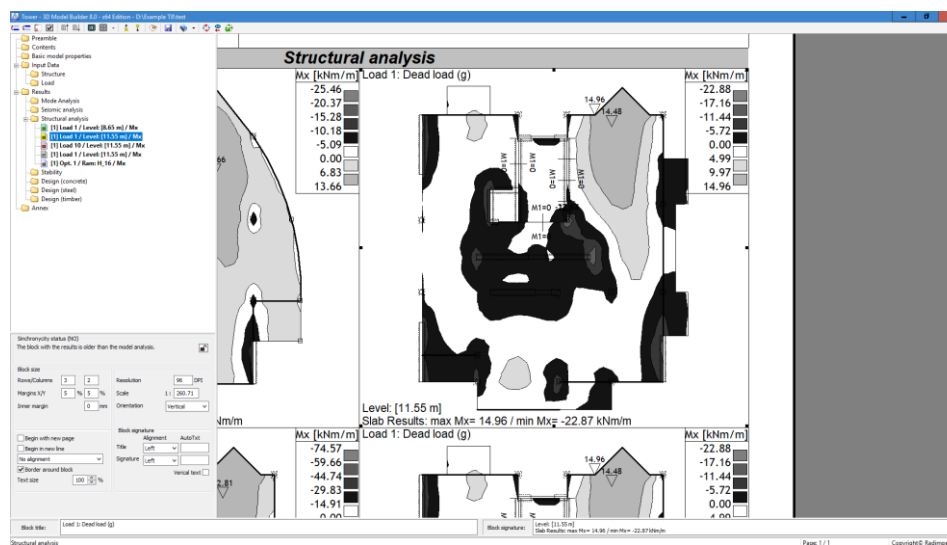
We will also note that if the user did not place the load case from the block in the appropriate load case in the model, the program displays the corresponding message:



### Update of selected blocks

Considering that after every modification in the model (deletion and addition of structural elements, change of contour, change of material and dimensions of sections, deletion and addition of load cases, change in load intensity, ...), most of the previously created blocks no longer correspond to the state of the model, the program allows the existing blocks from the project documentation to automatically align with the model.

It is necessary that the user first selects an arbitrary number of blocks in the tree of the report, which does not match the current state of the model. All unmodified blocks have in front of their name in the tree of the report  icon, while the part of the screen below the report tree shows the reason for their mismatch.

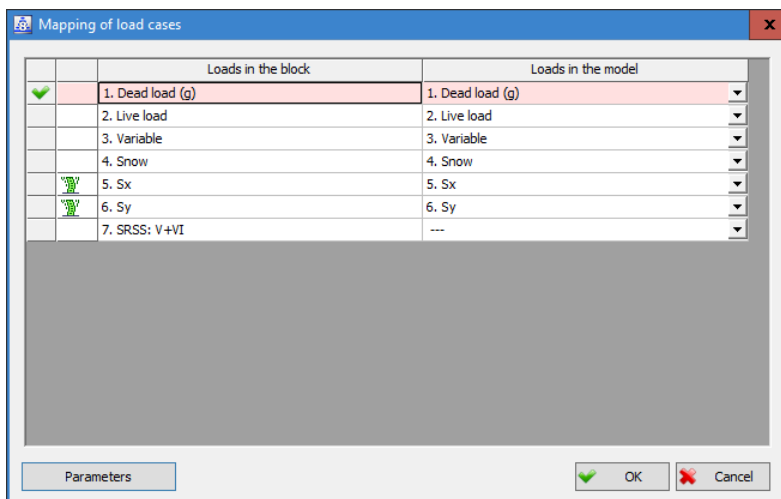


The selected block is incompatible with the current situation in the model




Selecting this icon or the **“Update of selected blocks”** command from the drop-down menu that opens right mouse click over the block name in the report tree, it is possible to automatically align the assigned blocks from the project documentation with the current state of the model.

If the selected block contains the results of the static analysis, after activating the “Update of selected blocks” command, the following dialog will be opened:

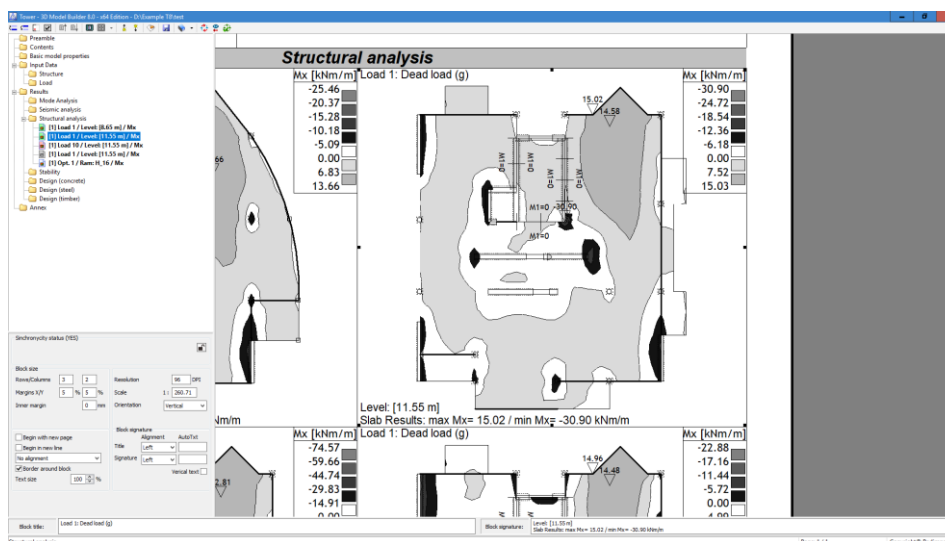


Dialog for mapping load cases

Operating in this dialog has been explained earlier, and we will not explain it again.

Exiting from this dialog by the "OK" button, the program executes the automatic alignment of the assigned block with the current state in the model, and with the name of the block in the tree of the report, the icon  now appears.

Note that if the selected blocks did not contain the results of a static analysis, the load mapping dialog will not open. Also, if there was no change in the number and type of load between the two analysis, the program will automatically adopt the default load mapping.

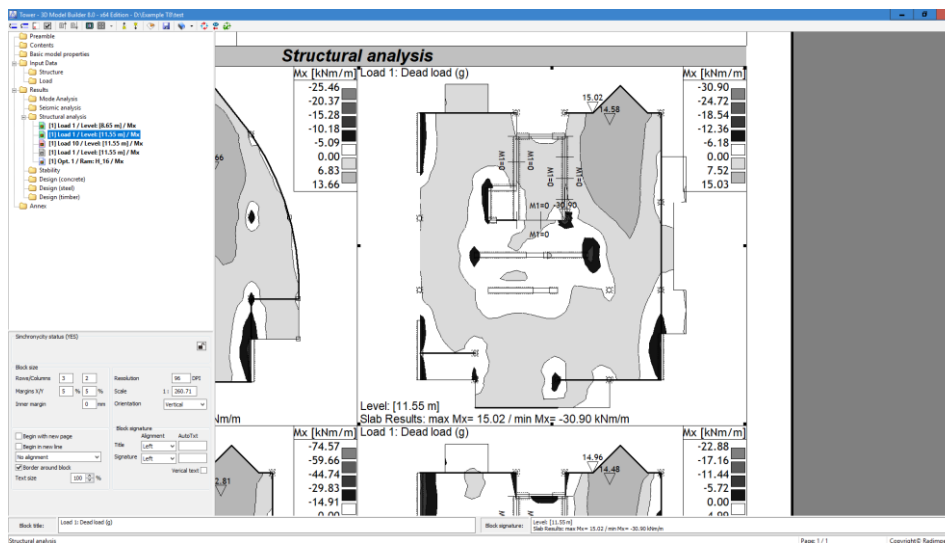


The layout of the selected block after the command "Update of selected blocks"

### Original display of the current block

The program is enabled to go from the report directly to the same location and to that state in the results processing module, which corresponds to the status and layout of the selected block.

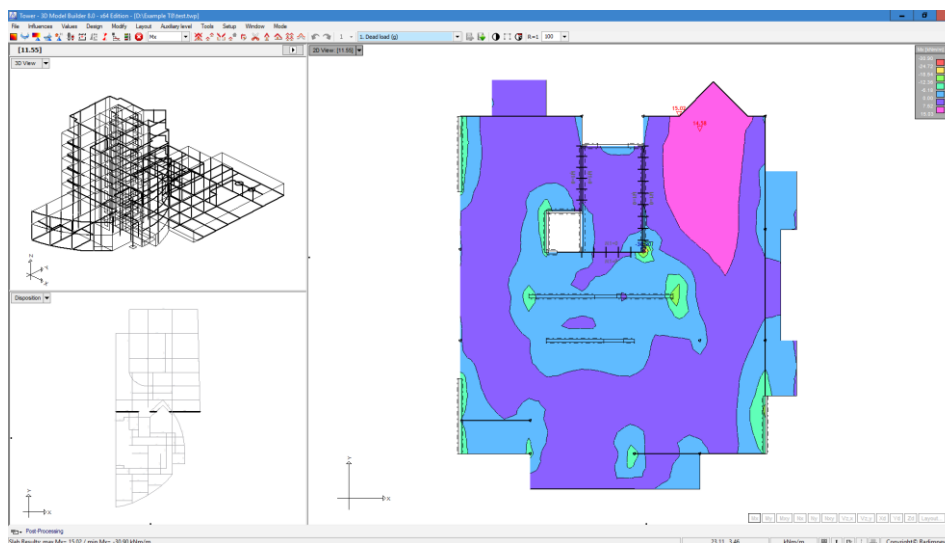
It is necessary first select the desired block in the paper editor:



The layout of the static analysis result



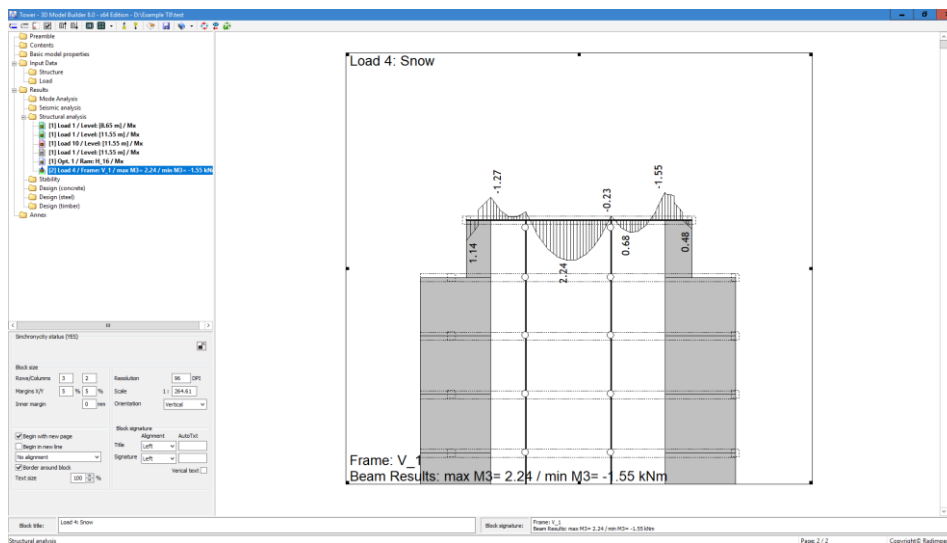
Selecting this icon or the **“Original display of the current block”** command from the drop-down menu that opens right mouse click over the block name in the report tree, the same result will be displayed in the result-processing module that was at the time of exporting the selected block, in the window for which is the export of the block executed.



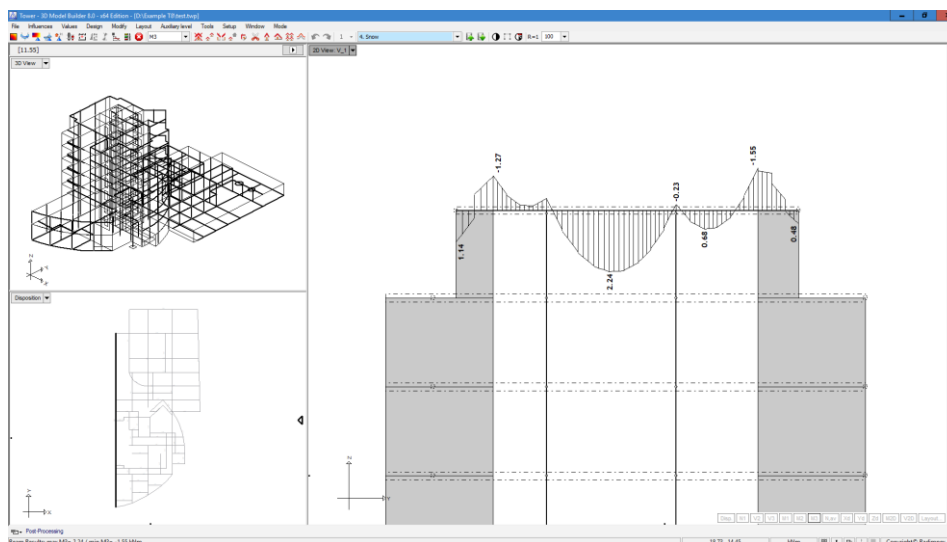
In the 2D window is set layout corresponding to the layout of the exported block

The program has set the same view in the results processing module in the corresponding window, the same load case and the same static effect that was displayed at the moment of export of the selected block.

Note that if a block is selected where the part of the construction is “zoomed” and the command **“Original display of the current block”** is started, the same part will be enlarged in the corresponding window in the results processing module.



The selected block shows only a part of the construction

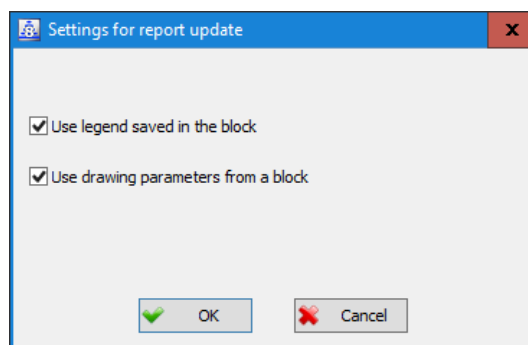


In the processed results are shown the same part of the construction that was shown in the block as well

### Settings for report update

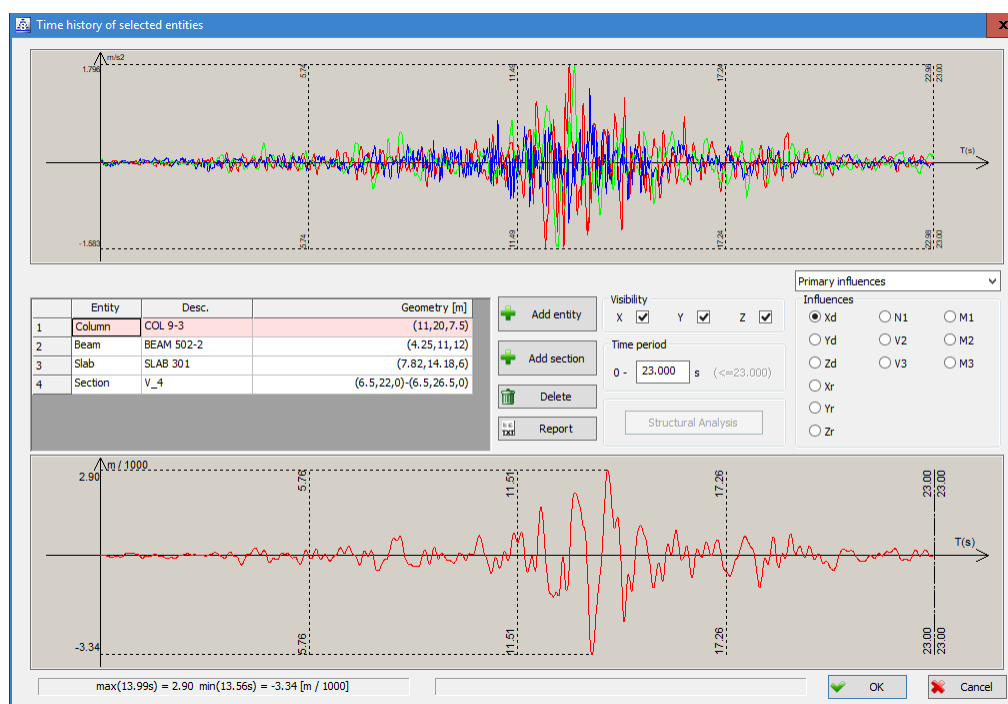


Selecting the **"Settings for report update"** icon opens a dialog where you can adjust the parameters that affect the refresh of the selected blocks. This is exactly the same dialog as the one that opens by clicking on the "Parameters" button in the dialog for mapping load cases.



## 8.25 Direct Dynamic Analysis (available only in the Expert Configuration of the Program)

### 8.25.2 Time history of selected entities



Dialog for displaying the history of influence in the selected entities

#### Add section

The button which selection closes the dialog and enters the intersection assigning procedure for which the history of influence will be displayed. The command line gets the following form:

**First Point (<End>):**

After setting the first point, the program requires users to set the second section point:

**Second Point (<End>)**

After setting a section, the dialog opens and the entered section is displayed in the table.

#### Description

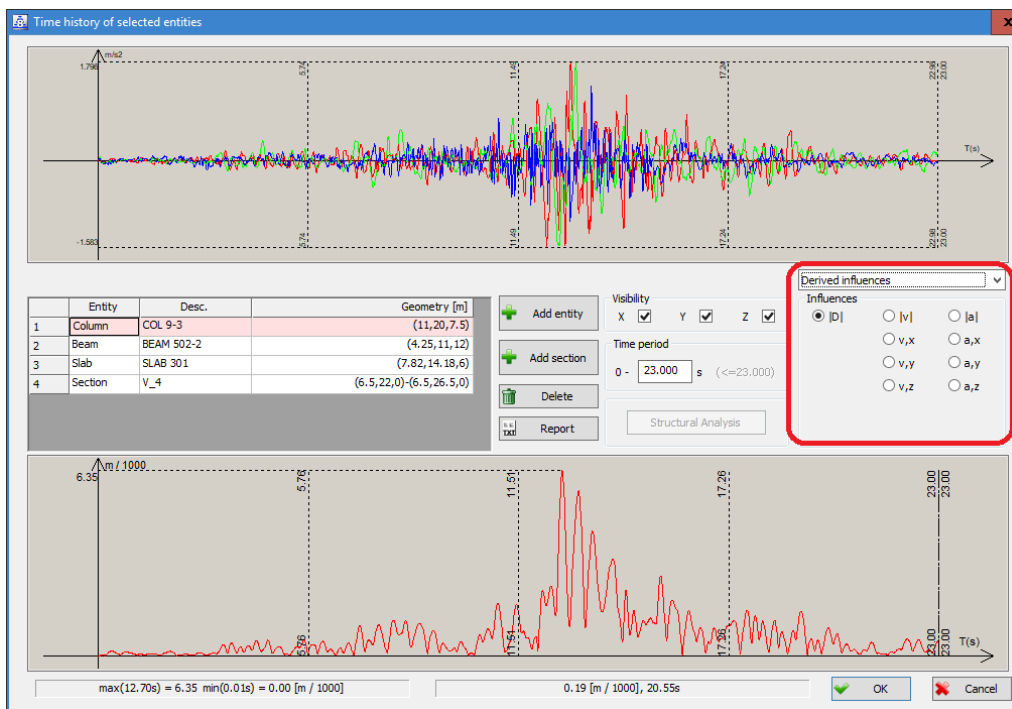
Column showing the descriptions of the positions of the entities for which the influence history will be displayed. In case an entity has no defined position description, this cell will be empty. If a section is specified, in this column the name of the assembly to which the section belongs will be printed. If it does not belong to any assemblies, cells will be empty.

#### Geometry [m]

A column showing the coordinates of the points selected from the entities for which the history of influence will be displayed. If a section is specified, the coordinates of the start and end points of an assigned section will be displayed in this column.

## Influences

In the dialog, two types of influence can be displayed: “**Primary influences**” and “**Derived influences**”. The choice of the type of influence is done from a closed list, which is above the part of the dialog with the names of the available influences. After each change in the type of influence, the influences that are available in this part of the dialog are also changed.

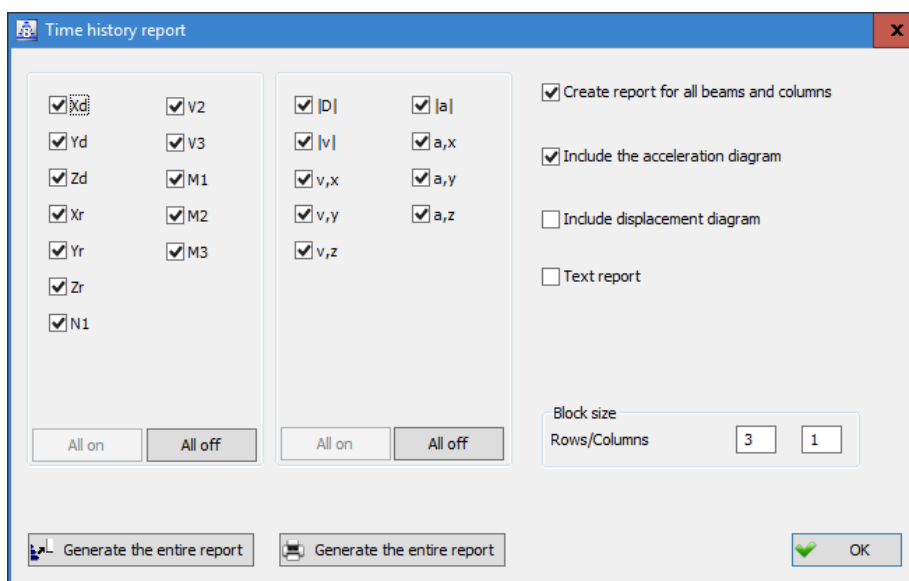


The selected influences in the assigned point of the column are shown in the list

### Derived Influences:

- |D|** - absolute deflection (the root of the sum of the squares of the component deflection)
- |v|** - absolute speed
- v,x** - speed towards the global X axis
- v,y** - speed towards the global Y axis
- v,z** - speed towards the global Z axis
- |a|** - absolute acceleration
- a,x** - acceleration towards the global X axis
- a,y** - acceleration towards the global Y axis
- a,z** - acceleration towards the global Z axis

## Report



The layout of the dialog for the history of influences in beams and pillars

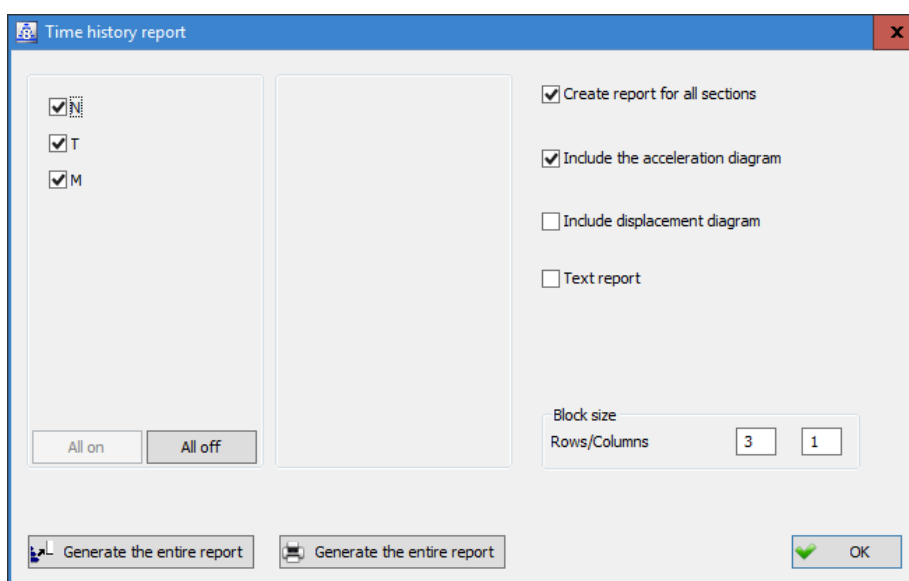
**All on** The button which turns on the check boxes in front of all the displayed influences in the dialog.

**All off** The button which turns off the check boxes in front of all the displayed influences in the dialog.

### **Text report**

If this check box is set on, the program will, in addition to graphic blocks, also generate tables with influence values at certain times for the selected influence.

The way of operation with this command is exactly the same, and when the table is selected, the only difference is in the type of influence that can be found in the report:



The layout of a dialog for generating a history of impact reports in sections

## 8.26 Columns buckling length calculation

The buckling length of the column is defined as the distance of the zero points of the moment diagram from the second-order influences. It is usually represented as the multiplication of the effective length coefficient and the actual length of the column.

The procedure of approximate determination of the buckling length is based on determining the degree of fixture at the ends of the column caused by the surrounding elements of the structure. The buckling length is determined separately for each of the directions of the local axis of the column (local axes 2 and 3).

The degree of fixture of the column at one end of it is influenced by the existence of structural elements that affect the freedom of rotation of that end of the column. These elements are supports, walls and beams that intersect in that node. When determining the stiffness of the beams that are in monolithic connection with the slab, it can be considered and joint action with the width of the slab.

The column located above is affected by the degree of fixing of the upper end of the column, and the column below is affected by the degree of fixing of the lower end of the column.

The method of calculating the length of the fixing of the column is influenced by the horizontal displacement of the structure. The structures can be classified into horizontally unmovable and horizontally movable. (shorter – unmovable and movable).

The program allows an approximate calculation for the buckling length of the column by one of the following procedures:

- 1) Approximate procedure from Eurocode 2 EN 1992
- 2) Approximate procedure from Eurocode 3 ENV 1993 Annex E
- 3) Approximate procedure of Magdy I.S.

### Approximate procedure from Eurocode 2 EN 1992

This procedure is most often used for the coefficient calculation of the length of the concrete columns. The analysis is calculated according to the following formulas:

Braced construction:

$$\frac{l_0}{L} = 0.5 \sqrt{\left(1 + \frac{K_A}{0.45 + K_A}\right) \left(1 + \frac{K_B}{0.45 + K_B}\right)}$$

Unbraced construction:

$$\frac{l_0}{L} = \max \sqrt{1 + 10 \frac{K_A K_B}{K_A + K_B}}; \left(1 + \frac{K_A}{1 + K_A}\right) \left(1 + \frac{K_B}{1 + K_B}\right)$$

Where  $K_A$  i  $K_B$  are the coefficients of flexibility of the top (A) and bottom (B) column:

$$K = \frac{\sum E \frac{I_c}{L_c}}{\sum \alpha E \frac{I_b}{L_b}}$$

In this formula subscript "b" means beams, and subscript "c" means columns.

Approximate procedure from Eurocode 3 ENV 1993 Annex E

This procedure is most often used to calculate the coefficients of the buckling length of the steel columns. The analysis is calculated according to the following formulas:

Braced construction:

$$\frac{l_0}{L} = 0.5 + 0.14(K_A + K_B) + 0.055(K_A + K_B)^2$$

Unbraced construction:

$$\frac{l_0}{L} = \left[ \frac{1 - 0.2(K_A + K_B) - 0.12K_A K_B}{1 - 0.8(K_A + K_B) + 0.6K_A K_B} \right]^{0.5}$$

Where  $K_A$  i  $K_B$  coefficients of distribution on the top (A), at the bottom (B) of columns.

$$K = \frac{\sum E \frac{I_c}{L_c}}{\sum E \frac{I_c}{L_c} + \sum \alpha E \frac{I_b}{L_b}}$$

In this formula the subscript "b" means beams, subscript "c" means columns.

Approximate procedure Magdy I.S.

Factor K represents the degree of limiting rotation at the end of the column and is given as:

$$K = \frac{\sum E \frac{I_c}{L_c}}{\sum \alpha E \frac{I_b}{L_b}}$$

In this formula the subscript "b" means beams, subscript "c" means columns.

The flexibility factor  $\beta$  is given as:

$$\beta = \frac{1}{(1 + K)}$$

The coefficients of the fixing length for braced construction are:

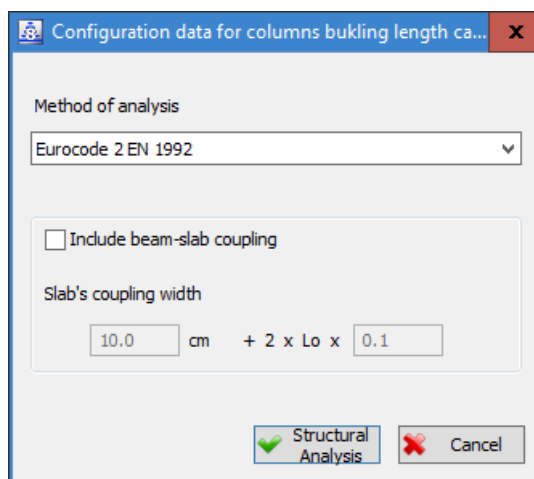
$$\frac{l_0}{L} = 1 - \frac{1}{5}(\beta_A + \beta_B) - \frac{1}{10}(\beta_A^2 + \beta_B^2) + \frac{19}{200}\beta_A\beta_B$$

Coefficient of the buckling length for unbraced construction:

$$\frac{l_0}{L} = \left( \frac{0.97K_A K_B + 3.3(K_A + K_B) + 6.7}{K_A + K_B + 6.9} \right)^{0.6}$$

if  $K_A$  and  $K_B$  are less than 10, ie:





**Method of analysis** - from the closed list, the procedure to be used for buckling coefficients calculation is selected. The offered options are: **Eurocode 2EN 1992**, **Eurocode 3 ENV 1993-1-1 Annex E** and **Procedure Magdy I.S.**

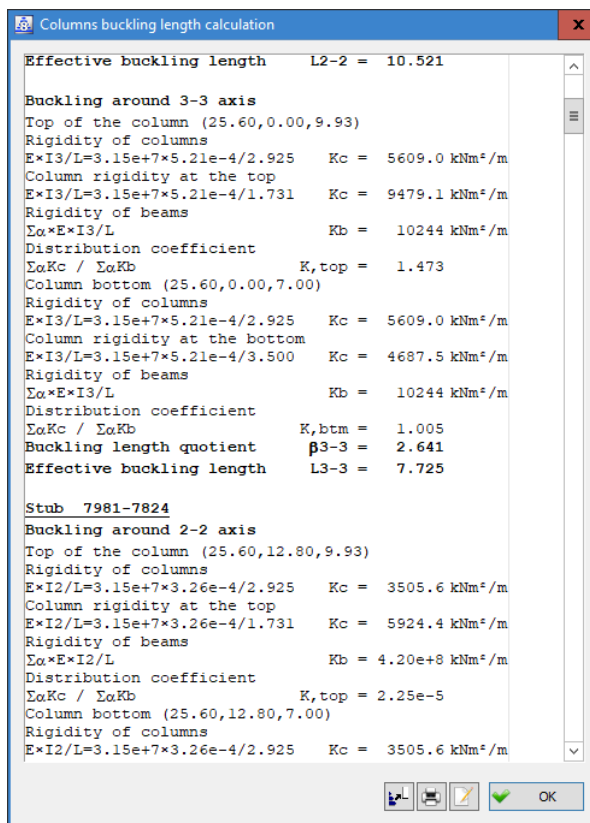
**Include beam-slab coupling** - The joint action width of the slab that affects the increase the stiffness of the beam is determined by the expression given below the check box. The data " $L_0$ " used in the formula represents a light opening of the slab, or the distance between the beams in the slab. In order for the formula to be complete in the edit fields, the minimum width of the slab must be entered as well as the coefficient by which the bright opening is multiplied.

Clicking the "**Structural Analysis**" button, the dialog box closes, and the program again calculates buckling coefficients with the parameters specified therein.

**Report** - the option by which the report of buckling coefficients calculation for selected columns is created. Choosing it you enter the usual procedure for selecting entities, so the command line gets a new look:

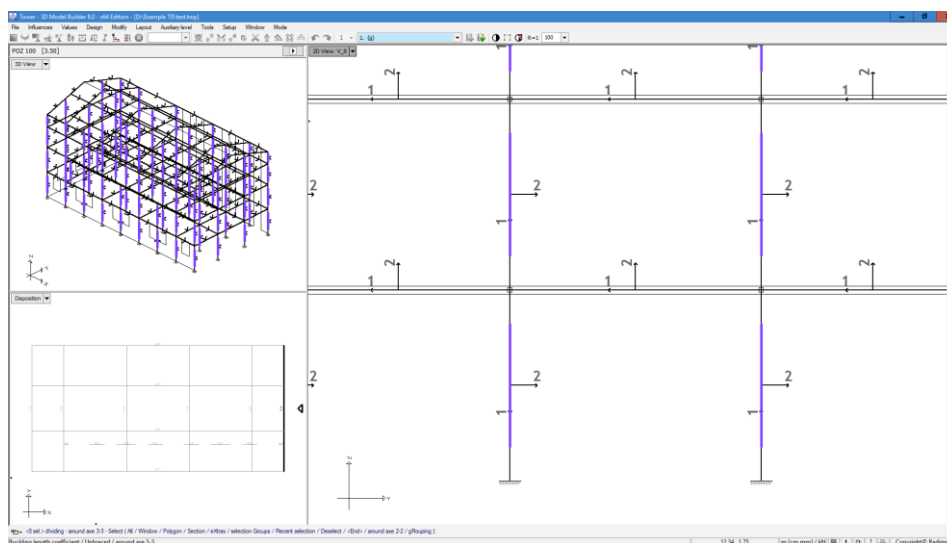
<0 sel.> Select beams for report - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End>):

When the selection of preferred columns ends, the program generates a report and displays it in a dialog from which it can be sent to the project documentation, text file or directly to the printing.



## 8.26.2 Columns Grouping

By using the “**Columns grouping**” command, you can merge the collinear columns so that the program treats them as if they were one column in the buckling coefficients calculation. This grouping can be done independently for axis 2 and axis 3, for example, columns are treated as one when calculating the coefficient for axis 2, while calculating the coefficient for axis 3 columns are treated separately. This also gives the condition that the columns that are grouped must have the same orientation of the local axes. By selecting the command, the program marks with a special colour all the columns for which the buckling coefficients are calculated:



At the same time, the command line gets the following view:

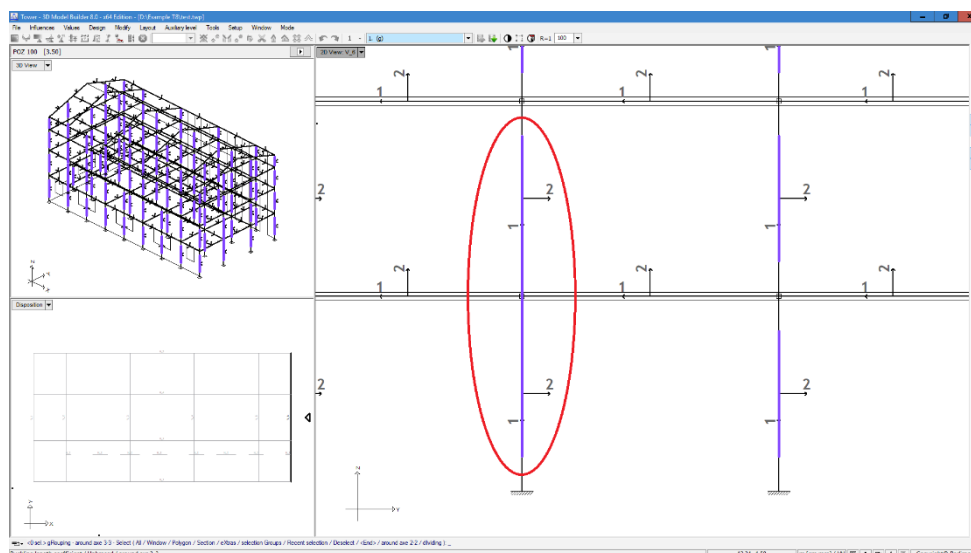
`<0 sel.> gRouping - around axe 3-3 - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End> around axe 2-2 / dIviding):`

This is the usual form of command line for entity selection. The message on the command line also indicates which local axis is currently active, or for which grouping columns is done. In addition to options that make it easy to select an entity, there are two options specific to this command:

**around axe 2-2** - every mouse click over this option changes the local axis for which columns are grouped. The axis 2 or axis 3 can be selected for the active one.

**Dividing** - an option by which previously grouped columns can be divided again.

After selecting the desired columns, the program executes their grouping, but the command isn't interrupted, the new columns can be selected, and so long as the right mouse button or the option "**End**" does not end the command. At the same time, the grouped columns are marked on the screen in the same way as they together make up one column.



Two grouped columns are marked as if one column was marked

## 8.27 Show area entities extremes



The command "**Show area entities extremes**" defines whether the values of influence and the corresponding symbols will be printed when displaying influences in surface entities, at the points where the extremes are located. When the command is an active icon, it is imprinted in front of its name, ie framed by a blue square (the way active icons are displayed depends on the operating system used). Printing of these values can also be enabled using the check box of the same name, which is located in the dialog of the "Functionality" command, in the "Results" file (chapter "3.5.2").

## 8.28 Show diagram values in 3D



In the "3D view" window, numerical values can be printed on the diagrams of the displayed influences, in both line and surface entities. The display of these values is activated by selecting the command "**Show diagram values in 3D**" from the drop-down menu "Values". When the command is an active icon, it is imprinted in front of its name, ie framed by a blue square (the way active icons are displayed depends on the operating system used). Showing of these values can also be enabled using the check box of the same name located in the dialog box of the "Functionality" command, in the "Results" file (chapter "3.5.2").

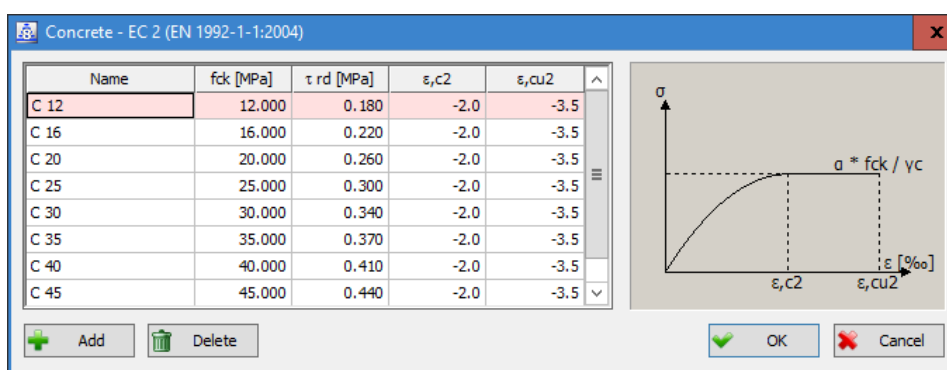
# 9. Concrete Design

## 9.1 Selecting the Codes for Design (Codes)

A new regulation for dimensioning concrete SRPS EN 1992-1-1 has been added. In that regulation, the default value for  $\alpha$ , cc is 0.85.

### EUROCODE

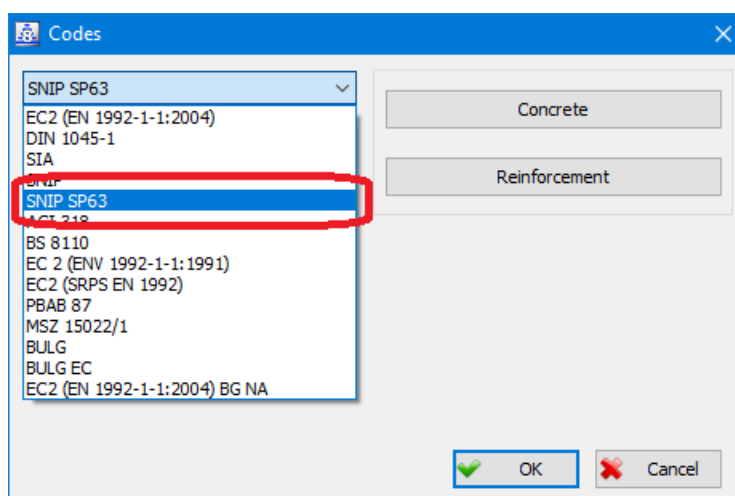
By selecting the "**Concrete**" command field, a dialog for defining concrete characteristics opens:



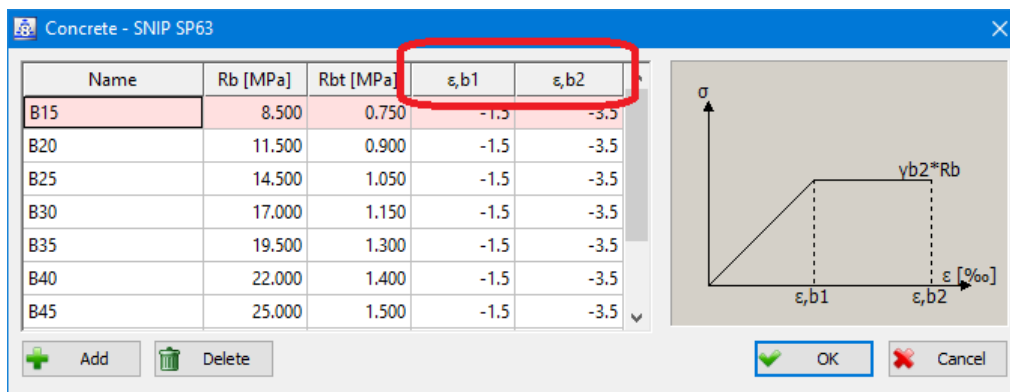
- "Name" column with the types of concrete.
- "fck" calculating compressive strength.
- " $\tau_{rd}$ " calculating shear strength.
- " $\epsilon_{c2}$ " i " $\epsilon_{cu2}$ " boundary dilatations of concrete, which enable the definition of the working diagram of concrete.
- " $\alpha \cdot cc$ " coefficient for taking into account long-term effects on concrete strength.

### SNIP SP63

A new Russian concrete regulation SNIP SP63 has been added.



In the dialog for defining the characteristics of this concrete, limit dilatations can be specified, which enable the definition of the working diagram of the concrete " $\epsilon_{b1}$ " and " $\epsilon_{b2}$ ".



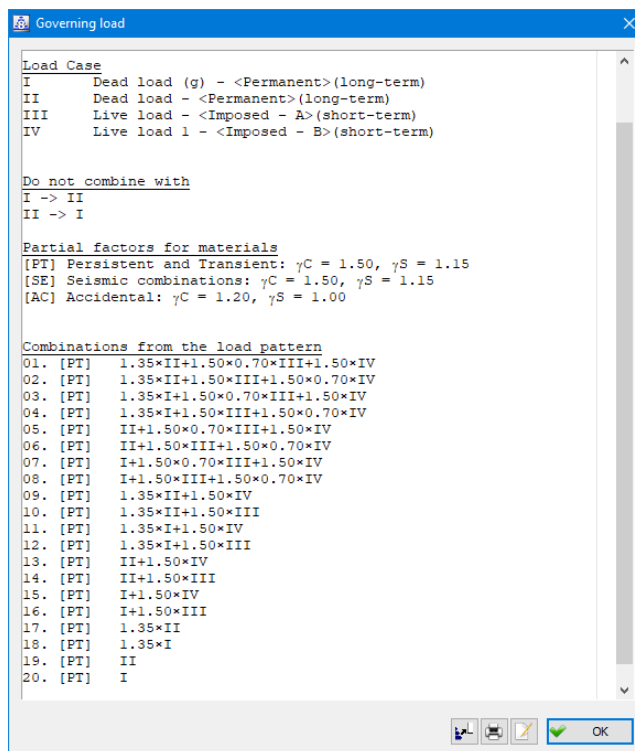
Limit dilatations of concrete

## 9.2 Defining the Load Combining Pattern

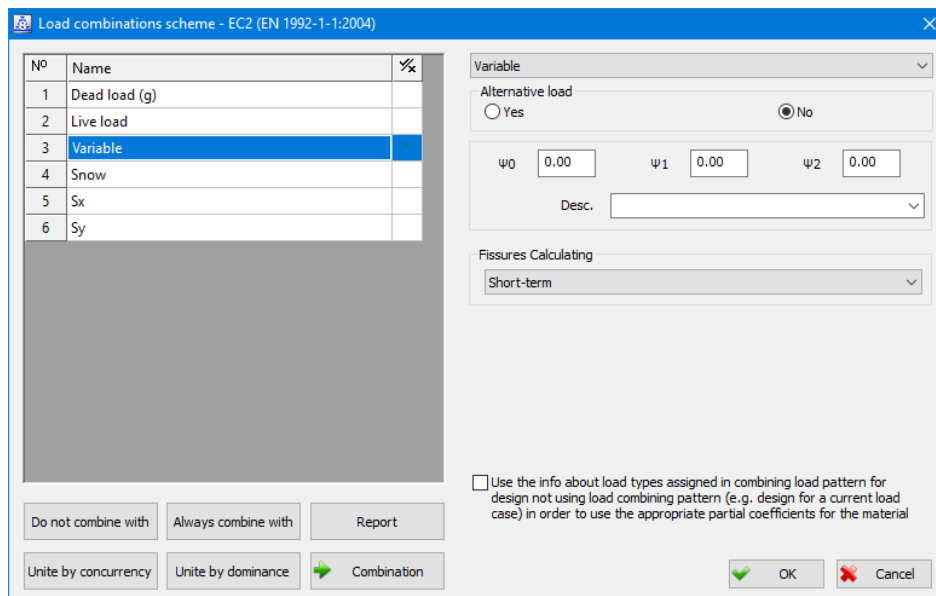
If we have several loads that are given the character "Permanent", and at the same time they are given not to combine with each other, the program will make an independent set of combinations for each of them. We will take as an example the case with two permanent (I and II) and two imposed load cases (III and IV). If we include that permanent load cases are not combined with each other, we get the following combinations:

$1.35 \times II + 1.05 \times III + 1.5 \times IV$   
 $1.35 \times II + 1.5 \times III + 1.05 \times IV$   
 $II + 1.05 \times III + 1.5 \times IV$   
 $II + 1.5 \times III + 1.05 \times IV$   
 $1.35 \times II + 1.5 \times IV$   
 $1.35 \times II + 1.5 \times III$   
 $II + 1.5 \times IV$   
 $II + 1.5 \times III$   
 $1.35 \times II$   
 $II$   
 $1.35 \times I + 1.05 \times III + 1.5 \times IV$   
 $1.35 \times I + 1.5 \times III + 1.05 \times IV$   
 $I + 1.05 \times III + 1.5 \times IV$   
 $I + 1.5 \times III + 1.05 \times IV$   
 $1.35 \times I + 1.5 \times IV$   
 $1.35 \times I + 1.5 \times III$   
 $I + 1.5 \times IV$   
 $I + 1.5 \times III$   
 $1.35 \times I$   
 $I$

It can be seen from the attached that the combination of imposed load cases with each of the permanent ones has been performed.



If the EC 2 regulation (EN 1992-1-1: 2004) is selected for the dimensioning of concrete and calls the command for defining the load join scheme, the following dialog will appear:

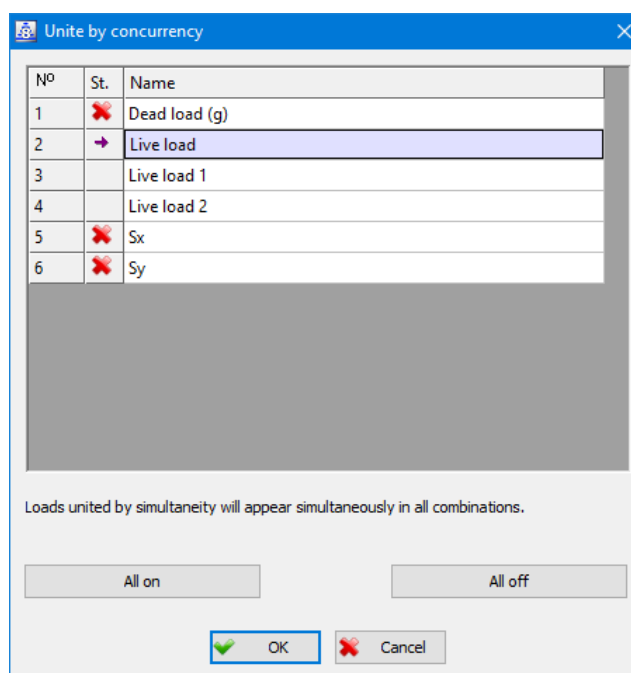


The layout of a dialog for defining a load join scheme

When the load case is assigned the character of the occasional load, the edit fields for entering the value of the  $\Psi$  combination factor become available. In case there are more loads of this character, each of them can be assigned different values of the  $\Psi$  join factor and in the edit field of the "**Description**" give a short description that will associate you with the entered data. Clicking on the arrow on the right side of this closed list opens the list content from which one of the previously defined descriptions can be selected.

**Unite by concurrency** – command field by which the basic load cases can be grouped by concurrency, meaning that they will appear simultaneously in all combinations. Load cases with different character can be combined. Exceptions are loads that are assigned one of the following characters: Permanent, Prestress, Seismic and Incident. They can only be combined with load cases that have the same character.

First, you need to select the load cases that you want to combine with some other cases, and then click on the “Unite by concurrency” button, after which a dialog of the following appearance will open:



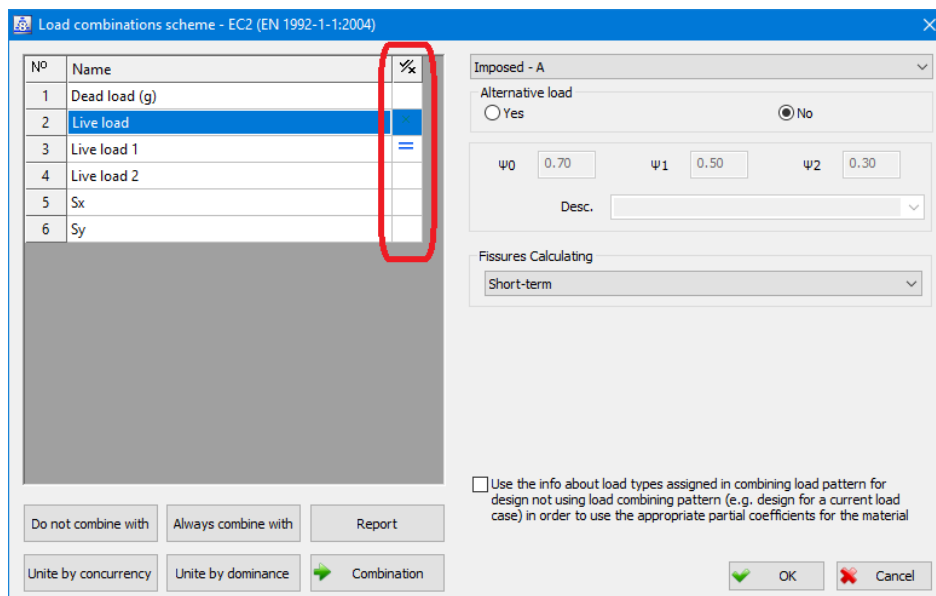
Dialog for defining the united Load cases by concurrency

The displayed list contains all the given basic load cases. In the column “St.” the load for which this dialog is opened shows the arrow “➔”. Now it is necessary to click the mouse to mark all the loads that will be combined with it. The symbol “☑” will be placed in the “St.” column for the selected loads. Deselecting a previously selected load case is done by clicking again with the mouse over its name, after which the “☑” symbol will be removed.

The symbol “✘”, which can be found in front of the names of some load cases, indicates that these loads cannot be merged with the loads for which this dialog is open.

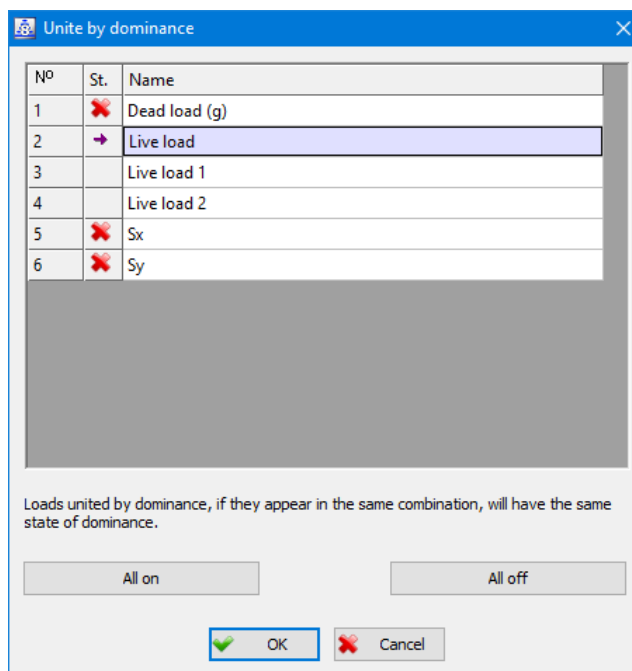
When you have selected all loads for unification, activate the “OK” button, after which the dialog will be closed.

In the “☒” column, the program places the “☑” symbol in front of all load cases that are united by concurrency with the currently selected load case in the list.



The placed symbols "=" in the column "x" indicate that they are the second and third load case united by concurrency



**Unite by dominance** - In addition to grouping load cases by concurrency, uniting by dominance is also enabled. Load cases combined in this way will have the same  $\Psi$  combination factor in all combinations. It is necessary to select the load cases that you want to combine with some other cases, and then to click on the "Unite by dominance" button, after which the following dialog opens:

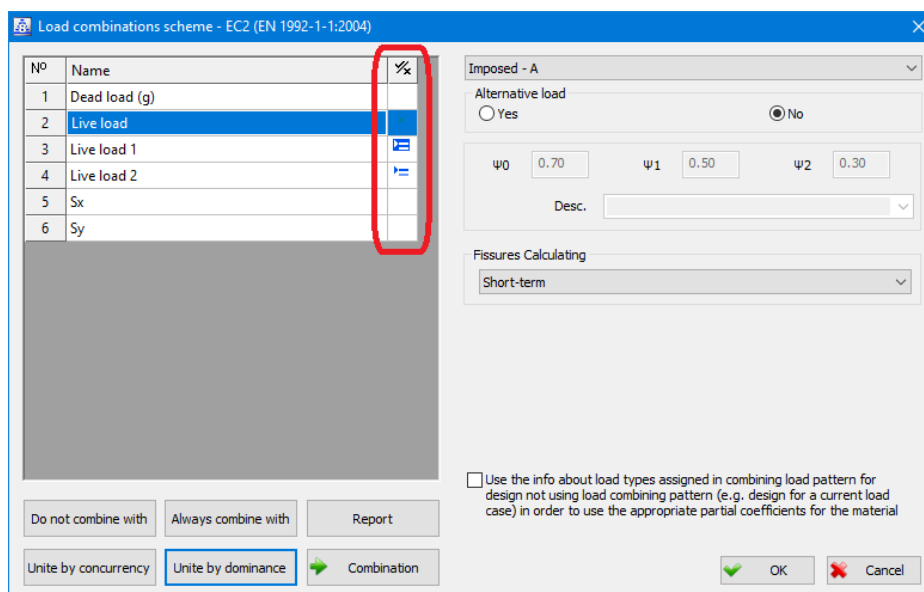



Dialog for defining the united Load cases by dominance

Working with this dialog is exactly the same as with the "Unite by Concurrency" dialog, so we won't explain it again.

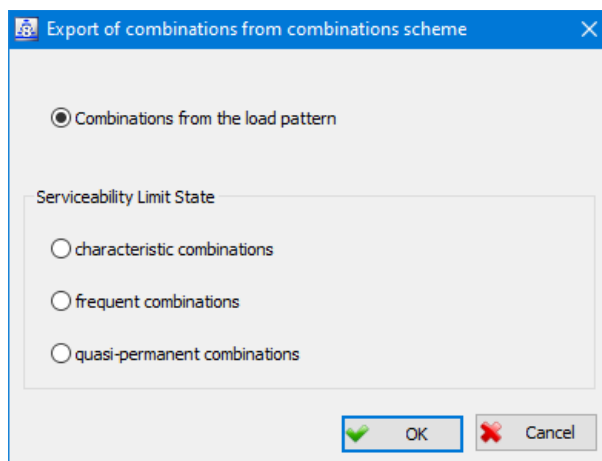
After exiting the dialog, in the column "x" the program places the symbol "=" next to all the load cases that are united by dominance with the currently selected case.

If the selected load case is combined with another load case both by dominance and simultaneity, the program places the symbol "" next to it in the column ".



The placed symbols in the "" column indicate that they are the second and fourth load case united by dominance, and that they are the second and third united both in terms of simultaneity and dominance

In all regulations based on the EC2 regulation, it is possible to export all combinations that are automatically generated for the calculation of cracks and deflections (combinations for the Limit state of usability). Selecting the "**Combinations**" button opens a dialog of the following appearance:



Dialogue layout for exporting combinations from the combining scheme

By setting one switch to the on state, a selection of combinations to be exported is made. The meaning of each of the offered switches is the same as in the previously described dialog for generating the combination scheme report.

## 9.3 Slab Design

### 9.3.1 Input Data

#### 9.3.1.1 Input data - Global

Selecting this command, the following dialog is opened:

The layout of the dialog for defining global input data

**Mxy** – For slabs that are dimensioned with the check box “**Mxy**” on, the Mxy torque values as well as the bending moment values M1 and M2 are displayed in the report.

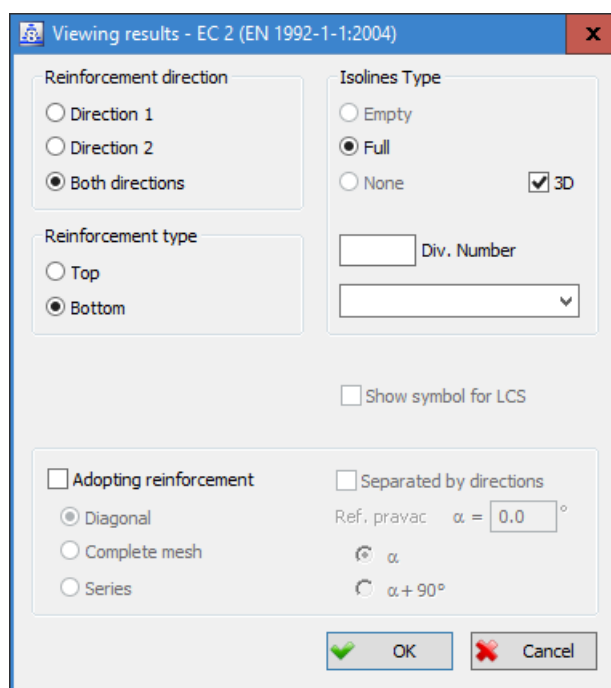
**Laying one reinforcement region over multiple slabs** - The state of this check box determines the default behaviour of the program when setting the reinforcement area that traverses across multiple slabs. If the check box is off, the program will split the specified reinforcement area into multiple areas so that they monitor the contours of the slabs over which the reinforcement area is passed. If the check box is turned on, only one reinforcement area will be set, no matter if it goes over several slabs.

The program enables the dimensioning of concrete slabs for loads from all phases of construction, and not only from the last phase. If the model contains construction phases, at the bottom of the dialog becomes an active closed list from which the user selects how the dimensioning will be performed:

**Design performed for last construction stage only** - the dimensioning of concrete slabs will be done for loads from the last phase of construction.

**Design for all construction stages** - the dimensioning of concrete slabs will be done for influences from all phases of construction.

### 9.3.3 Presenting the Design Results



Layout of the dialog for slabs design results

By turning on the check box "**Adopting reinforcement**", the user is allowed to select one of the three offered modes of display of the adopted reinforcement:

"**Diagonal**" – Setting on the switch it displays the diagonal on which the diameter and distance of the adopted reinforcement is printed.

"**Complete mesh**" – Setting on the switch the program draws each bar separately, respecting the specified distance between the bars.

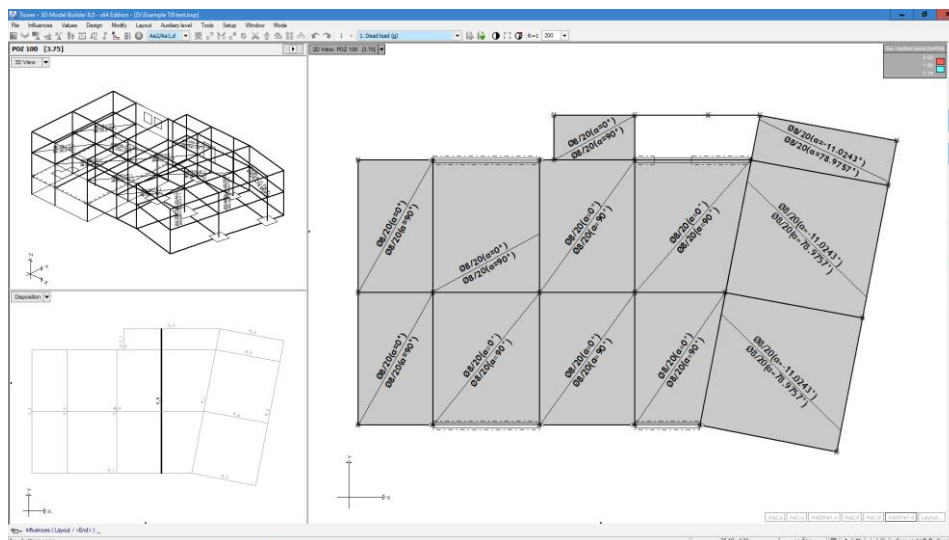
"**Series**" – Setting on the switch, for each area of reinforcement, draws the direction lines which shows the spreading direction of series and one of the bars in the series. In the section point of the bar and the line of the spreading direction of series, the angle with the diameter of the bars and the distance between them is shown.

For each of the selected methods of displaying the reinforcement, it is possible to display only one direction of reinforcement by turning on the check box "**Separated by directions**".

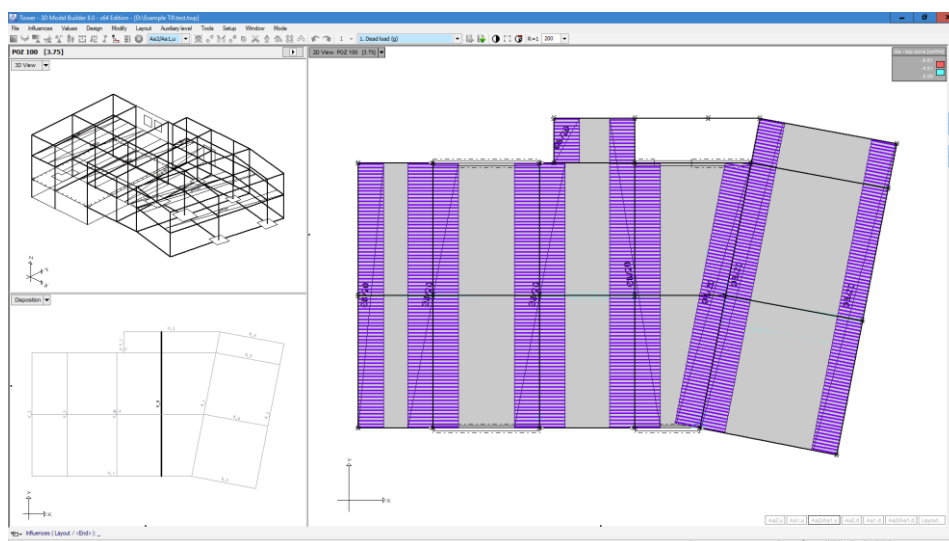
" $\alpha =$ " - Edit field for setting the angle of the main reinforcement direction. With the switches underneath this edit box, a selection of the direction will be displayed:

" $\alpha$ " - reinforcement will be displayed that extends in the direction of the angle " $\alpha$ " and its close angles.

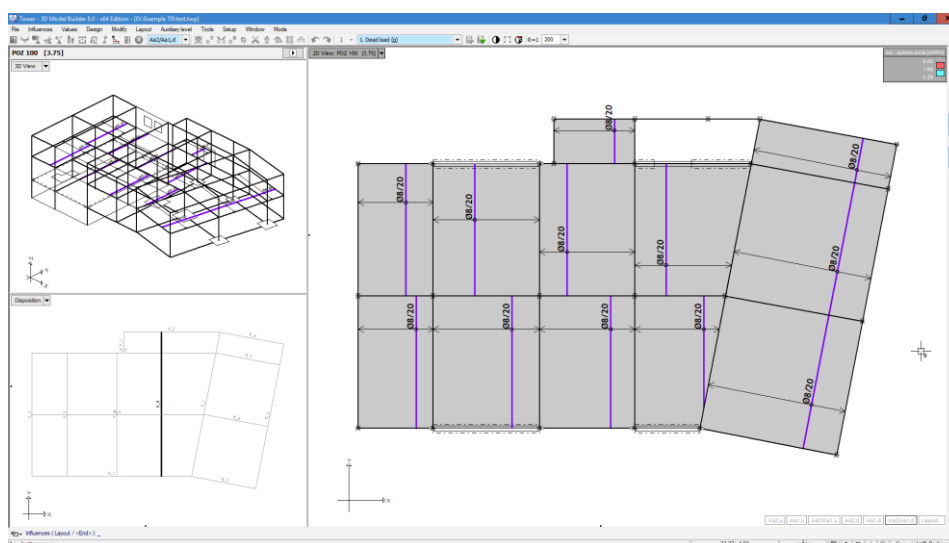
" $\alpha + 90^\circ$ " - reinforcement will be displayed that extends in the direction of the angle " $\alpha + 90^\circ$ " and its close angles.



The layout of area of reinforcement with a diagonal

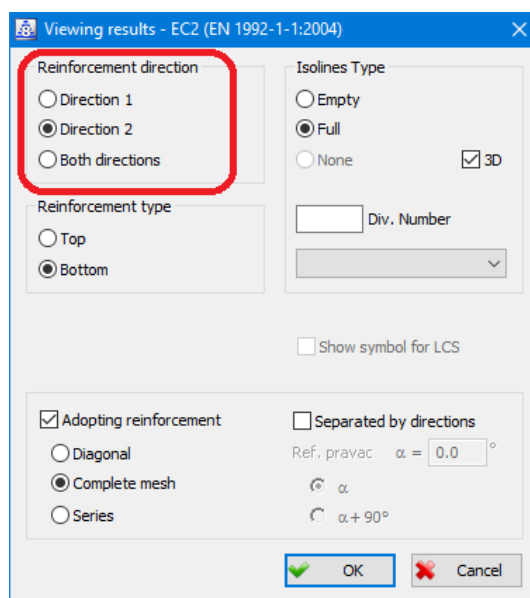


Full layout of bars in the direction " $\alpha=0^\circ$ " angle



The layout of the bar series in the direction " $\alpha+90^\circ$ " angle

If the switch "Separated by directions" is turned off, the selection state of switches in the "Reinforcement directions" group will determine showing the reinforcement in only one or both directions. This applies to each of the ways of displaying reinforcement - "Diagonal", "Complete mesh" or "Series".



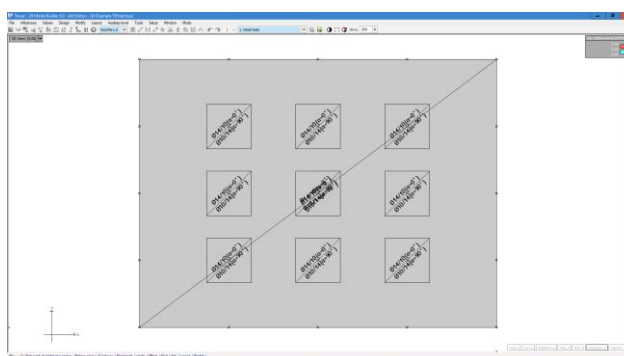
### 9.3.6 Manipulating the Reinforcement Regions

#### Joining the contours of reinforcement regions

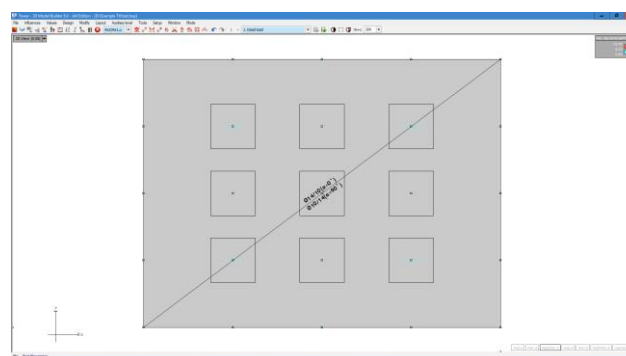
The program allows join the contours of the reinforcement region of the same set of data that belong to different slabs. By selecting the command "**Joining the contours of reinforcement regions**" the command line gets the following view:

<0 sel.> Join Contours - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End>):

Now, users are expected to select all desired contours of reinforcement regions that need to be merged.



Reinforcement areas divided by slabs before the command "Joining the contours of reinforcement regions"

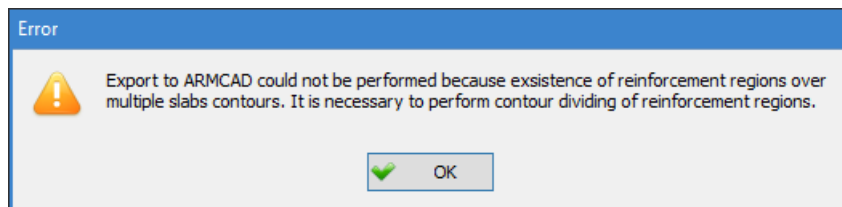


The reinforcement areas are merged after the command "Joining the contours of reinforcement regions"

After the selection was completed, the program performed the merging of the selected contours of the reinforcement regions of the same set.

### Divide contours of reinforcement region

ArmCAD program does not allow the import of the adopted reinforcement in which the reinforcement regions lays over multiple slabs. In such situations, during the "Export in ArmCAD" command, the following warning will appear:

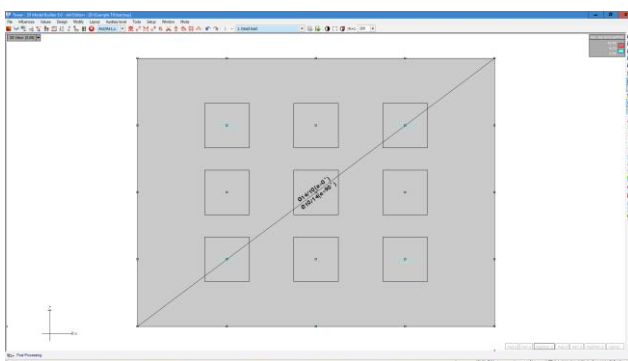


The user could divide the reinforcement regions using the "**Divide contours of reinforcement region**" command, so that the adopted reinforcement can be exported into the ArmCAD program.

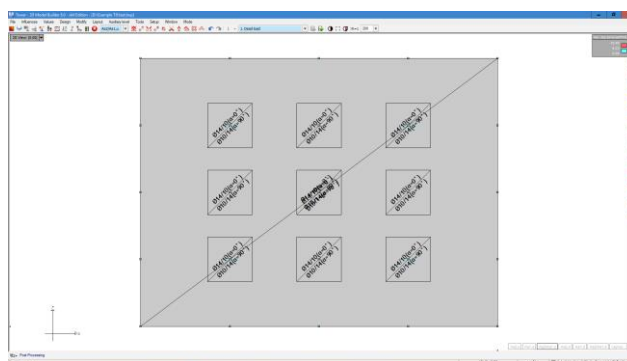
By selecting the command "Divide contours of reinforcement region", the command line gets the following layout:

<0 sel.> Dividing reinforcement regions over slabs - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End>):

Now, users are expected to select the contours of the reinforcement areas that need to be divided.



The reinforcement areas are joined after the command "Joining the contours of reinforcement regions"



The reinforcement areas are divided into slabs after the command "Divide contours of reinforcement region"

After finishing the selection, the program divided all selected contours of the reinforcement area into slabs.

### Reinforcement zone change

It is possible to move the reinforcement area from one reinforcement zone to another, as well as copy the reinforcement area from one zone to another.

By selecting the "**Modify Sets**" command, the command line gets the following appearance:

<0 sel.> Modify Sets - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <eNd>):

Users are now expected to select the contours of the reinforcement area to be moved or copied from one reinforcement zone to another. After the selection is completed, the following dialogue opens:

As,reqd.: 3.29 cm<sup>2</sup>/m    As',reqd.: 1.87 cm<sup>2</sup>/m  
 As,prov.: 2.51 cm<sup>2</sup>/m    As',prov.: 2.51 cm<sup>2</sup>/m

Set	Name	φ1 [mm]	e1 [cm]	φ2 [mm]	e2 [cm]	n
→ 1		8.0	20.0	8.0	20.0	1

With Copy

Move to other zone

Copy to other zone

Buttons: Add, Up, Down, Library, OK, Cancel

Check boxes for moving and copying areas of reinforcement from one zone to another

### Move to other zone

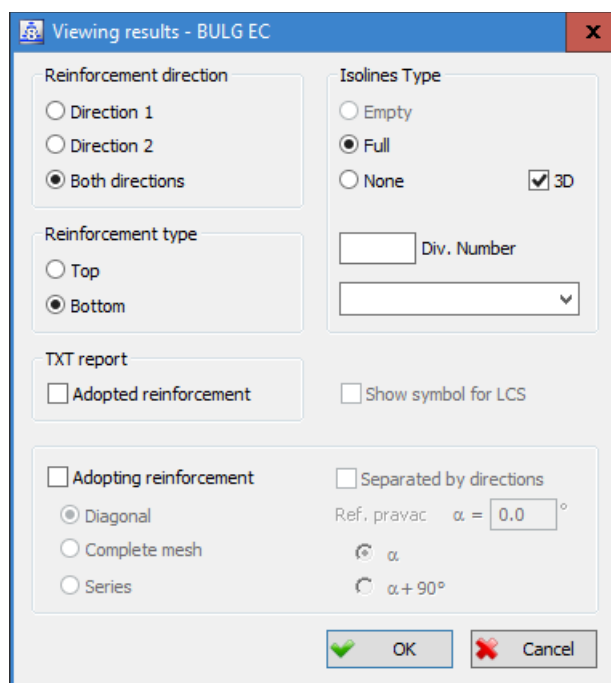
By setting the given check box to on, the selected contour of the reinforcement area will be moved from the current zone to another reinforcement zone.

### Copy to other zone

By setting the given check box to on, the selected contour of the reinforcement area will be copied from the current zone to another reinforcement zone.

### 9.3.7 Creating the Textual Report

By selecting the "**Layout**" sub-command from the command line, a dialog opens in which you can request one of the programmed methods for displaying the dimensioning results:



The layout of the dialog for slab concrete design results report

This dialog fully matches the dialog that opens within the command to display the slab concrete design results (see chapter 9.3.3), and we will not explain it again.

### 9.3.11 Slab punching verification

#### Support reaction force decrease

By setting the check box to the on state, the control of the foundation slabs makes it possible to take into account the reduction of the shear stresses in the slab due to the existence of reactive soil load in the perimeter of the breakthrough. The check box is only active if the "Foundation slab" check box is on, and "column axial force" is set to the strain source.

Slab punching verification

EC 2 (EN 1992-1-1:2004)

Level: [0.00 m]

Section 1 (-24.00,18.00,0.00)

C 30

Slab

Concrete: C 30

Distance from the slab edge to the steel centroid: 3.00 cm

Position of critical perimeter: 2.00 x hs

Foundation slab

Support reaction force decrease

Substitute with column with circular cross section

Settings

$\gamma_c = 1.50$   $K_1 = 0.10$

$CRd,c = 0.120$   $k, V_{min} = 0.035$

$v_{Rd,max} = 0.40 \cdot v \cdot f_{cd}$

$v_{Ed} / v_{Rd,c} \leq k_{max} = 1.50$

Calculate

Nº	Lh [m]	
1	0.313	Reinforcement not required
2	0.627	Reinforcement not required
3	0.940	Reinforcement not required

Section at the face of the column: Punching resistance not exceeded.

Codes: EC 2 (EN 1992-1-1:2004)

Punching shear reinforcement

Reinforcement type: S500N

Inclination angle: 90°

Bar spacing in radial direction: 20.00 cm

Source of the influences and stresses: column axial force

Perimeter length to inner column perimeter length ratio: 1.000

Eccentricity factor  $\beta = 1.000$

Current load case

Complete scheme

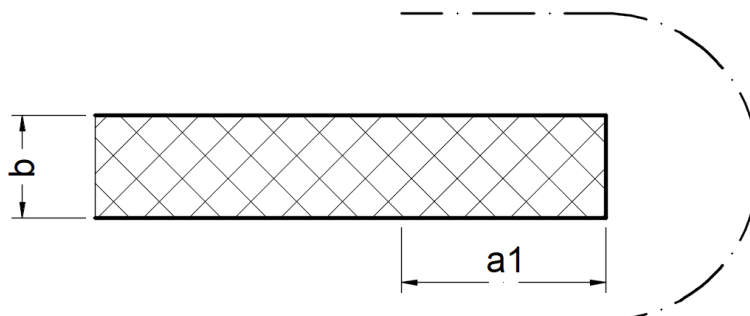
OK Cancel

Check box "Support reaction force decrease" is set to on

Slab punching verification can also be carried out at the end of the wall. The length of the wall edge zone ( $a_1$ ) which is considered by control of the slab punching and from which forces are taken is given as:

$$a_1 = \min(2b, 6d_m - b)$$

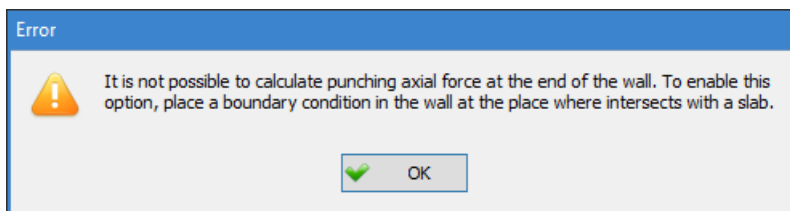
$d_m$  is the static height of the slab



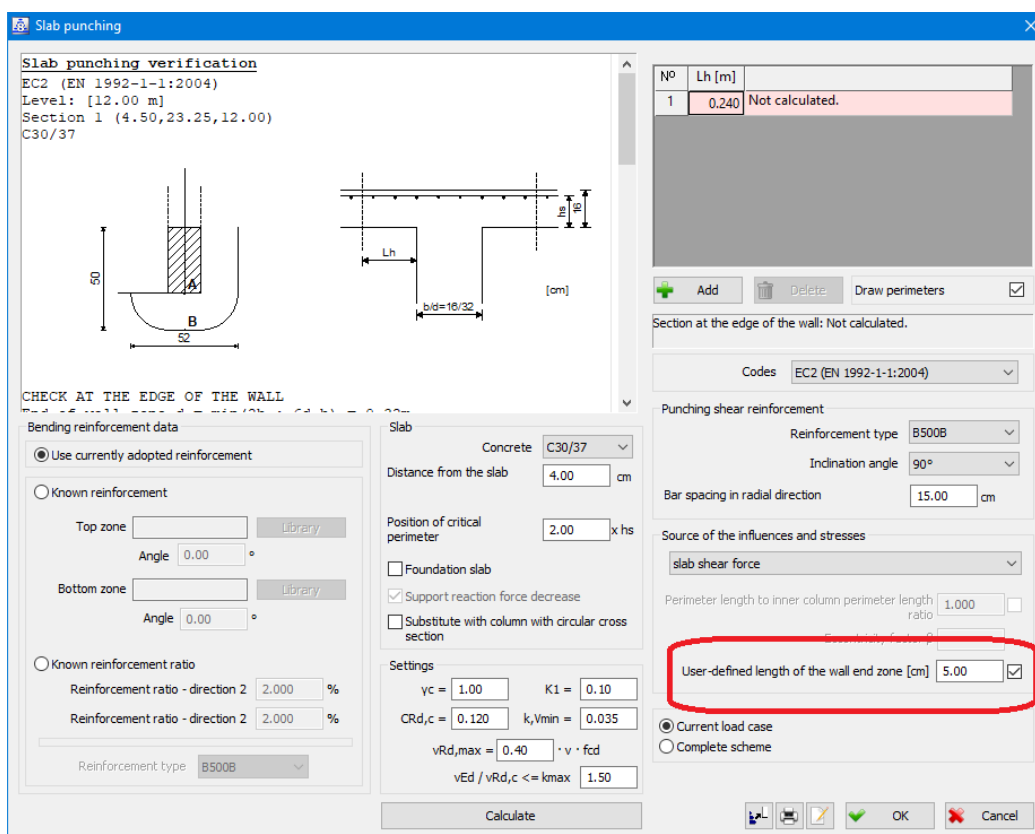
$$a_1 = \min(2b ; 6d_m - b)$$

If at the selected place in the slab there is a wall above and below the slab, the "axial force at the end of wall" option can not be used to calculate the slab punching control if a fictitious boundary condition is not set on the wall at the connection point with the slab. Setting the

boundary conditions is necessary to calculate the difference in force above and below the slab. In such situations, the program will issue an appropriate notice:



The option of user input of the edge zone of the wall has been introduced in the slab punching control at the end of the wall:



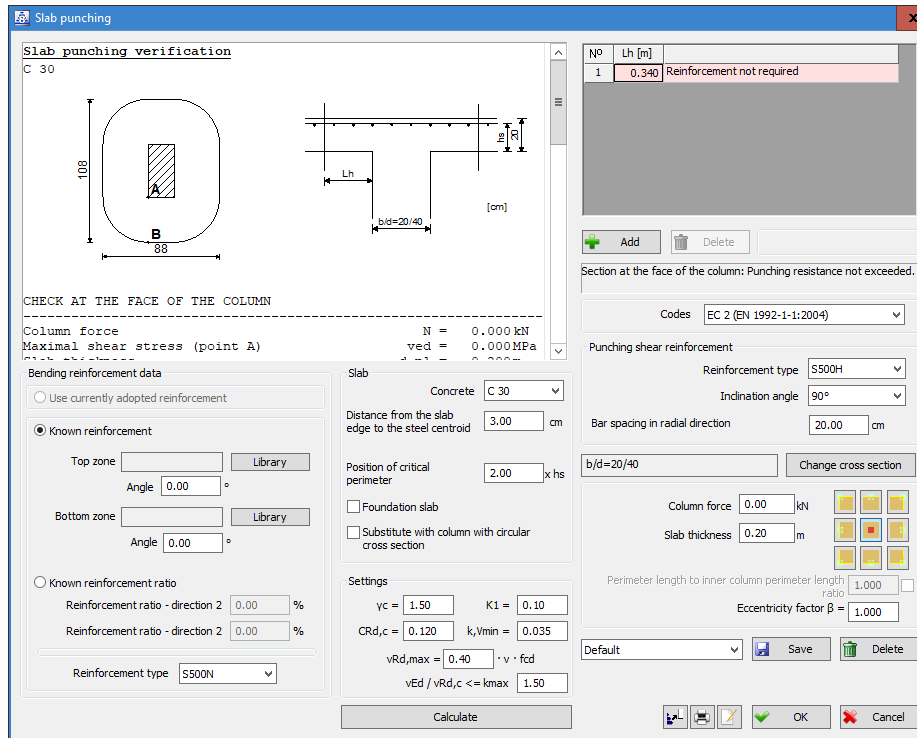
Part of the dialog for entering the edge zone of the end of the wall

### User-defined length of wall end zone [cm]

By turning on the check box, an edit field becomes available, in which the length of the edge zone of the wall is set, which is considered by controlling the punching slab and from which the forces are taken into account.


### Calculator


Selecting the "Calculator" sub-command from the command line, a dialog opens in which the user has the ability to control the slab punching - to arbitrarily set the cross-section of the column and arbitrarily set the force in the column:



A dialog opens by selecting the "Calculator" sub-command from the command line

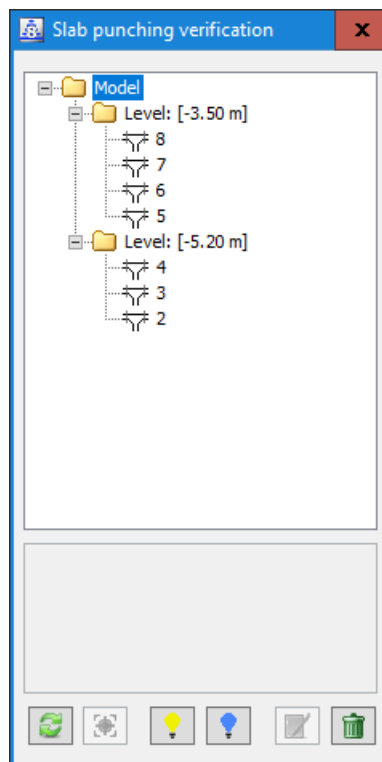
The closed list in the lower right corner of the dialog is provided for selecting some of the previously recorded configurations.

 **Save** By selecting this button, you are saving the current state of the parameters in the dialog.

 **Delete** By selecting this button you are deleting the current configurations.

### 9.3.12 Review of slab punching verification

By choosing the “**Review of slab punching verification**” command, which is located in the “Design ► Concrete” menu, the following dialog opens:



In the dialog, a list of all the places on the slab, grouped by assemblies, is displayed, in which punch control is performed. Every place in the list is represented by a symbol and an ordinal number corresponding to the ordinal number of the graphic symbol in the drawing.



Conducting new analysis of slab punching control, as well as deleting and editing existing ones, can cause the list in the dialog to no longer correspond to the current state of the model. By clicking this button, the list is refreshed, ie its alignment with the model.



The activation button in the 2D view is set for the current array containing the selected slab punching control. The graphic symbol of the currently selected slab punching control will be highlighted in the drawing. Setting in the 2D view of the array to which the punch control belongs can also be done by the left double click of the mouse over its name in the list, as well as by choosing the “**Go to assembly**” command from the drop-down menu that opens with the right mouse click over its name.



Clicking on this button will turn on the visibility of the selected punch slab controls, if previously disabled. If assemblies are selected in the trunk, the visibility of all the punch slab controls that belong to them will be turned on. The visibility of the control of the punching slab can also be activated by selecting the option “**Visible**” from the drop-down menu, which is opened by right-clicking on its name or the name of the assembly.



Clicking on this button excludes the visibility of the selected punch slab controls. If assemblies are selected in the trunk, the visibility of all puncture slabs controls that belong to them will be excluded. The visibility of piercing control can also be switched off by selecting the **"Not visible"** option from the drop-down menu, which is opened by right-clicking on its name or the name of the circuit.



A button used to edit the slab punching control that is currently selected in the list. Clicking on this button opens the following view at the dialog:

The layout of the dialog for editing previously executed slab punching control

This dialog is the same as the dialog that opens when calculating the slab punching control, so we will not re-explain it (see chapter 9.3.11).



The activation button currently removes the control panel of the slab punching control from the model. The command to delete the slab punching control is also found in the menu that opens with the right mouse click over its name in the list. Since the punch controls in the list are grouped by arrays, the right mouse click over the name of the box opens a menu with the **"Delete all"** option, which makes to erase all slab punching controls from the given array.

## 9.4 Beam Design

### 9.4.1 Input Data

#### 9.4.1.1 Input data - Global

The layout of the dialog for global input data

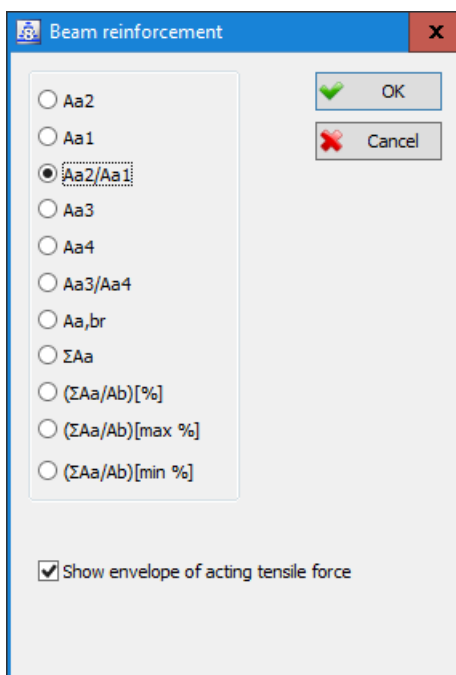
**Shift** - It is possible to set the global value of the shift of the line of tensile forces. The shift value is equal to the product of the static girder height and the coefficient specified in the edit field. The default value of the coefficient for beams for Eurocode regulations is 1.125. The shift value for any beam can be subsequently changed in the dialogue for defining local input data (see section "9.4.1.2").

The program allows the dimensioning of concrete beams for all influences from all stages of construction, and not only from the last stage. If the model contains construction phases, at the bottom of the dialog becomes an active closed list from which the user selects how the dimensioning will be performed:

**Design performed for last construction stage only** - the dimensioning of the concrete beams will be done for influences from the last phase of construction.

**Design for all construction stages** – the dimensioning of the concrete beams will be done for influences from all phases of construction.

### 9.4.3 Presenting the Design Results



Appearance of the dialog box for selecting the diagrams of required reinforcement

#### $(\Sigma Aa/Ab)[\%]$

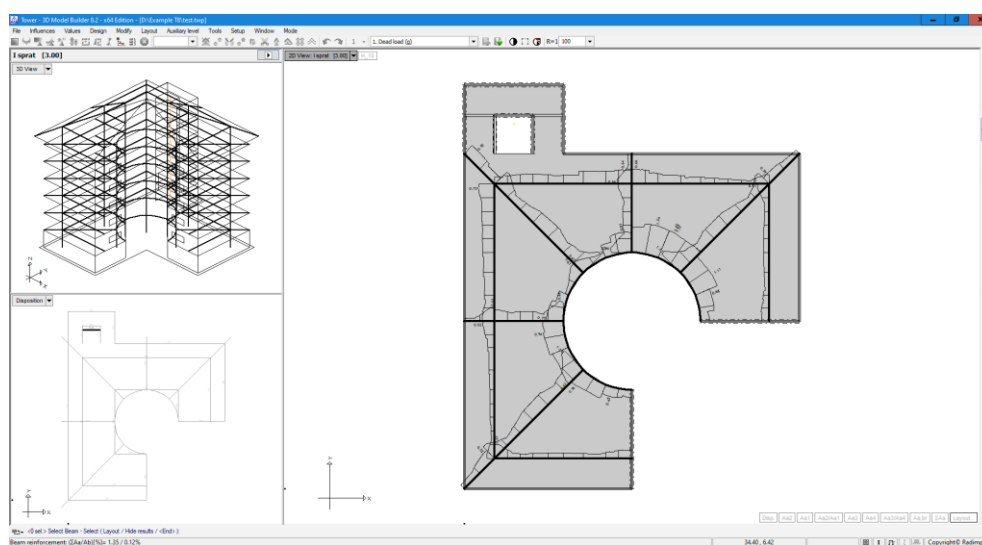
Display the diagram of the percentage of reinforcement of beams / columns for the required reinforcement.

#### $(\Sigma Aa/Ab)[\text{max } \%]$

Display of the maximum value of the percentage of reinforcement on all beams / columns for the required reinforcement.

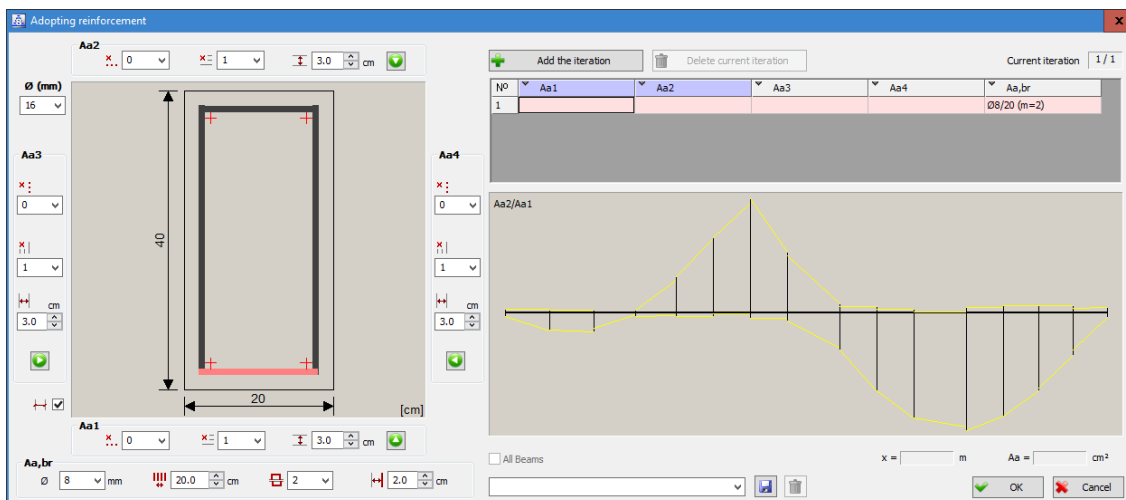
#### $(\Sigma Aa/Ab)[\text{min } \%]$

Display of the minimum value of the percentage of reinforcement on all beams / columns for the required reinforcement.



Display the diagram of the percentage of reinforcement of beams

## 9.4.4 Adopting the Reinforcement



The layout of the dialog for reinforcement adoption in beams

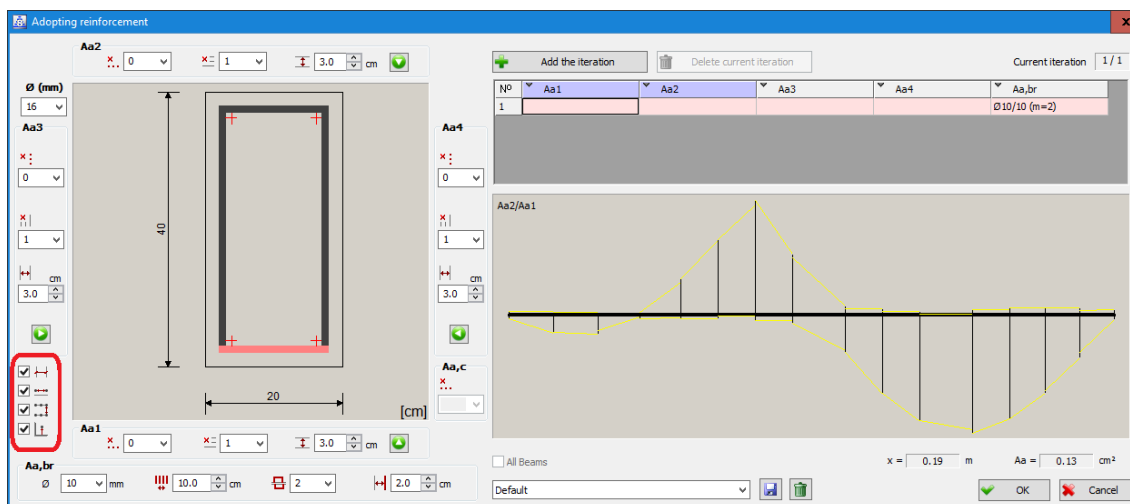
### Diagrams

In the right part of the dialog, diagrams of the calculated reinforcement along the selected beams are displayed.

Above the diagram of the calculated reinforcement there is a table with data on the adopted reinforcement. In the columns with the data of the adopted reinforcement "Aa1", "Aa2", "Aa3", "Aa4", the surface of the adopted reinforcement in the section is written in brackets.

### Cross section

In the left part of the dialog there is a space for the cross-section for which the reinforcement is adopted.

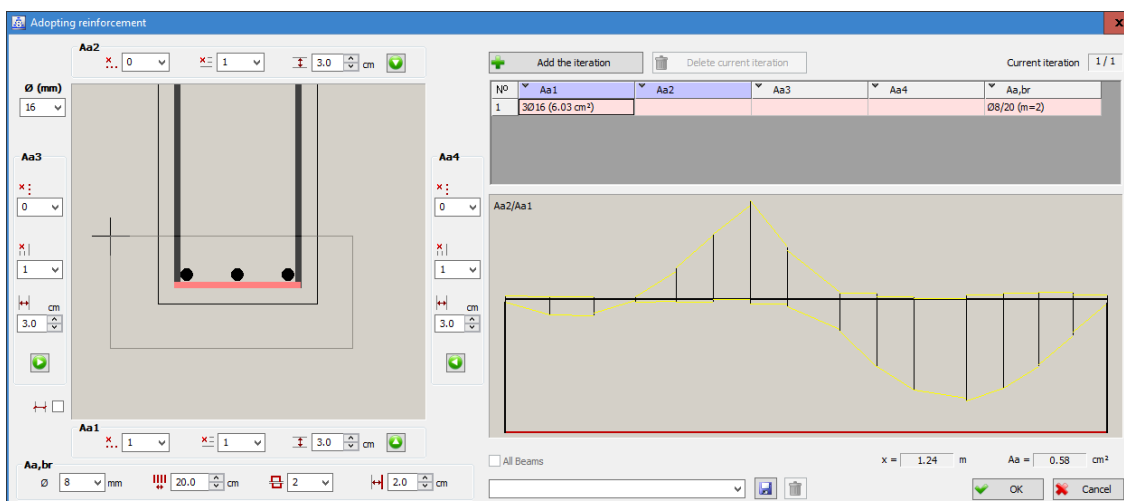


Check boxes whose status determines whether they will be in a dialog quote the cross-section and bar spacings



Check box whose status determines whether the window will display a cross section with all the required data scanned or only an angular cross section will be displayed.





Mass removal of the fitted armature by selecting the window

**Ø (mm)** Edit field in which the diameter of the bar you want to place is assigned. Diameter setting can also be done by selecting from the list, which opens with the mouse on the arrow on the right side of this edit box.

In the sections of the dialogs "**Aa1**", "**Aa2**", "**Aa3**", "**Aa4**", the longitudinal reinforcement is set in the cross-section:



Edit field in which the number of bars is in order. Setting the number of bars can also be done by selecting from the list, which opens by clicking the arrow on the right side of this edit box.



Edit field in which the desired number of rows of armatures is assigned. Setting the number of rows can also be done by selecting from the list, which opens by clicking the arrow on the right side of this edit box.

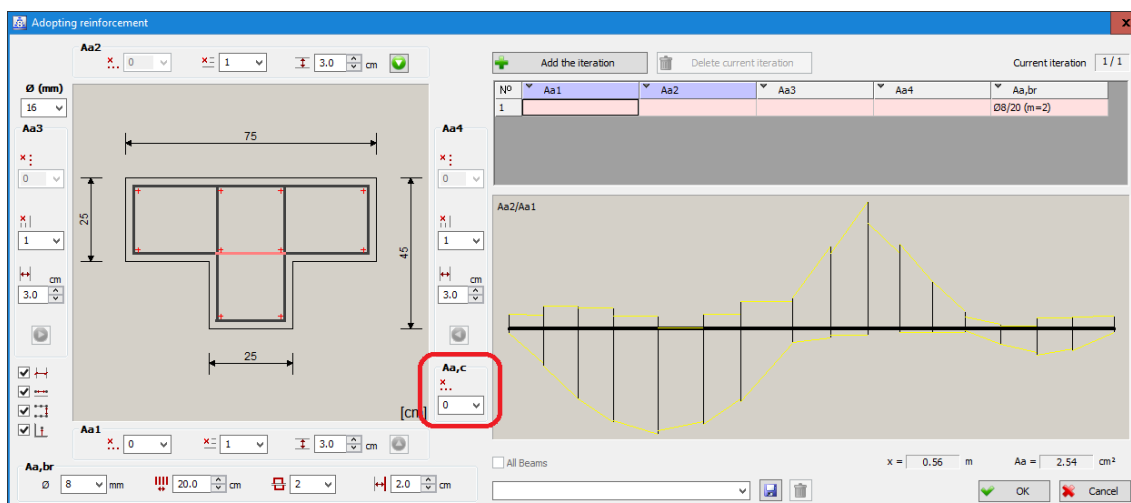


Edit field in which the axle distance between the rows of the reinforcement is assigned. The distance between rows can also be adjusted using the arrows that are on the right edge of this edit field.



In case the user set the data for one armature, for example, "**Aa2**", by activating this button, copying all given data into another appropriate array, in this case "**Aa1**", is performed. We will note that this does not reflect the installed reinforcement, but only defines the position on which the array can be positioned.

In cross-sectional segments not related to the main reinforcement ("**Aa1**", "**Aa2**", "**Aa3**", "**Aa4**"), it is possible to specify the longitudinal structural reinforcement "**Aa, k**":

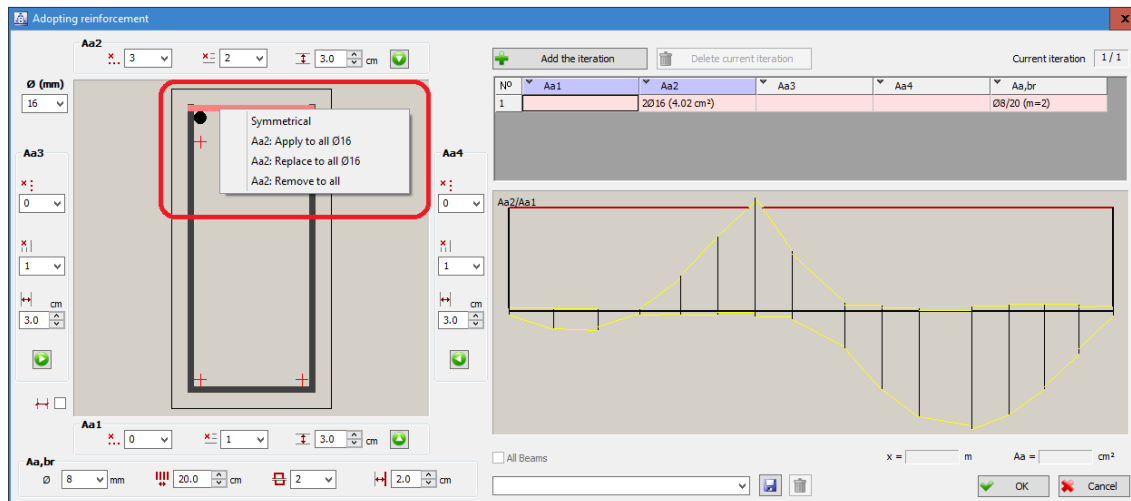


The part of the dialog that deals with specifying the structural reinforcement in cross section



Edit field specifying the number of bars in the row. You can also specify the number of bars by selecting from the Aa list, which opens by clicking the arrow on the right side of this edit box.

If the segment of the stirrup is selected along which the reinforcement is selected, so that it becomes red, and then clicked on the right mouse button, a drop-down menu opens with a series of commands that allow the user to perform symmetrical reinforcement, mass installation of the reinforcement, massive reinforcement in a quick and simple manner replacing the diameter of the reinforcement as well as removing the pre-fitted reinforcement:



Drop-down menu that opens with the right click of the mouse above the segment of the stirrup

### Symmetrical

In the case of symmetrical reinforcement, it is sufficient that only one reinforcement, such as "Aa2", defines the exact position of the bars, while the other, in this case "Aa1", can be automatically mapped. It is enough to select command "Symmetrical" from the drop-down menu and the program will automatically axially map the previously installed armatures to the appropriate segment of the stirrup.

### Apply to all

By selecting this command from the drop-down menu, the program will automatically place the bars over all the crosses belonging to the selected segment of the stirrup. We will note that it is necessary for the user to first specify the diameter of the bar in the edit field "Ø (mm)", which will be assigned to the fitted armature. In the

command name, the program shows in which segment the armature will be placed and which diameter will be assigned, e.g. "Aa2: Apply to all  $\varnothing 16$ ", which means that the Aa2 armature with the diameter of the bar  $\varnothing 16$  will be installed.

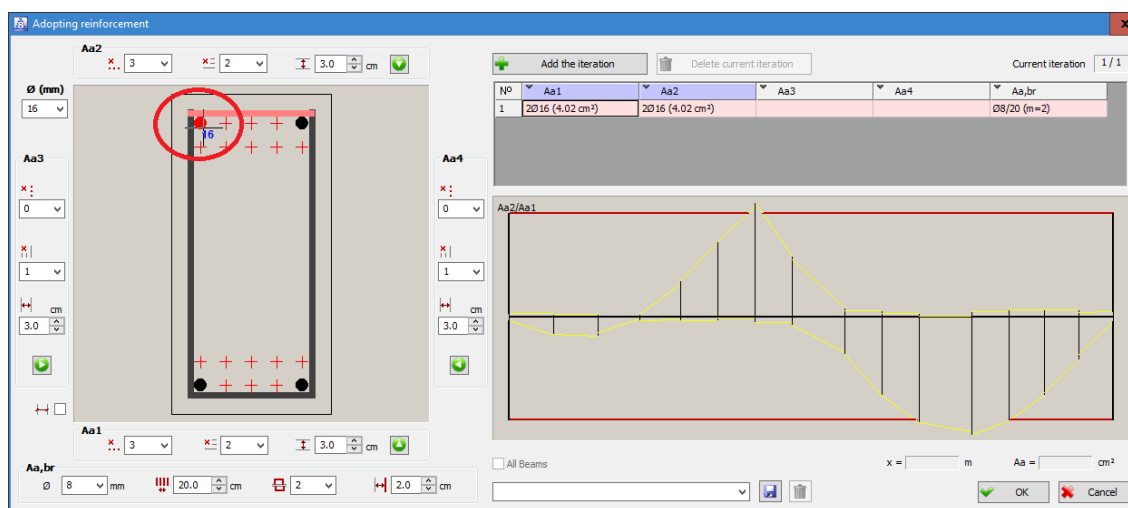
### Replace to all

The user is enabled to replace the diameter in a quick and simple way, already fitted with reinforcement. You need first to set the desired diameter in the edit box " $\varnothing$  (mm)", then select the "Replace to all" command from the drop-down menu, which will give the new diameter to all the placed bars in the selected segment. In the command name, the program shows in which segment the stirrups will be replaced by the diameter of the bar and which diameter will be assigned, for example, "Aa2: Replace to all  $\varnothing 20$ ", which means that the diameter of the bar  $\varnothing 20$  will be assigned to the armature Aa2.

### Remove to all

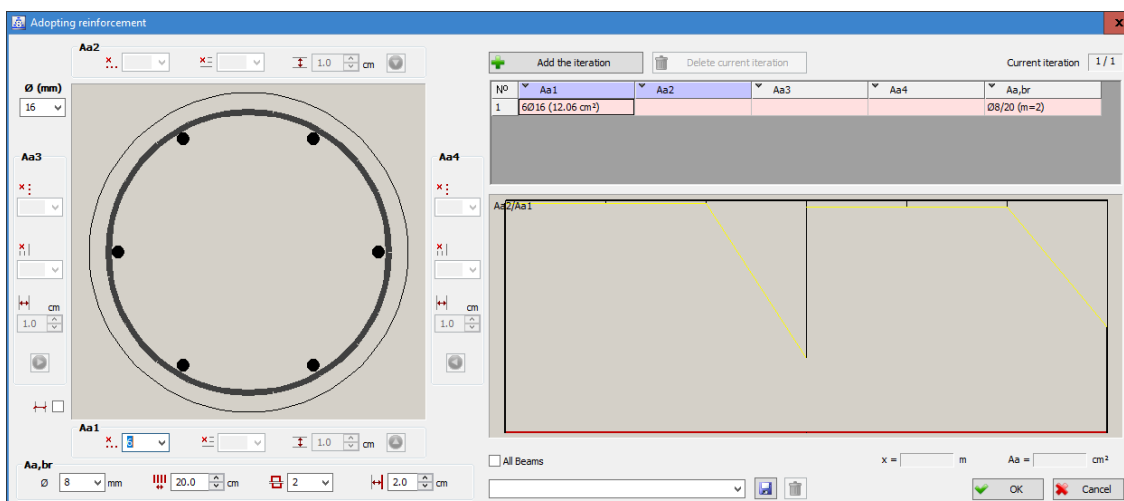
By selecting this command from the drop-down menu, the program will remove the already placed bars from the selected segment, and red shadows will appear in their place. In the name of the command, the program shows which segment of the stirrups will remove the bars, for example, "Aa2: Remove to all", which means the program will remove the installation of the reinforcement bar Aa2.

Since the bars are not quoted in the cross-sectional drawing, it is certainly of interest to know at any time during the work, which diameter of the bars you placed in which place. It is a procedure for identifying the placed rods. So, it is enough to point by mouse in the surroundings of the bar whose diameter you are interested in and it will appear in blue. In this way, the identification of the bars that have not yet been set is done, but only their positions are shown in the cross-section, through the red crosses:




Display the diameter of the bar in which the mouse pointer is placed

When adopting the reinforcement in a circular or tubular cross-section, there is no possibility of setting the position of the bars, but immediately the reinforcement which the program automatically set up appears.







The layout of the dialog for the adoption of the reinforcement in a circular cross-section

In edit fields "Ø (mm)" and  it is necessary to specify the diameter of the bars and the desired number, and they will automatically be set homogeneously in the scope of the section.

#### Setting the stirrups in the cross - section

In the part of the dialog "Aa,br", the assignment of the stirrups in the cross-section is done:

-  Edit field in which the diameter of the stirrup is assigned. Diameter setting can also be done by selecting from the list, which opens with the mouse arrow on the right side of this edit box.
-  Edit field in which the distance of the stirrup is assigned. The distance can also be adjusted using the arrow located on the right edge of this edit field.
-  Edit field in which the cutting of the stirrup is assigned. The task can also be done by choosing from the list, which opens with the mouse arrow on the right side of this edit box.
-  Edit field in which the size of the protective layer is assigned, that is, the distance from the edge of the section to the outer edge of the stirrup. The distance can also be adjusted using the arrows located on the right edge of this edit field.

#### Work with iterations


##### **Add the iteration**

This button adds a new iteration, which gets an ordinal number greater by one than the last iteration (the program automatically takes over the already installed armatures from the last one until then and joins the newly formed iteration).

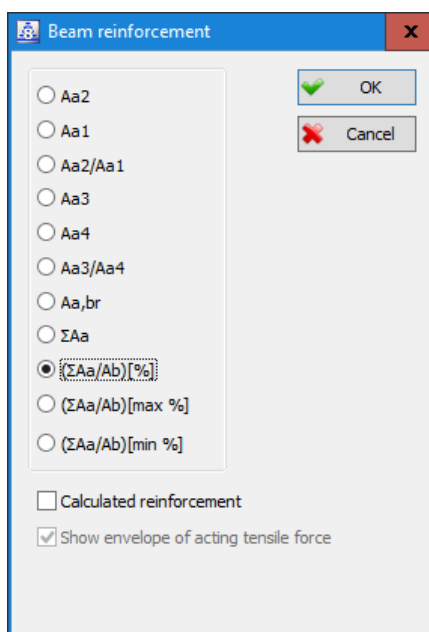
##### **Delete current iteration**

Selecting this button you delete the current iteration

 **Save** Selecting this button, the current state of the parameters in the dialog is saved.

 **Delete** Selecting this button, the current configuration is deleted.

The closed list located to the left of the " " button is intended for selecting some of the previously saved configurations.



Appearance of the dialog box for selecting the diagram to be presented on the drawing

### **$(\Sigma Aa/Ab)[\%]$**

Diagram of percent reinforcement of beams / columns for adopted reinforcement.

### **$(\Sigma Aa/Ab)[max \%]$**

Display of maximum value of percentage of reinforcement on all beams / columns for adopted reinforcement.

### **$(\Sigma Aa/Ab)[min \%]$**

Display of the minimum value of the percentage of reinforcement on all beams / columns for adopted reinforcement.

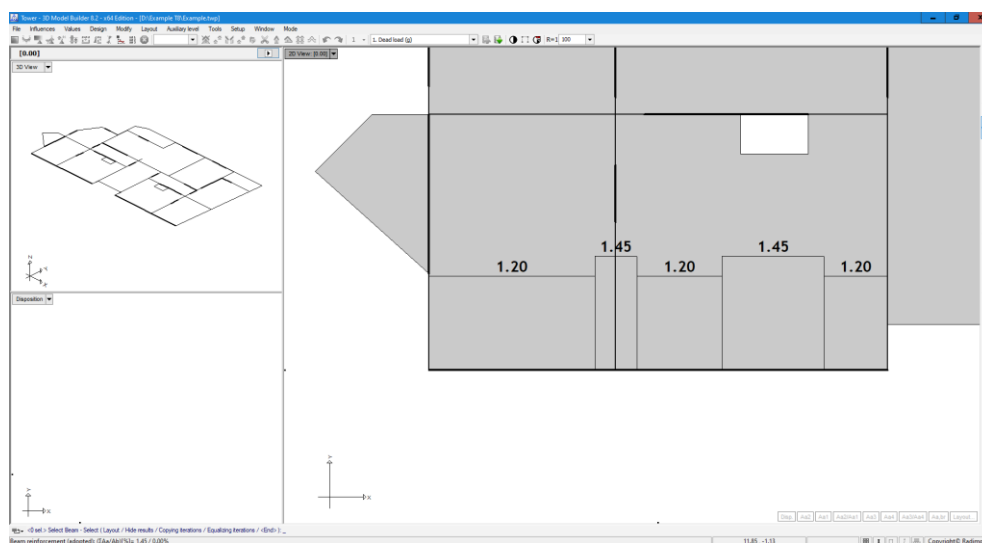
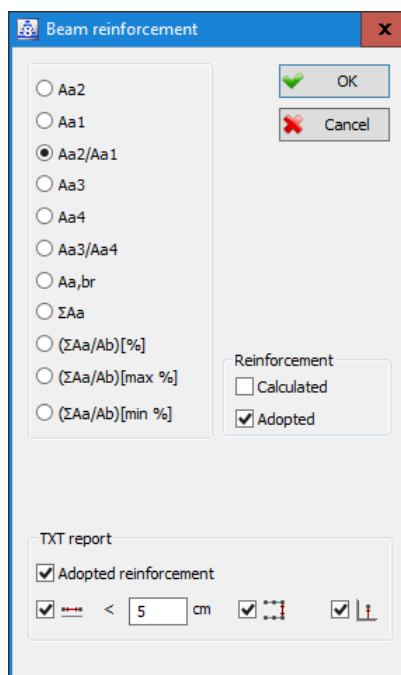


Diagram of percent reinforcement of beams for adopted reinforcement

## 9.4.5 Generating the textual report



Parameter tuning dialogue when creating text reports

The section of the “**TXT Report**” dialog box contains check boxes whose status determines whether bars for the adopted reinforcement will be quoted in the report.



Setting the check box to ON in the beam dimensioning report for adopted reinforcement will quote the clear distance between the bars. Next to this check box is an edit box where the user has the ability to specify the maximum distance between the bars for which the angles will be displayed. This will eliminate the display of excess angles, ie distances between bars larger than the set value.



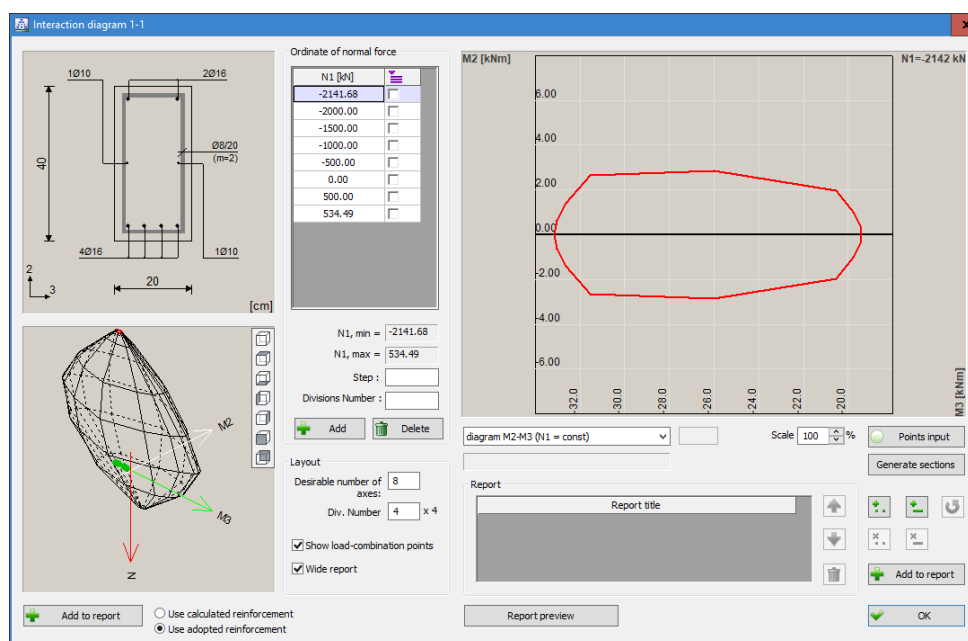
Setting the check box to ON in the beam dimensioning report for adopted reinforcement will quote the clear distance between the bars placed in multiple rows.



By setting the check box to the ON in the dimensioning report for the adopted reinforcement beams, the protective layer, ie the distance from the edge of the section to the bar, will be quoted.

**Vrd,max** – for Eurocode-based regulations, the report dimensioning the beams also lists the maximum values of shear forces that the beam can accept in the direction of local axes 2 and 3 (Vrd,max,2 and Vrd,max,3).

## 9.4.10 Interaction diagram of the beam



Layout of the "Interaction diagram" dialog

### "Diagram M2-M3 (N1=const)"

Since this interaction diagram represents cross sections through a 3D interaction diagram in different planes corresponding to different values of normal force  $N_1$ , in order to know for which plane a diagram is shown, the 2D interaction diagram in the upper right corner shows the ordinate of the normal force.

**Scale** Edit field for setting the text scaling factors on 2D interaction diagrams. In this way, the program allows you to adjust the font size for texts that are displayed on interaction diagrams. The text size can also be adjusted using the arrows that are located on the right edge of the edit field. We will note that the adjusted text size on the interaction diagram will also refer to their presentation in the report.

For the purposes of arbitrary setting the points and sections on the 2D interaction diagram, the program offers a number of commands:

#### Points input

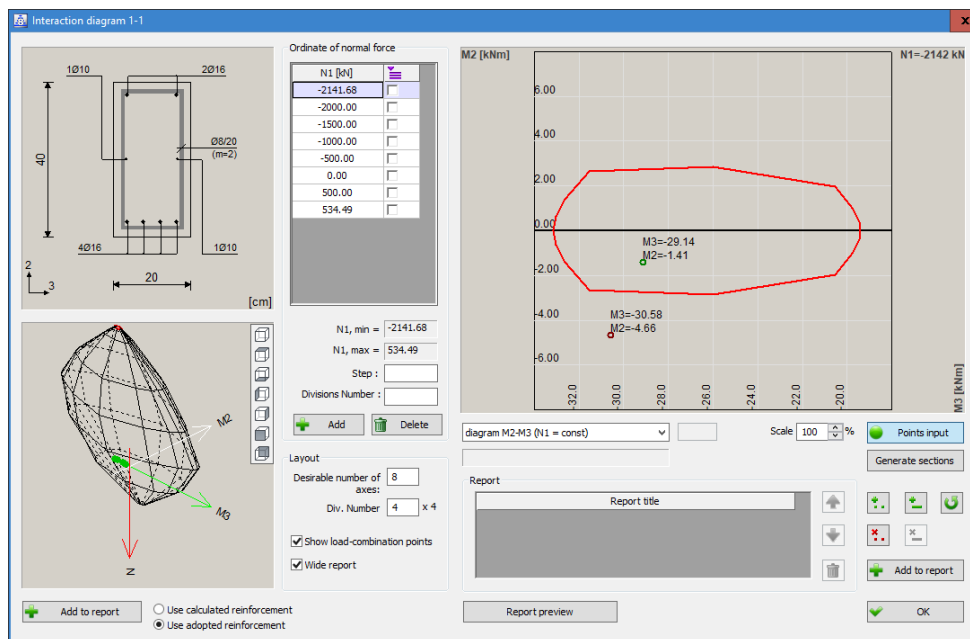
Selecting the button, the user has the ability to set an arbitrary point on a 2D interaction diagram, showing its values. The points are represented by the appropriate symbol, whereas the green colour indicates that the section with its reinforcement can withstand these influences, while the red colour indicates that the section is not able to withstand the influence. Thumbnail with the name of the button, provides the user with information about whether the program is in the process of setting points or not:



the command is started and the user can enter arbitrary points on the interaction diagram.



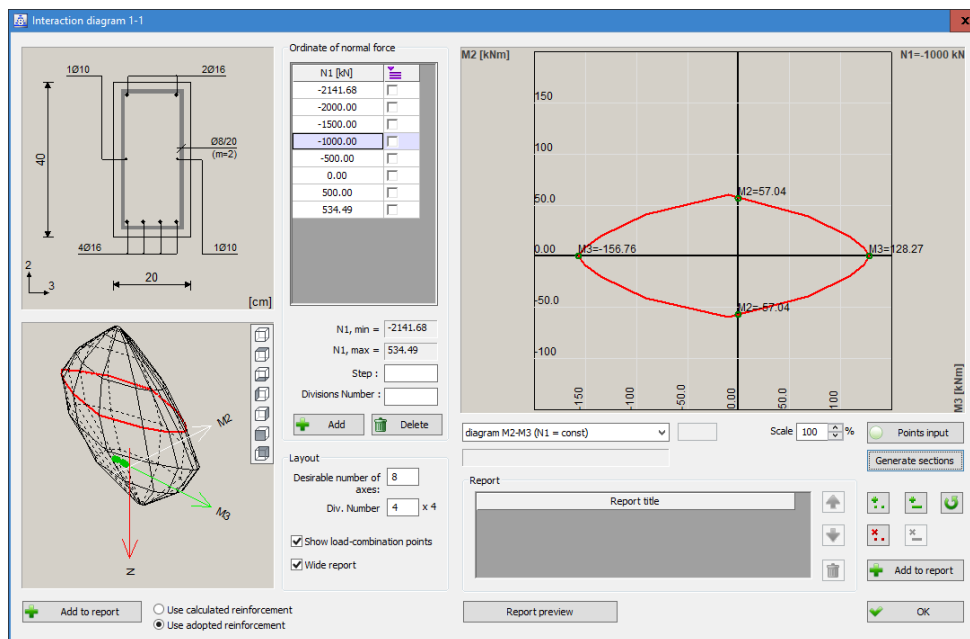
the command is not started.



Layout of the entries entered with your ordinates in the interactive diagram

### Generate sections

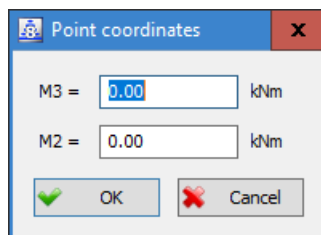
The program allows the automatic generation of points, with the display of their values, at the section point of the interaction diagram with the coordinate axes.



Layout of the generated points on the interaction diagram by using the command "Generate sections"



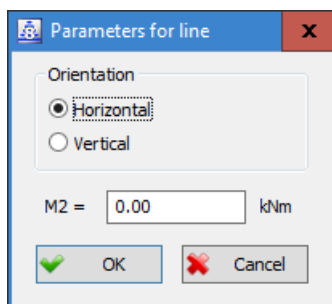
A button which selection opens a dialog where the user has the ability to set the coordinates of the point he wants to display on the interaction diagram:



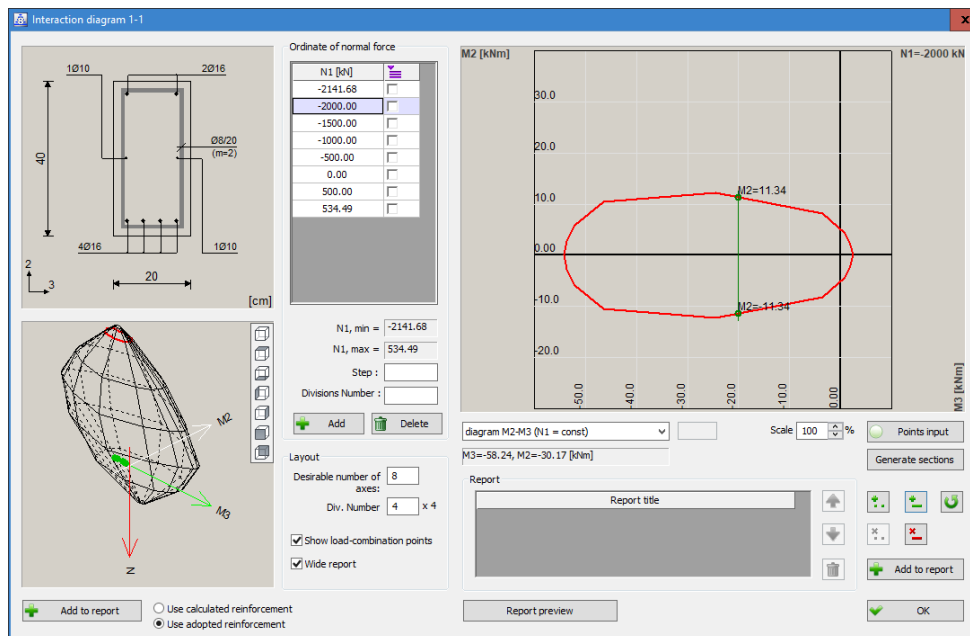
Upon exiting this dialog, the "OK" button will be displayed on the interaction diagram.



The button which selection opens a dialog where the user has the ability to set the line parameters that will be displayed on the interaction diagram:

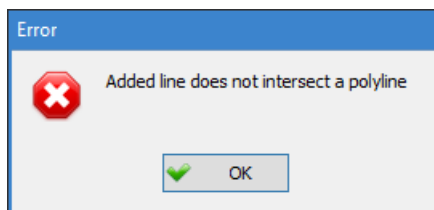


Using the two "Horizontal" and "Vertical" switches, the line orientation is selected, and the corresponding coordinates are entered in the edit field. By going from the dialog to the "OK" button, the interactive diagram will show the assigned line as well as the section points of this line and the interaction diagram.



Layout the generated points in the section of the interaction diagram and the assigned line

If there are no intersecting points of the assigned line and the interaction diagram, after exiting the line parameters dialog, the program will issue the appropriate notice:



The button is removing the last assigned point from the interaction diagram.



The button removing all assigned points from the interaction diagram.



The button removing all assigned sections from the interaction diagram.

We will note that all the points and sections that a user has set in the interaction diagram will also be displayed in the report.

## 9.4.11 Designing the concrete according to EC8 (Capacity design)

### 9.4.11.2 Method of implementing dimensioning of concrete beams and columns according to EC8 EN

#### 2) ANALYSIS

Capacity Design is also enabled when the seismic calculation is performed with the EC8 transverse force method.

#### 4) CHECKING THE CAPACITY OF STRUCTURE FOR DUCTILE BEHAVIOR

Design parameters EC8 (Capacity design)

Ductility class  
Ductility class 'MEDIUM' (DCM)

Overstrength factors  
Beams:  $\gamma_{Rd}$  1.00 Columns:  $\gamma_{Rd}$  1.10

Type of non-structural elements  
Non-structural elements fixed in a way so as not to interfere with structural deformations

	Name	$\alpha$ [°]	q	$\Sigma M_c / \Sigma M_b$	T
1	Sx	0.00	2.600	1.300	1.021
2	Sy	90.00	2.600	1.300	0.824

Frames or frame-equivalent dual systems  
 $k_w = 0.67$   
 $q_0 = 3.90$

Show check 5.5.1.2.1 - (4) / 5.4.1.2.1 - (2)  
 Show check 5.5.1.2.1 - (5) / 5.4.1.2.1 - (3)  
 Show check 5.5.1.2.4 - (2a) / 5.4.1.2.5 - (2a)  
 Show check 5.5.1.2.4 - (2b) / 5.4.1.2.5 - (2b)  
 Show check 5.5.1.2.2 - (1)  
 Show check 5.5.1.2.1 - (1)

Report OK Cancel

Izgled dijaloga za zadavanje parametara proračuna za EC8 (Capacity design)

Part of the dialog “**Overstrength factor**” allows you to set the factor “ **$\gamma$ ,  $R_d$** ” as defined in the regulation EC8 EN, in sections 5.4.2.2 for beams and 5.4.2.3 for columns.

### 9.4.11.5 (EC8 CD) Critical Regions

It is possible to perform multiple edit of critical regions at the same time, by selecting a larger number of beams:

#### Editing of critical regions (Multiple/<End>):

By selecting the “**Multiple**” sub-option from the command line, the beam selection procedure is entered:

<0 sel.> Beams - Select (All / Window / Polygon / Section / eXtras / selection Groups / Resent selection / Deselect / <End>):

After the selection of beams/columns, whose critical areas you want to change, a dialog with the following appearance opens:

Dialog for editing critical regions

The left part of the dialog shows the data of the starting nodes, while the middle part shows the data of the end nodes of the beam. By entering a new value in the “**Current length**” edit field, the “**Default length of critical regions**” will change for all selected beams. By clicking on the “**Default**” button, the manually set critical area is canceled, and the program uses the default lengths during the calculation.

## 9.5 Walls Design (Shear walls)

### Influences Governing the Design

Material

Corner reinforcement: B500A

Main reinforcement: B500A

Concrete grade: C30/37

$a_1 = 15.0$  cm

$a_2 = 15.0$  cm

$a_3 = 2.0$  cm

$d = 300.0$  cm

$b/d = 40/300$  cm

$Ab = 12000$  cm<sup>2</sup>

Results

Med = -1528.03 kNm I+1.50xII

Nd = -202.70 kN I+1.50xII

Td = -299.56 kN I+1.50xII

Separated by modes

Report with reinforcement and influences

Nº	N	T	M
I	-202.4	0.0	0.0
II	-0.2	-199.7	-1018.7
III	0.0	14.8	75.3
IV	0.0	0.0	0.0

Multipliers of seismic forces

$T \times 1.00$   $M \times 1.00$

**Multipliers of seismic forces** - Impacts from multimodal seismic load cases can be magnified by assigning arbitrary multipliers. The check box determines whether multipliers are used, so that when the check box is turned on for change, the following fields become available:

**T x** - with the value given in this field multiplies the transverse force by the multimodal seismic load cases. When the multiplier changes, so will the values of the transversal forces in the table.

**M x** - the value given in this field multiplies the bending moment by the multimodal seismic load cases. When the multiplier is changed, the values of the moments in the table will be changed as well.

Material

Corner reinforcement: B500A

Main reinforcement: B500A

Concrete grade: C30/37

$a_1 = 15.0$  cm

$a_2 = 15.0$  cm

$a_3 = 2.0$  cm

$d = 300.0$  cm

$b/d = 40/300$  cm

$Ab = 12000$  cm<sup>2</sup>

Results

Med = -1528.03 kNm I+1.50xII

Nd = -202.70 kN I+1.50xII

Td = -299.56 kN I+1.50xII

Separated by modes

Report with reinforcement and influences

Nº	N	T	M
I	-202.4	0.0	0.0
II	-0.2	-199.7	-1018.7
III	0.0	29.5	150.5
IV	0.0	0.0	0.0

Multipliers of seismic forces

$T \times 2.00$   $M \times 2.00$

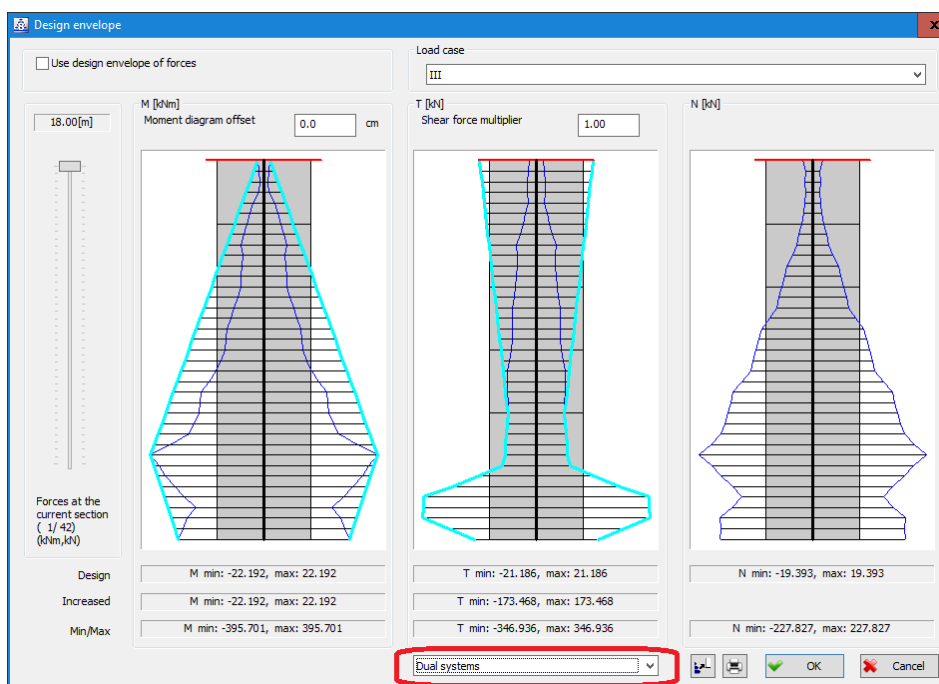
By setting multipliers, the effects of seismic load cases have changed

## 9.6 Dimensioning a series of walls

### Design envelope

#### Type of constructive system

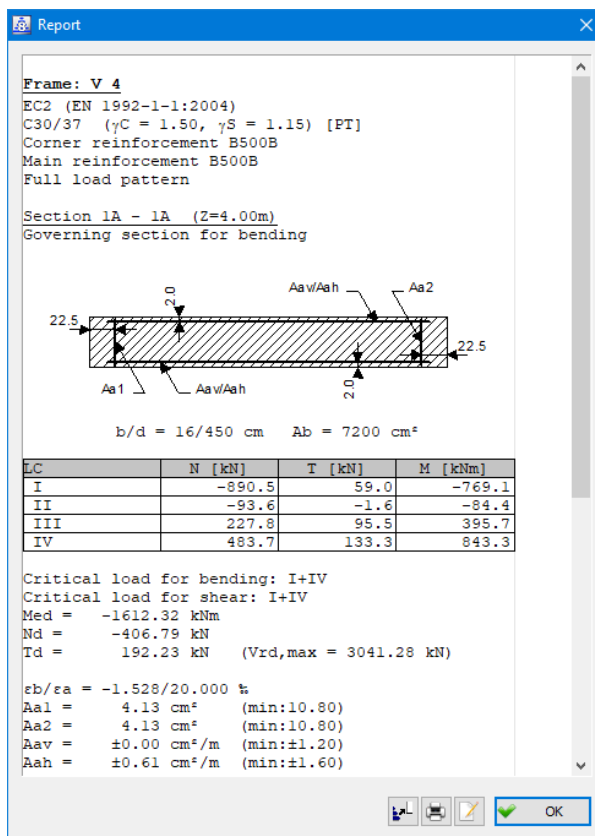
Selecting from the closed list, which is at the bottom of the dialog, determines the type of structural system with the walls. Four options are offered: "**Wall system**", "**Dual systems**", "**Wall systems - squat walls**" and "**Dual systems - squat walls**". For each of them a specific diagram of the shear force envelope is defined.



Dual system is selected

### Creating a text report

By selecting the sub-option "**Report**" from the command line, the program opens a window reserved for the display of a text report, in which the implemented dimensioning is displayed in all sections obtained by the command for dimensioning a array of walls, as well as in all sections obtained by the command for dimensioning walls (see chapter 9.5).

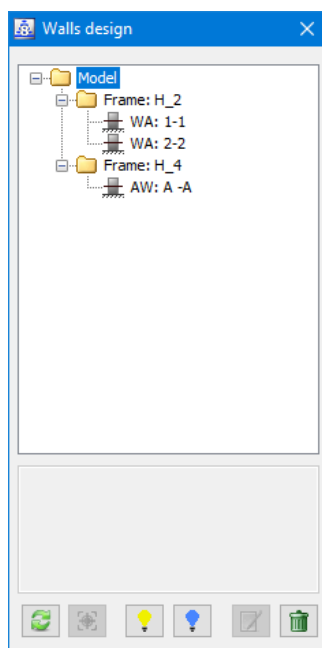


Dialog layout for displaying a text report

Working with this window is exactly the same as with other commands in the program that create text reports.

## 9.7 Review of walls designing

Selecting the “**Review of walls designing**” command, which is located in the module for processing the calculation results in the menu “Dimensioning ► Concrete”, the following dialog opens:



In the dialog, a list of all sections in the model is displayed, in which dimensioning of the walls is performed. Sections are grouped by assemblies. Since the list shows the sections obtained by the seismic wall dimensioning commands, as well as the commands for dimensioning of a series of walls, the dialog with the symbol of the section and the mark of its axis shows the mark indicating which command obtained the intersection:

**WA** - sections obtained by the dimensioning of seismic walls command.

**AW** - sections obtained by the dimensioning of row of walls command.

By selecting a section from the list, in the lower part of the dialog, the adopted, or computably required armatures of a set section will be displayed. In case that a reinforcement is adopted in the section, the reinforcement will be displayed in the dialog. If the reinforcement is not adopted, the required amount of reinforcement will be displayed. The "\*" symbol next to the amount of reinforcement indicates that this is the required reinforcement.



Adding and deleting sections in which dimensioning is performed can result that the list of sections in the dialog no longer corresponds to the condition in the model. By clicking this button, the list is refreshed, ie its alignment with the current state of the intersection in the model.



The activation button in the 2D view is set for the current assembly containing the selected section. The currently selected section in the list will be particularly highlighted in the drawing. Setting in the 2D view of the assembly to which the section belongs can be done by the left double click over its name in the list, as well as selecting the **"Go to assembly"** command from the drop-down menu that opens with the right mouse click over the section name.



Clicking on this button switches on the visibility of the selected sections in which the sizing of the walls was carried out, if it had previously been turned off. If assemblies are selected in the trunk, the visibility of all sections belonging to them will be turned on. Cross-section visibility can also be activated by selecting the option **"Visible"** from the drop-down menu, which is opened by right-clicking on the section name or assembly.



Clicking on this button turns off the visibility of the selected sections. If assemblies are selected in the trunk, the visibility of all sections belonging to them will be excluded. Cross-section visibility can also be turned off by selecting the “**Not visible**” option from the drop-down menu that opens by right-clicking on the section name or assembly.



The button used to edit the section currently selected in the list. If one of the sections selected from the section for selecting seismic walls is selected in the list, clicking the mouse on this button opens the same dialog as in the dimensioning procedure for this section:

**Material**

Corner reinforcement: S500H  
Main reinforcement: S500H  
Concrete grade: C 30

$a_1 = 43.5$  cm  
 $a_2 = 43.5$  cm  
 $a_3 = 2.0$  cm

Vertical concreting

$A_{a1}/A_a = 0.50$   
 $A_{a2}/A_a = 0.50$   
Library  
Known Aav  Greater moment

**Adopting reinforcement**

Aa1:  I  
Aa2:  I  
Aav/Aah:

Control of retained reinforcement:

**Results**

$M_d = -246.37$  kNm 1.35xI+1.05xII+1.50xIII+0.90xIV  
 $N_d = -338.79$  kN 1.35xI+1.05xII+1.50xIII+0.90xIV  
 $T_d = 264.81$  kN 1.35xI+1.05xII+1.50xIV

$d = 869.9$  cm  $b/d = 25/870$  cm  
 $A_b = 21749$  cm<sup>2</sup>

Report with reinforcement

No	N	T	M
I	-237.6	188.2	-238.4
II	-7.1	4.6	-0.9
III	-6.4	-8.5	51.0
IV	-1.0	3.9	-0.1

Settings  
Calculate

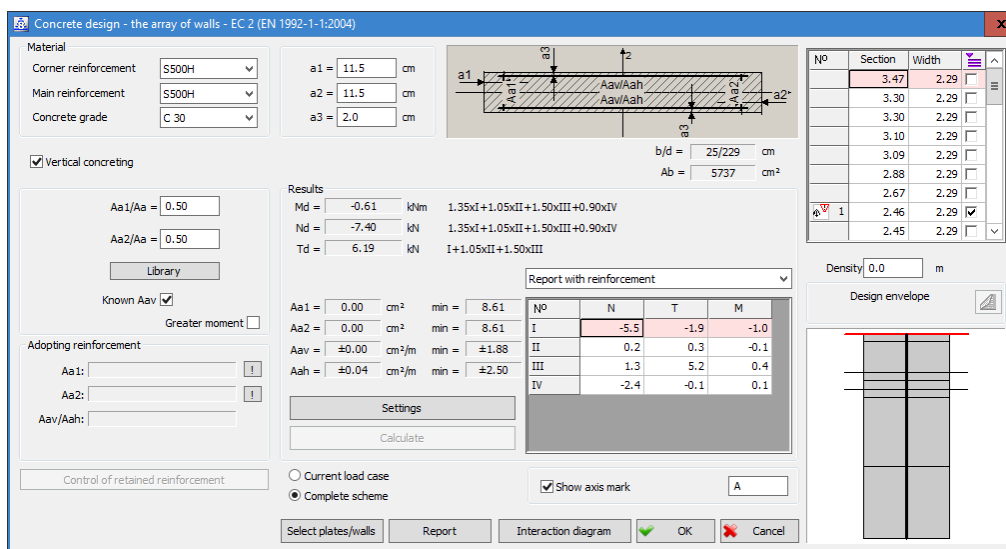
Current load case  
 Complete scheme

Interaction diagram  OK  Cancel

The layout of the dialog for editing the earlier dimensioned section

All offered parameters are now available for changing and implementing a new dimension.

If a list is selected in the list that is obtained using the dimensioning command of a row of walls, a dialog with all the previously given data is opened in the dimensioning procedure for a row of walls:

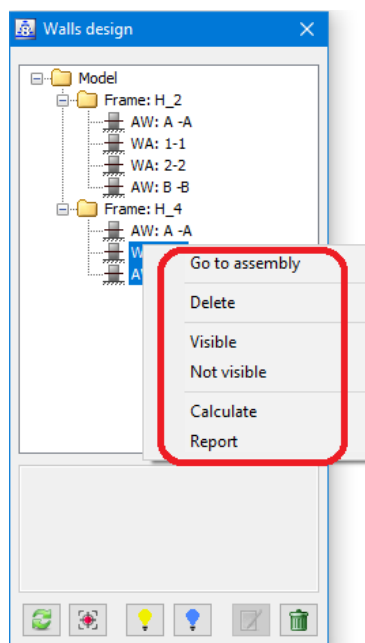


The layout of a dialog for editing a previously dimensioned row of walls



The activated button is currently removed the selected section from the drawing. The section deletion command is also found in the menu that opens with the right mouse click over the name of the section in the list. Since the sections in the list are grouped by assemblies, the right mouse click over the name of the box opens a menu with the option "**Delete all**", which is for deleting all sections from the set assembly.

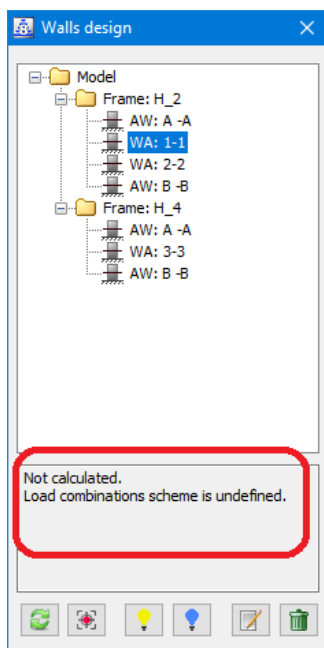
Right-clicking the mouse over a section in the list opens a drop-down menu containing commands for working with the selected section. In addition to the already described commands that can be called via the icons in the dialog, there are also the "Calculation" and "Report" commands.



Right-click drop-down menu  
mouse over the section name in the list

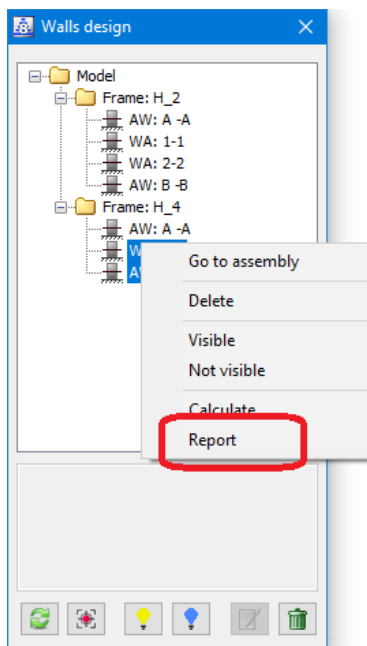
**Calculate** - by choosing this command, the calculation of the selected cross-section is performed without entering the dimensioning command dialog, with the parameters that were set when the command was last used. If several cross-sections are selected in the tree at the same time, the selection of the command will carry out the dimensioning of each of them individually. By selecting the command from the menu that is opened by right-clicking on the name of the assembly or on the "Model" branch, all sections that are in that assembly, that is, all sections in the model, will be dimensioned.

If it is not possible to perform the calculation for one of the selected sections, by selecting it in the lower part of the dialog instead of the reinforcement, a message will be written saying why the calculation was not performed.

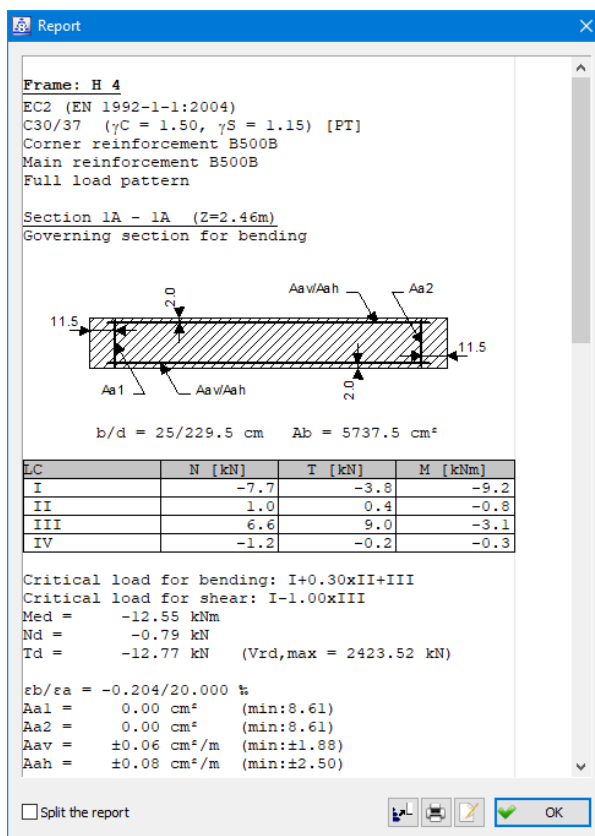


A message indicating why the given Intersection is not dimensioned

By selecting the "**Report**" option from the drop-down menu that is opened by right-clicking the name of the section or assembly, the program opens a window for displaying a text report for all selected and calculated wall dimensioning sections, as well as for all selected and calculated dimensioning of arrays of walls.



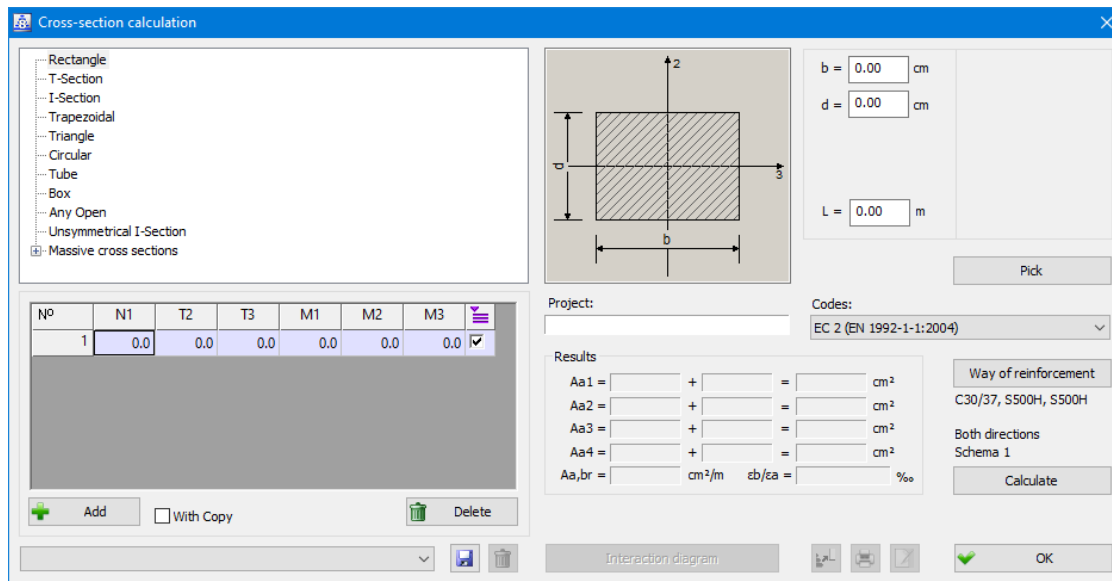
The command to create a text reports in all selected sections



Dialog layout for displaying a text report

Working with this window is exactly the same as with other commands in the program that create text reports.

## 9.8 Cross-section Calculation

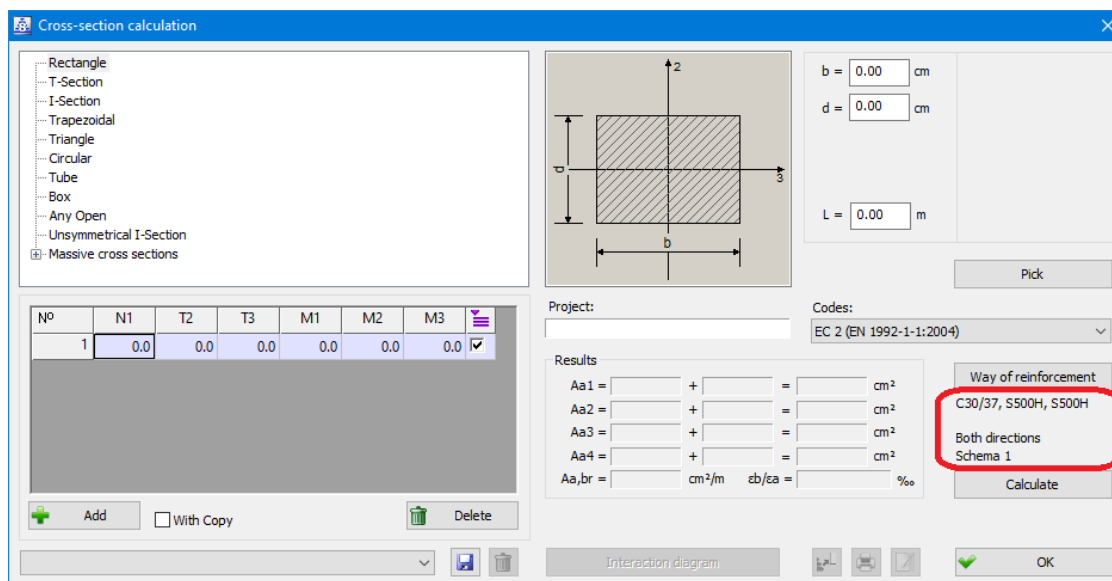


Layout of the analysis of section dialog

L= The program enables the approximate consideration of the influence of buckling in the section calculation. The edit field specifies the length of the beam, which will be used when calculating the buckling lengths.

When a value is set in this edit field, the input data for the buckling calculation should be defined in the dialog that opens by clicking on the "Way of reinforcement" button.

The assigned input data for the dimensioning of the cross-section are displayed in the basic command dialog, so you can at any time see which data analysis will be executed without entering the "Input Data - Local" dialog box.



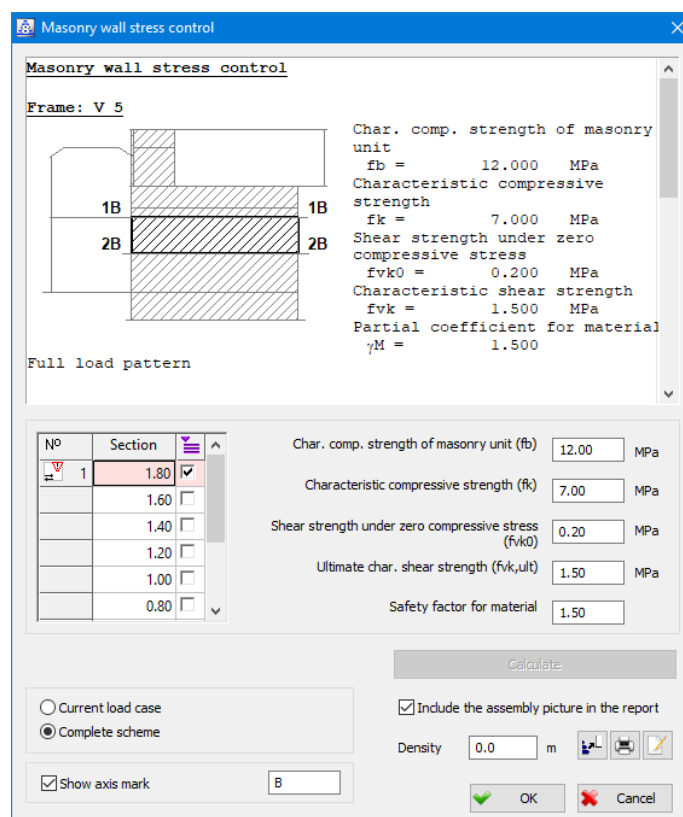
Part of the dialog where the input data for dimensioning the section is displayed

 **Save** The button is saving the current state of the parameters in the dialog.

 **Delete** The button is deleting current configurations.

The closed list located in the lower left corner of the dialog is intended for selecting some of the previously saved configurations.

## 9.9 Masonry wall stress control



Layout of the shear control dialog in built walls

In the upper part of the dialog there is a field showing the complete report of shear control in built walls. When changing any parameters, the report is not updated automatically, but it is necessary to activate the **"Calculate"** button, which brings to a repetition of the calculation and generating a new report.

Below the field in which the report is displayed, there is a table with all sections where shear control is performed in built walls, while all the selected built walls are displayed in the report, as well as all the cut-out sections that the program automatically detects. The number of sections that appears in the table depends on the given density.

Note that in the case of the check box "Control of masonry walls - separated" in the functionality dialog (see chapter 3.5.2), the shear control is performed individually for each of the selected walls. For each of them, the relevant sections are determined and a complete report is written, and the sectional table in the dialog is inactive in that case. The drawing shows a separate axis of dimensioning for each of the selected walls. If this check box is turned off, the shear control is performed simultaneously in all selected walls. The table shows sections of all selected walls, while the report prints the results in the relevant sections that apply to all of them. In this case, a single axis of dimensioning is set in the drawing, so that all walls that are dimensioned together can be edited.

**N°** Column with all sections in which shear control in built walls was carried out. If icons appear in this column, it means that these sections are relevant and that a textual report will be displayed for them.



- means the set section is relevant for bending and shearing



- means the set section is relevant for bending.



- means the set section is relevant for shearing.

All sections which ordinal number are shown in this column will be displayed in the report.

### Section

A column with a section position, in the coordinates x, y, or z if it is in the direction of the global axis, and if it is not, then the position is written locally from the beginning of the coordinate system.



A column with check boxes which status determines which sections will be created, in addition to the relevant ones, to create a text report. All selected sections are numbered by ordinal numbers that appear in the column "N°". Right mouse click over the name of this column opens the menu with the following options:

**All on** - all check boxes will be set to on.

**All off** - all check boxes will be set to off state.

### Density

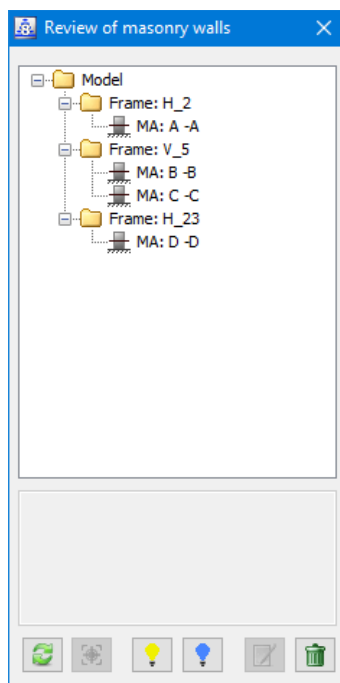
Edit field where the user can subsequently set a new value for the intersection position, with the program automatically executing the dimensioning calculation. If the density of the task is zero, then the program will work with default density. All of the previously mentioned for setting the density by calling the "Density" option from the command line is also valid here.

### Show axis mark

When this check box is set to on, after the exit from the dialog box, the dimensioning axis mark is displayed on the drawing. To the right of this check box there is an edit field in which an arbitrary text can be assigned to the axis mark.

## 9.10 Review of masonry walls

By selecting the command "**Review of masonry walls**", which can be found in the "Design ► Concrete" menu in the module for processing calculation results, a dialog with the following appearance opens:



The dialog shows a list of all sections in the model, in which the tension control of the bricks walls was performed. Sections are grouped by assemblies and marked with the mark "**MA**".

By selecting a section from the list, the status of that section will be displayed in the lower part of the dialog, i.e. whether the section is calculated and if so, whether the stress is exceeded or not.



Adding and deleting sections where stress control is performed in masonry walls can cause the list of sections in the dialog to no longer match the state in the model. By clicking on this button, the list is refreshed and synchronized with the current state of the section in the model.



A button which, when activated, in 2D view window, sets the assembly that contains the masonry wall on which selected cross section refers. The currently selected section in the list will be highlighted in the drawing. Setting as current 2D view of the assembly to which the section belongs can be performed by left double-clicking the mouse over its name in the list, as well as by selecting the "**Go to assembly**" command from the drop-down menu that opens with the right mouse click over the name of the section.



Clicking on this button turns on the visibility of selected sections in which stress control was performed, if it was previously turned off. If assemblies are selected in the tree, the visibility of all sections belonging to them will be turned on. Section visibility can also be turned on by selecting the "**Visible**" option from the drop-down menu that is opened by right-clicking the name of the section or assembly.



Clicking this button turns off the visibility of the selected sections. If assemblies are selected in the tree, the visibility of all sections belonging to them will be turned off. Section visibility can also be turned off by selecting the **"Not visible"** option from the drop-down menu that opens by right-clicking the name of the section or assembly.



A button used to edit the section currently selected in the list. Activating this button opens the same dialog as in the dimensioning procedure of that section:

Masonry wall stress control

Frame: V 5

Char. comp. strength of masonry unit  
fb = 12.000 MPa  
Characteristic compressive strength  
fk = 7.000 MPa  
Shear strength under zero compressive stress  
fvk0 = 0.200 MPa  
Characteristic shear strength  
fvk = 1.500 MPa  
Partial coefficient for material  
 $\gamma_M = 1.500$

Full load pattern

No	Section
1	1.80
	1.60
	1.40
	1.20
	1.00
	0.80

Char. comp. strength of masonry unit (fb) 12.00 MPa  
Characteristic compressive strength (fk) 7.00 MPa  
Shear strength under zero compressive stress (fvk0) 0.20 MPa  
Ultimate char. shear strength (fvk,ult) 1.50 MPa  
Safety factor for material 1.50

Calculate

Current load case  
 Complete scheme

Include the assembly picture in the report

Density 0.0 m

Show axis mark B

OK Cancel

All offered parameters are now available to change and implement new stress control (see chapter 9.9).



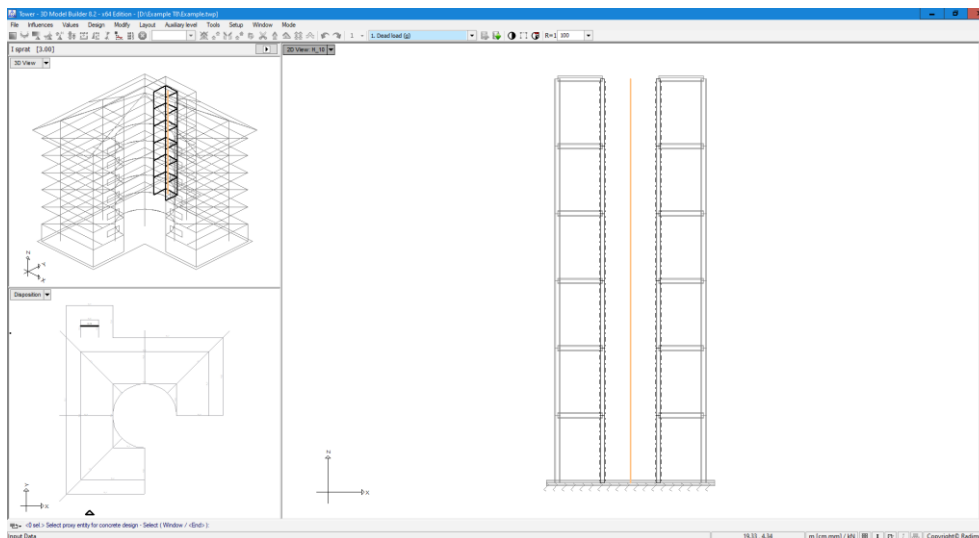
A button which, when activated, removes the currently selected section from the drawing. The command to delete a cross section is also in the menu that is opened by right-clicking the name of the cross section in the list. Given that the sections in the list are grouped by assemblies, right-clicking the mouse over the name of the assembly opens a menu with the option **"Delete all"**, which is used to delete all sections from the given assembly.

## 9.11 Design of proxy entity

This command can be used to jointly design several different structural elements that make up one whole, such as an elevator core. Both the surface (slabs / walls) and line (beams / columns) entities associated with the proxy entity can be design together. The Design of proxy entity command is located in the result processing module within the drop-down menu "Design

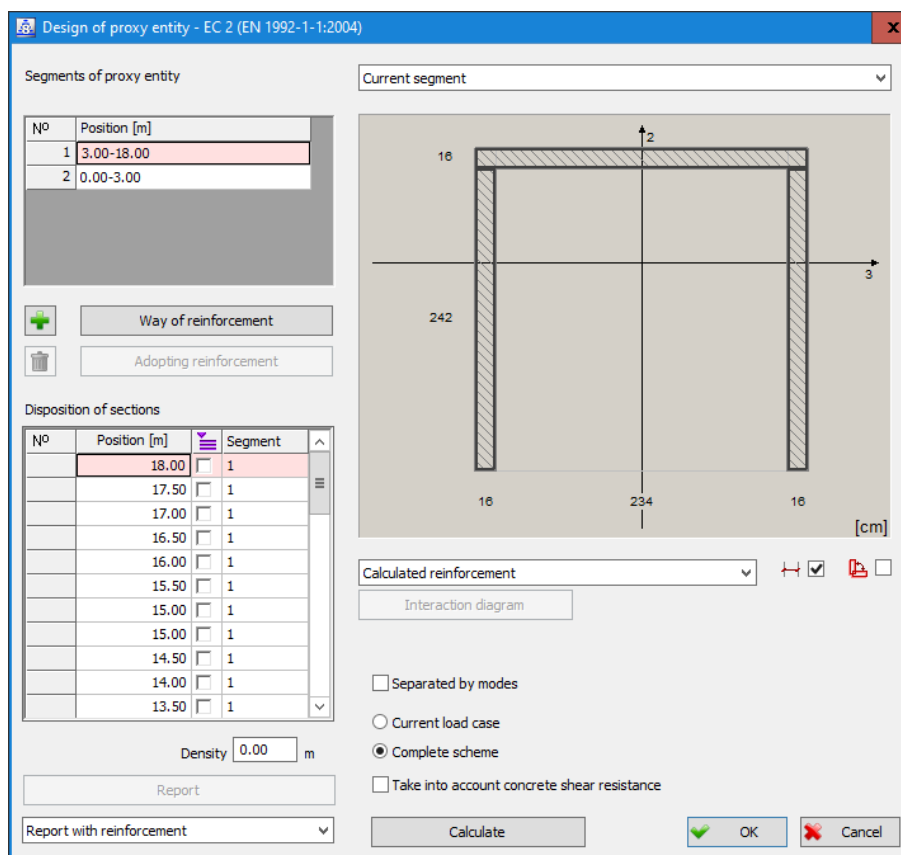
► Concrete". Selecting it from the command line requires you to select the proxy entity you want to design:

**<0 sel.> Select proxy entity for concrete design - Select (Window / <End>):**



A proxy entity has been selected and joined by elevator core elements

After the proxy entity is selected, a dialog opens which specifies all the necessary input data for design, performs concrete design and reinforcement adopting, as well as creating design report:





Design of proxy entity dialog

**Segments of proxy entity** - If the cross section of the entities associated with it changes along the proxy entity, the program automatically divides the proxy entity into segments with different cross sections and displays them in this table. Table columns have the following meaning:


**N<sup>o</sup>** - Column in which the ordinal numbers of the proxy entity segments are printed

**Position** - Column showing the beginning and end of each of the proxy entity segments. If the proxy entity is parallel to one of the global axes, the values shown are the coordinates of the projection of the beginning / end of a given proxy entity segment on that axis. If this condition is not met, the distance / start of each proxy entity segment from the start point of the proxy entity is printed.

**Disposition of sections** - This table shows all cross sections where design of proxy entity will take place. Table columns have the following meaning:

**N<sup>o</sup>** - Column in which the ordinal cross-sectional numbers for which the report is generated are printed. Also, in this column, the relevant sections are marked with the corresponding icons. The relevant bending cross section is indicated by an icon , while the relevant shear cross section is indicated by an icon .

**Position** - A column in which the position of each section is printed. The values shown are subject to the same rules as the values in the column of the same name in the "Segments of proxy entity" table.

 - Setting the checkboxes on this column to ON, selects the sections for which the design report will be generated. For the relevant sections, the report is always generated, so they do not have a check box in this column.

**Segment** - Column in which for each section is printed the serial number of the proxy entity segment to which it belongs.

**Density** - Edit field specifying the distance between sections where design of proxy entity is performed. The larger the specified gap, the smaller the number of sections, and therefore the shorter the duration, and vice versa. If set to 0, the program will determine the number of sections based on the density of the finite element network.

**Current Segment** - When this option is selected from the closed list, the figure in the dialog box shows the cross-sectional view of the selected proxy entity segment.

**Current section** - When this option is selected from the closed list, the figure in the dialog shows the cross section layout currently selected in the "Disposition of sections" table.




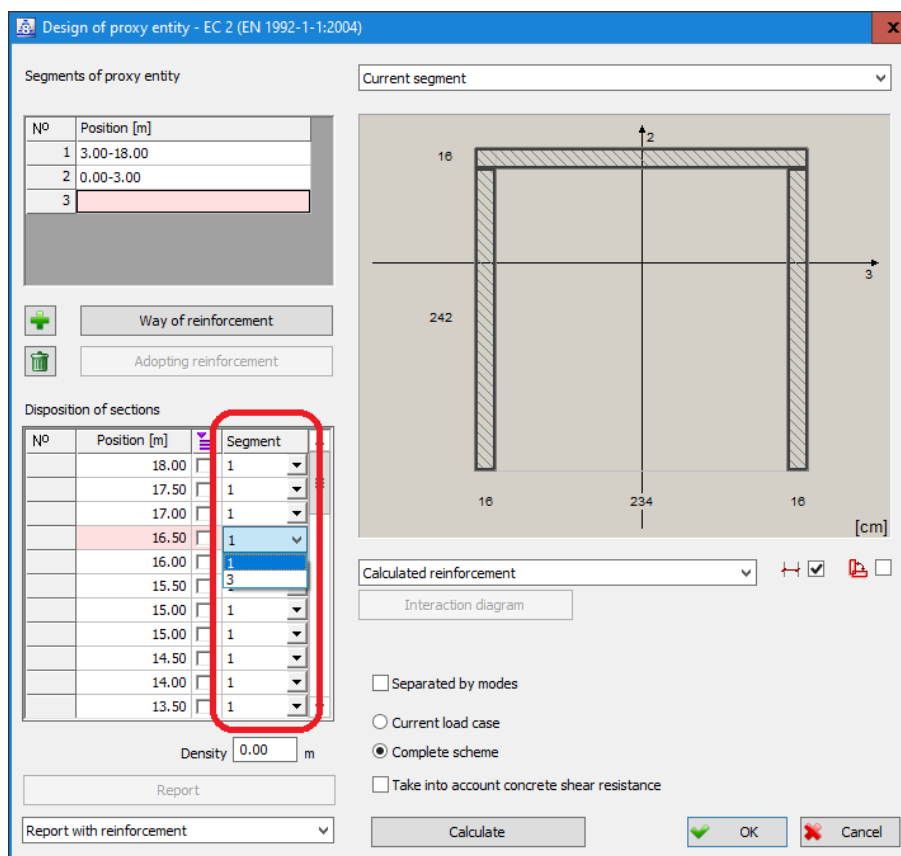
- Check box which condition determines whether or not the cross section of the image in the dialog is quoted.




- Check box by which the cross section is rotated by 90<sup>o</sup>. It is most often used for cross sections that have a larger dimension in one direction than the dimensions in the other direction.



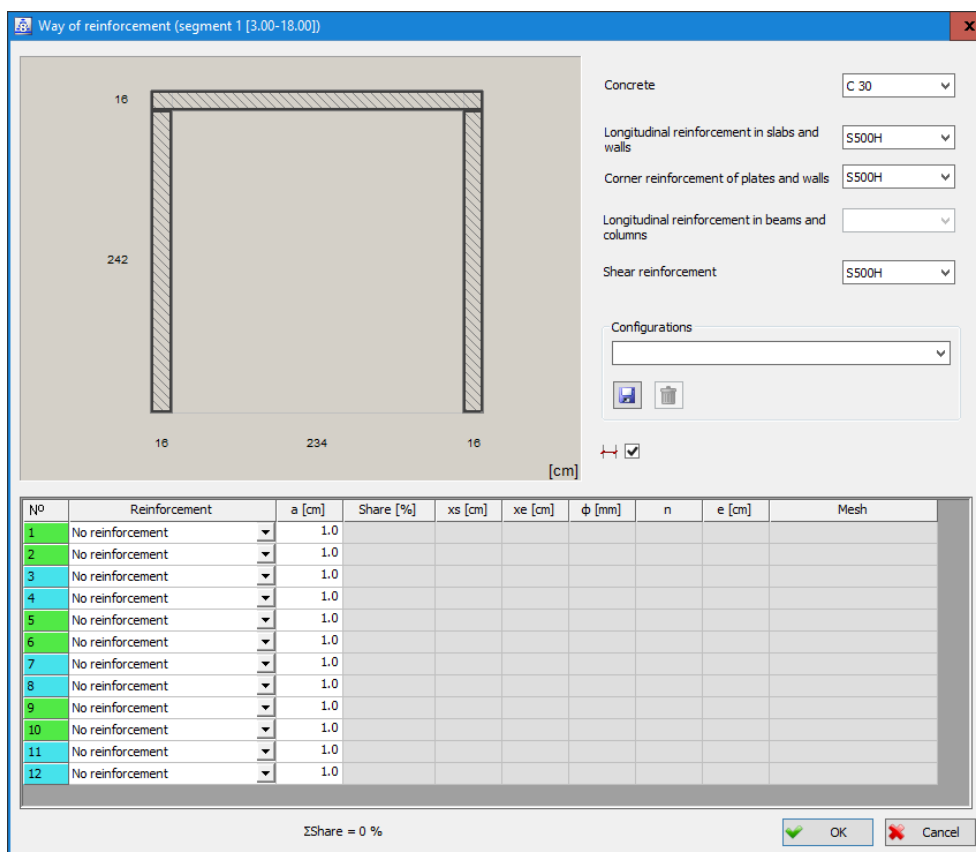
- If there is a need for different ways of reinforcing the same cross section, new reducer segments can also be added manually. Before adding a new segment, you should pay attention to which segment is currently selected, since only the sections that belong to it will be assigned to the new segment. Clicking on a field  inserts a new segment into the "Segments of proxy entity" table, and in the "Disposition of sections" table, in the "Segment" column, arrows appear by which a new segment can be assigned to a section. Clicking a mouse over one of the arrows opens a drop-down menu with the sequence numbers of the proxy entity segments that can be assigned to a given section:



Drop down menu from which to select the proxy entity segment to which the given cross section is assigned

 – By clicking this button, the currently selected proxy entity segment is deleted from the table. Only segments that have been manually added can be deleted.

**Way of reinforcement** - Clicking on this field opens a dialog where you specify all the parameters necessary for design of proxy entity current segment:



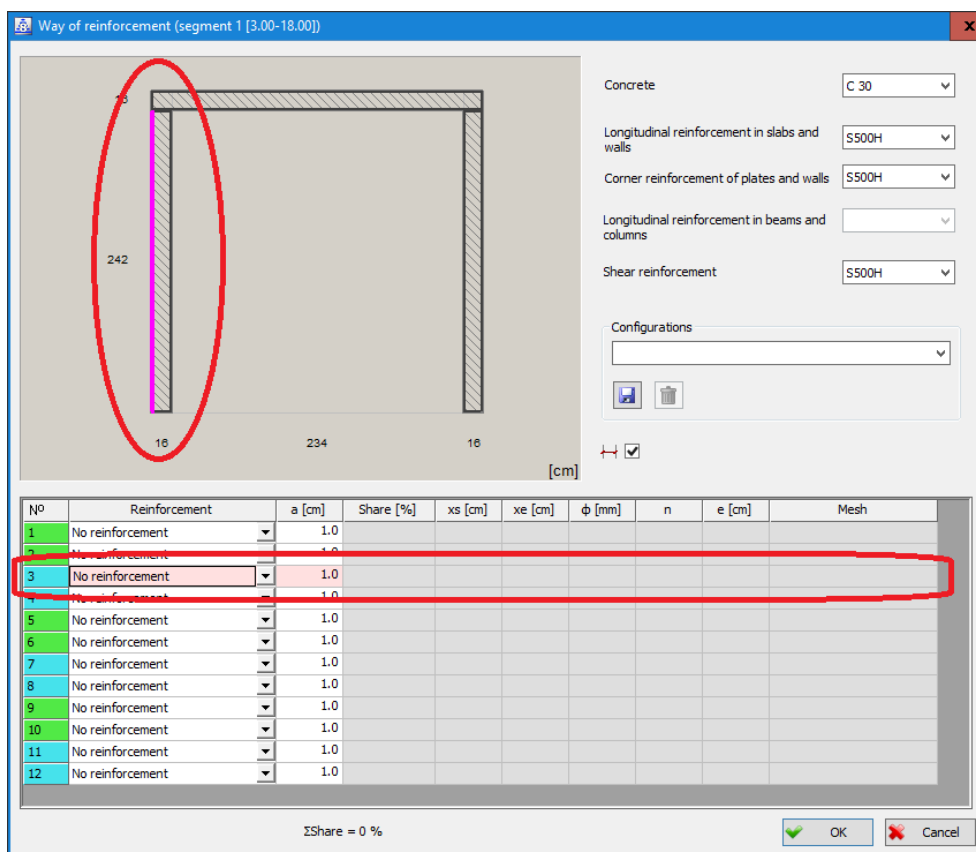
Dialog for defining the method of reinforcement of the current proxy entity segment

The upper left part of the dialog shows a cross-sectional view of the current proxy entity segment. The right part specifies the characteristics of the material, by selecting from closed lists.



- Check box which condition determines whether or not the cross section of the image in the dialog is quoted.

At the bottom of the dialog is a table showing all the lines that make up the cross section of a given proxy entity segment. By clicking the mouse over a row in the table, it becomes current, and the corresponding line in the cross-section drawing is marked with a special color. Selecting the desired line can also be made directly in the drawing by clicking the mouse over it, whereby the corresponding row in the table will automatically become current.

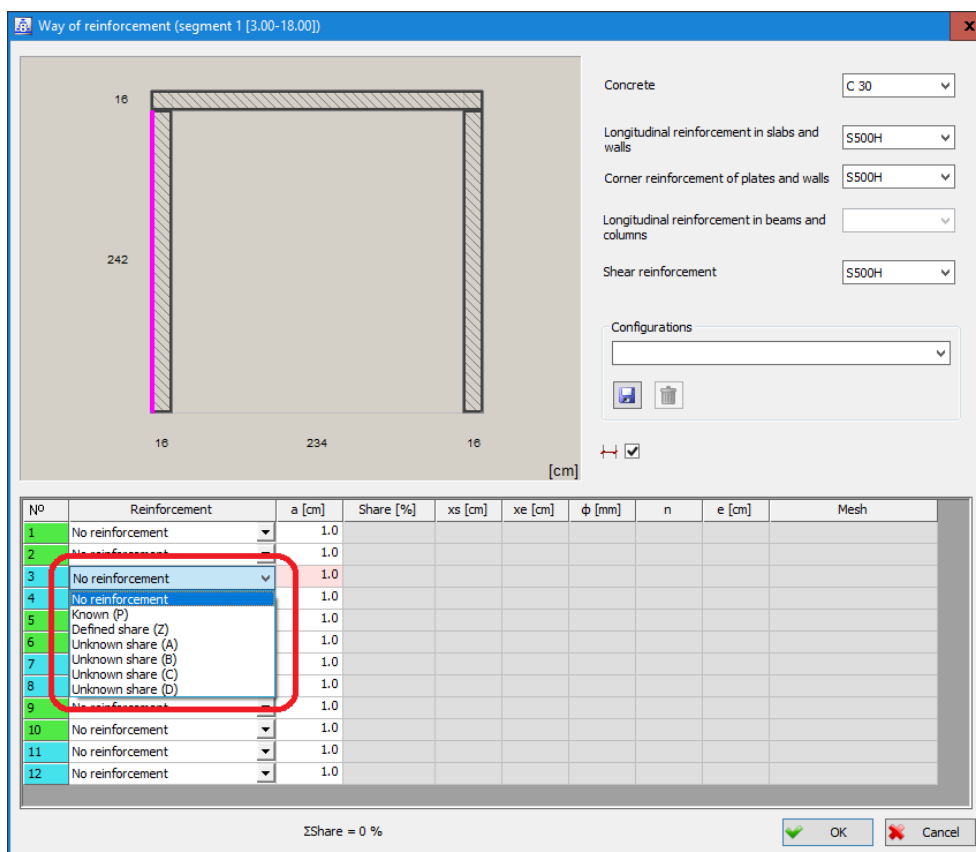


Clicking on the drawing selected the left edge of the cross section, and at the same time in the table as the current one is marked with row number 3

It is possible to define for each cross-section line whether a reinforcement is placed along it, and if it is to be fitted one of the offered methods of reinforcement must be selected. The columns in the table have the following meaning:

**N°** – The ordinal numbers of the intersection lines

**Reinforcement** - Clicking on the cells in this column opens a drop-down menu with the reinforcement options offered:



Contents of the list of reinforcement options

**No reinforcement** - Selecting this option indicates that no reinforcement is placed along the selected line.

**Known (P)** - Option indicating that pre-known reinforcement is placed along the selected line. The reinforcement is set in the corresponding columns of the table, which become available after selecting this option. Known reinforcement can be assigned to all gear reducer lines. In this case, the program does not calculate the required reinforcement, but only controls the known reinforcement.

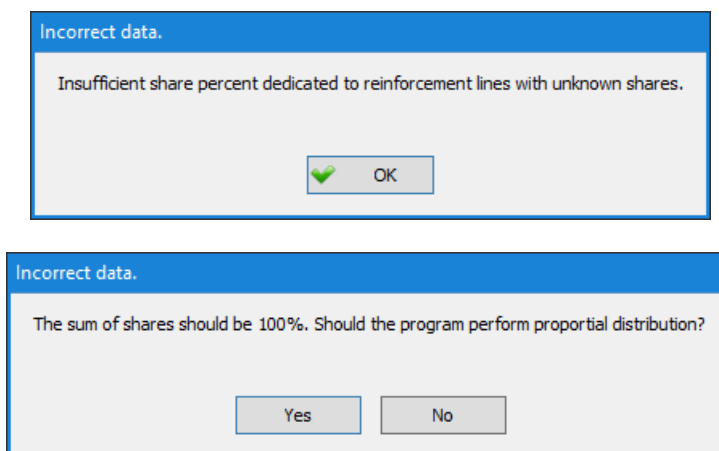
**Defined share (Z)** - An option indicating that an reinforcement is placed along the selected line which surface relative to the total unknown reinforcement (that is, all reinforcement in cross section except the reinforcement given as Known (P)) will have a percent participation equal to the data given in the column "Share".

**Unknown share** - This option is used for situations where a reinforcement is desired on a particular line but its surface or the ratio of its surface to the total unknown reinforcement is unknown in advance. In this case, the program will automatically determine the optimum participation for that line of reinforcement. There may be at least two and no more than four variation groups (A), (B), (C) or (D) in one section. Otherwise, the program will display a corresponding notification when attempting to exit the dialog. It is possible to assign two groups (A and B), three groups (A, B and C) or all four groups.

**a** - Column specifying the protective layer for a given reinforcement line.

**Share** - A column specifying the percentage of the total required reinforcement that will be placed along a given line when given the "Defined share (Z)" option in the "Reinforcement" column. If there are no "Unknown share" reinforcement lines, the sum of the given percentages for all intersection lines must be 100%. If there is an "Unknown share" line, the sum of the given percentages for all intersection lines must be less than 100%. A difference of up to 100% will be assigned to the "Unknown share" reinforcement lines. The current sum of the set percentages

is printed along the bottom edge of the dialog. If any of the conditions is not met, the program will issue an appropriate notification when exiting the dialog:



Selecting the **"Yes"** button instructs the program to automatically redistribute participation, respecting their current relationship, so that the sum is 100%, after which it will exit the dialog. Selecting the **"No"** button denotes leaving the dialog box and re-manually redistributing participation so that the condition is satisfied.

**xs** – By entering a positive value in this field, an extension can be made, while entering a negative value can shorten the start of the line along which the reinforcement is placed. In this way it is easy to adjust the placement of the reinforcement at the points where different entities in the proxy entity are connected. Note that the field is only available for reinforcement lines belonging to walls/slabs.

**xe** – By entering a positive value in this field, an extension can be made, while entering a negative value can shorten the end of the line along which the reinforcement is placed. Note that the field is only available for reinforcement lines belonging to walls/slabs.

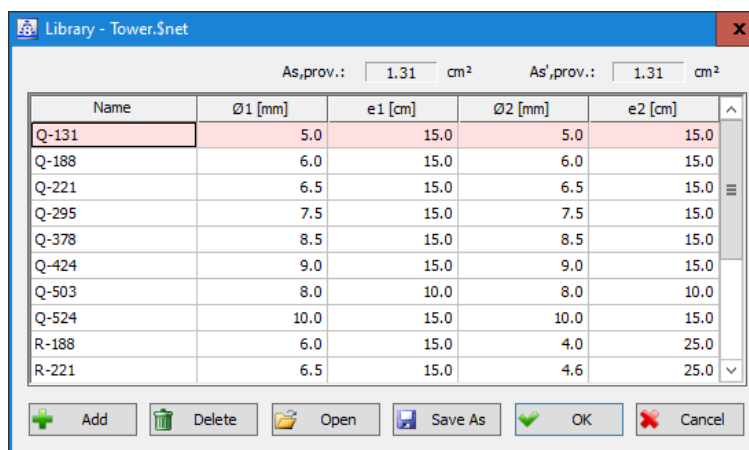
**ϕ** – The fields in this column are only available for lines that are assigned a known reinforcement and serve to define a cross-section of that reinforcement. Clicking on the box opens a list from which one can select one of the offered diameters.

**n** – The fields in this column are only available for lines that are assigned a known reinforcement and that belong to the beams. They are also available for lines belonging to the slabs, along which angular reinforcement is installed. They specify the number of bars of known reinforcement that are placed along a given cross-section line.

**e** – The fields in this column are only available for lines that are assigned a known reinforcement, and at the same time they belong to the slabs and longitudinal reinforcement is placed along them. They specify the distance between the bars of the known longitudinal reinforcement.

**Mesh** - The fields in this column are only available for lines that are assigned a known reinforcement, and at the same time they belong to walls/slabs and longitudinal reinforcement is placed along them. Right-clicking on the box in this column opens a drop-down menu with two options:

**Library** – An option by which a reinforcement mesh can be placed along a given cross-section line. Its choice opens a dialog of the following appearance:

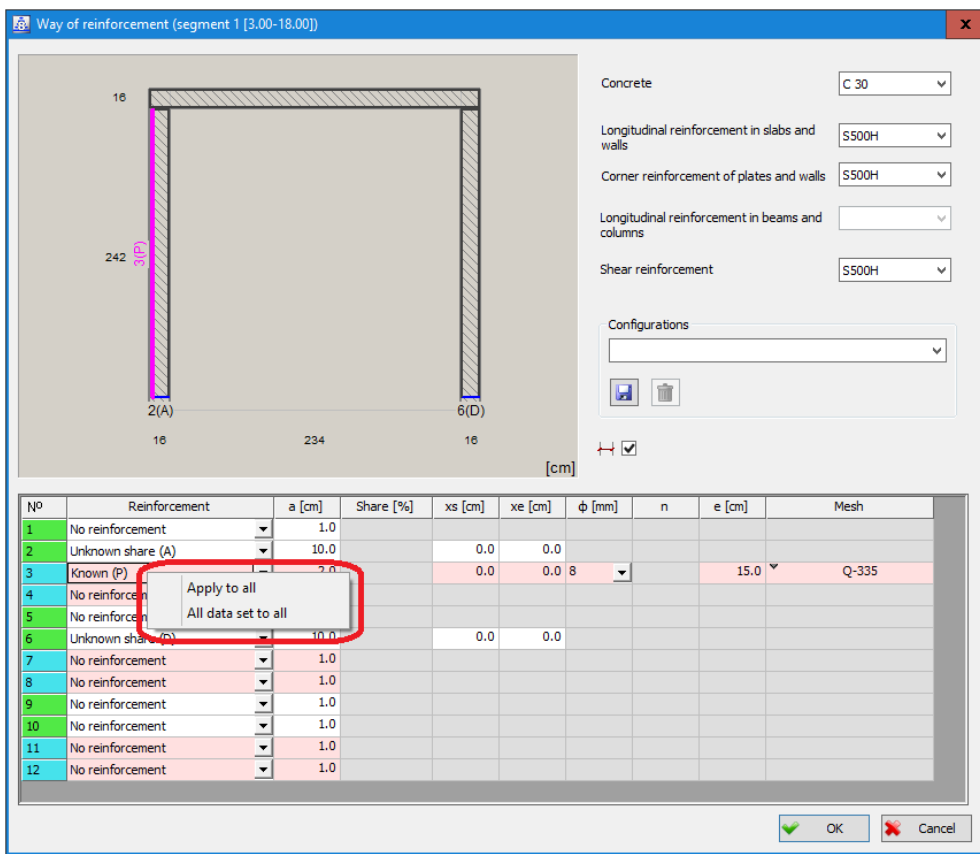


Mesh Selection Dialogue

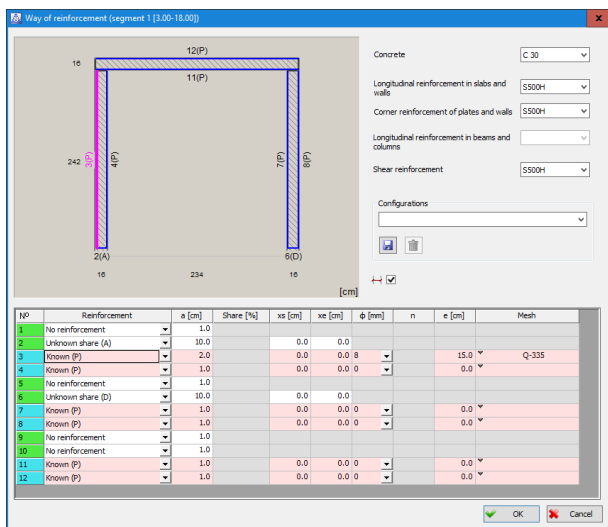
This dialog is the same as the dialog that opens when the "Library" command field is activated in the slabs reinforcement adoption procedure. By selecting the desired armature and activating the "OK" command field, the program returns to the basic dialog and retrieves the data selected from the library for the known longitudinal armature.

**Undo** - By selecting this option, previously adopted longitudinal reinforcement can be canceled.

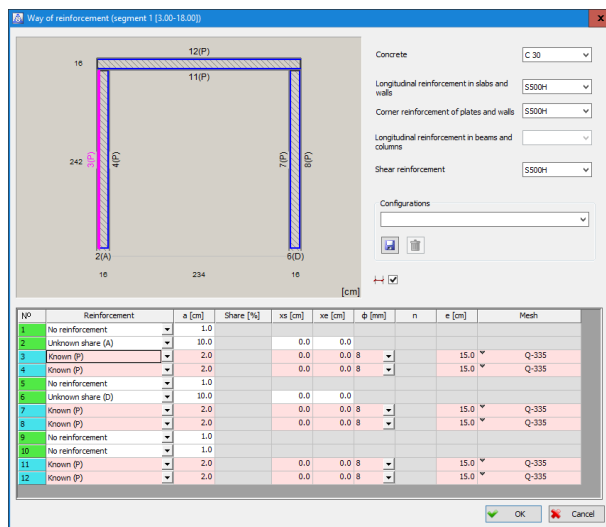
The program allows multiple selection of the section lines in the table, as well as the mass assignment of the desired method of reinforcement and all other data specified in the table, currently selected section lines. It is necessary to give one line of intersection some information, for example, the method of reinforcement, and then from the drop-down menu, which opens by right-clicking the cell with the mode of reinforcement, select the command "**Apply to all**". This way, all selected cross-section lines will be assigned the selected reinforcement method. In addition to the reinforcement method, the user is allowed to assign all the data corresponding to the selected reinforcement method to all selected cross-section lines in a quick and easy way by selecting the "**All data set to all**" command from the drop-down menu.



Drop down menu that opens to the right mouse click



After the "Apply to all" command, the reinforcement method information is assigned to all selected cross-section lines



After the "All data set to all" command, all data is assigned to all selected cross-section lines

**Configurations** - Selecting from the closed list for the current one can set some of the previously recorded configurations of the proxy entity reinforcement. The list shows only those configurations that were recorded for proxy entity with the same geometry as the current segment of the proxy entity being dimensioned.



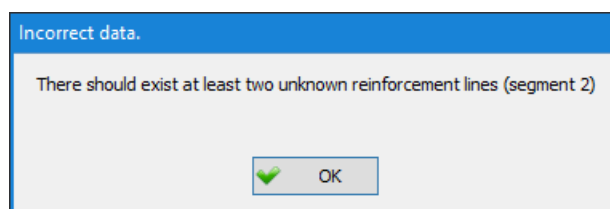
Clicking on this button saves in the configuration file, under the name you entered, all the reinforcement data specified in the dialogue.



By clicking this button, the currently selected reinforcement mode configuration is deleted from the list.

If any of the given data for the current segment design is incorrect, the program will issue an appropriate notification when exiting the dialog.

Reinforcement method information must be specified for each proxy entity segment separately. If not specified, the program will display a corresponding notification when attempting design.



**Separated by modes** - Check box which condition determines whether the calculation for multimodal load cases is performed separately for each mode (not to be used for design; it is used for informational purposes only to examine the impact for each mode of modal analysis separately).

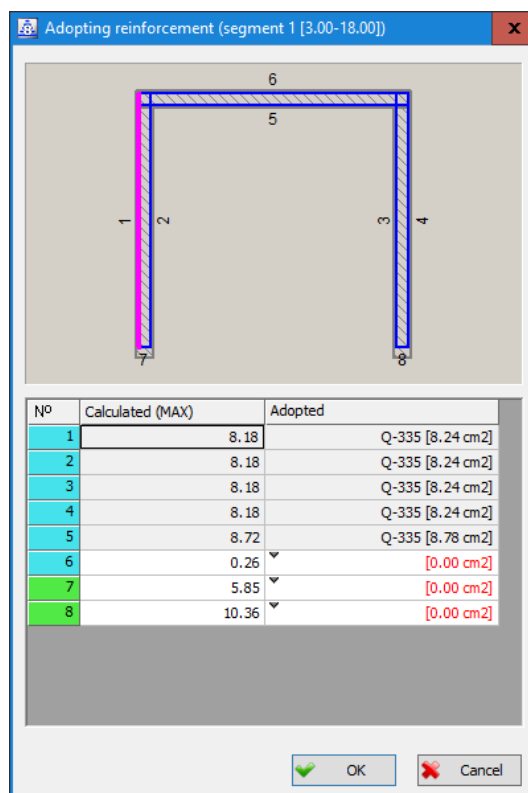
**Current load case** - When this switch is set to ON, design of proxy entity is performed for the current load case.

**Complete Scheme** - When this switch is turned on, design of proxy entity is performed for the complete load scheme.

**Take into account concrete shear resistance** - Check box which condition determines whether the shear strength of the concrete is taken into account when design of proxy entity is performed, or whether the shear resistance calculation will be performed according to the beam procedure (check box on) or the wall procedure. (check box off).

**Calculate** - Selecting this button starts the design of proxy entity according to the data specified in the dialog.

**Adopting Reinforcement** - This button becomes available only when the design of proxy entity was performed. The selection opens a dialog where the reinforcement for the current proxy entity segment can be adopted:



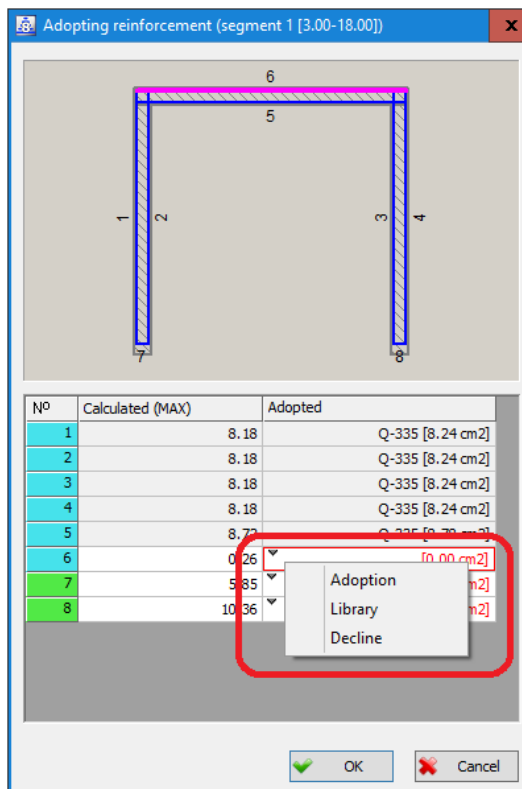
The layout of the reinforcement adoption dialog in the current segment of the proxy entity

The upper part of the dialog shows an image with all reinforcement lines marked with ordinal numbers, while the selected line is marked with a special color. At the bottom there is a table in which the reinforcement is adopted. Note that the "known reinforcement" specified in the dialog to define the reinforcement method cannot be modified in this dialog. Table columns have the following meaning:

**Nº** – Column in which the ordinal numbers of the reinforcement lines are printed.

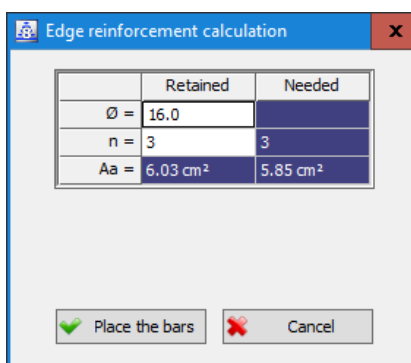
**Calculated (MAX)** - Column in which the required reinforcement for a given cross section is printed.

**Adopted** - The column in which the reinforcement is adopted and the previously adopted reinforcement is displayed. If the surface of the adopted reinforcement is smaller than the surface required, it will be printed in red. Right-clicking on cells in this column opens the drop-down menu with options for adopting reinforcement:



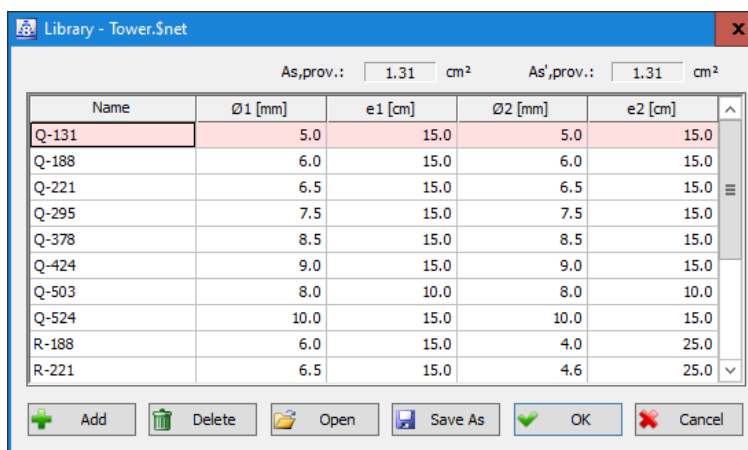
Drop down menu for reinforcement adoption

**Adoption** – Selecting this option opens a new dialogue where you can adopt the reinforcement:



This dialogue works in exactly the same way as the dialogue that opens in the procedure for the adoption of reinforcement in beams, when the command "Calculation of bars" is activated. After exiting the dialog, the selected reinforcement is displayed in the "Adopted" column.

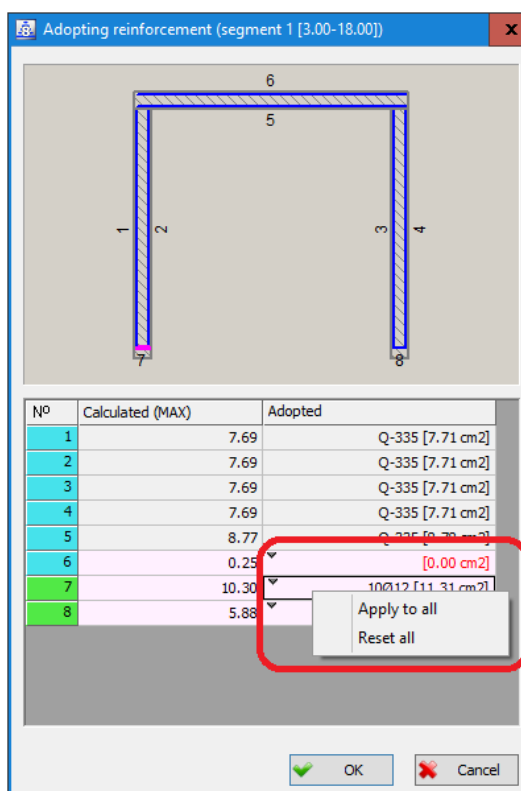
**Library** - This option is only available for reinforcement lines belonging to the slabs and along which longitudinal reinforcement is installed. It can be used to provide a reinforcement mesh along a given cross-section line. Its choice opens a dialogue of the following appearance:



This dialogue is the same as the dialogue that opens when the "Library" command field is activated in the slab reinforcement adoption procedure. After leaving the dialogue, the selected reinforcement mesh is displayed in the "Adopted" column.

**Decline** - By selecting this option, previously adopted reinforcement can be undone.

The program allows multiple selection of cross-section lines in the table, as well as the mass assignment and cancellation of the adopted reinforcement to all selected cross-section lines.



Drop down menu for reinforcement adoption

**Apply to all** - It is possible for the user to make the installation of the reinforcement in all selected cross-section lines in a quick and easy way. It is necessary to adopt the reinforcement one line, and then from the drop-down menu, which opens by right-clicking on the corresponding cell in the table, select the command "**Apply to all**". In this way, reinforcement will be adopted on all selected intersection lines.

**Reset all** - Selecting a given command from the drop-down menu will invalidate the previously adopted reinforcement for all selected cross-section lines.

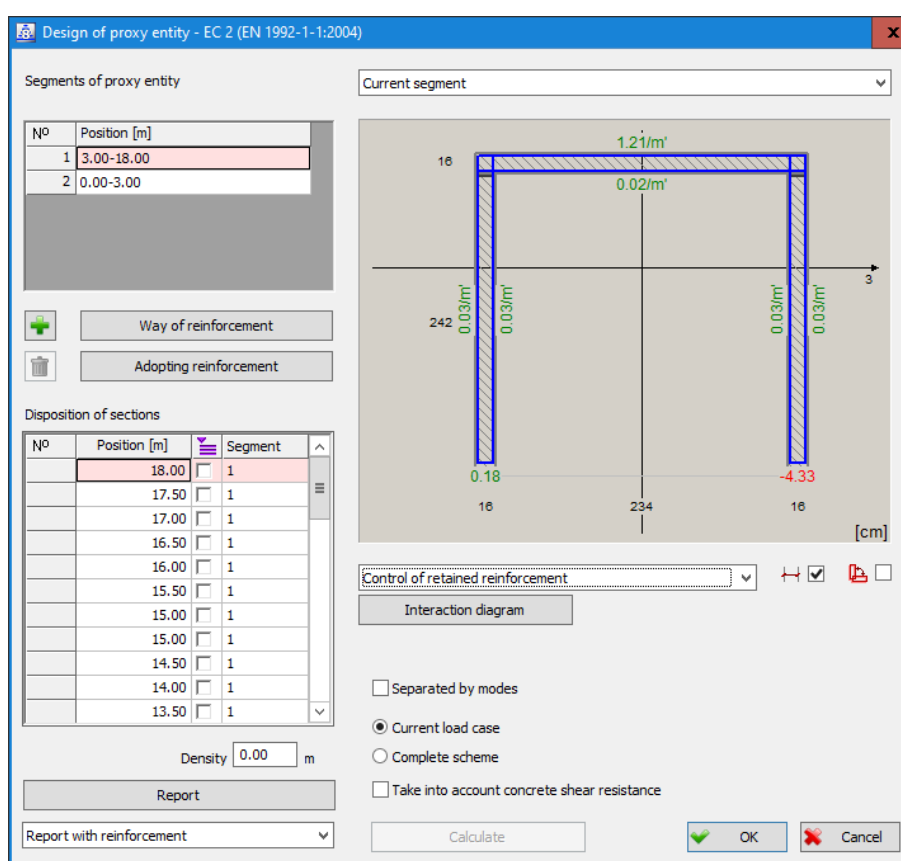
When the design of proxy entity is performed and the reinforcement is eventually adopted, the results in the command in the main dialog are displayed next to each armature line. What exactly will be printed is selected from the closed list below the image itself. The following options are offered:

**Calculated reinforcement**

**Adopted reinforcement**

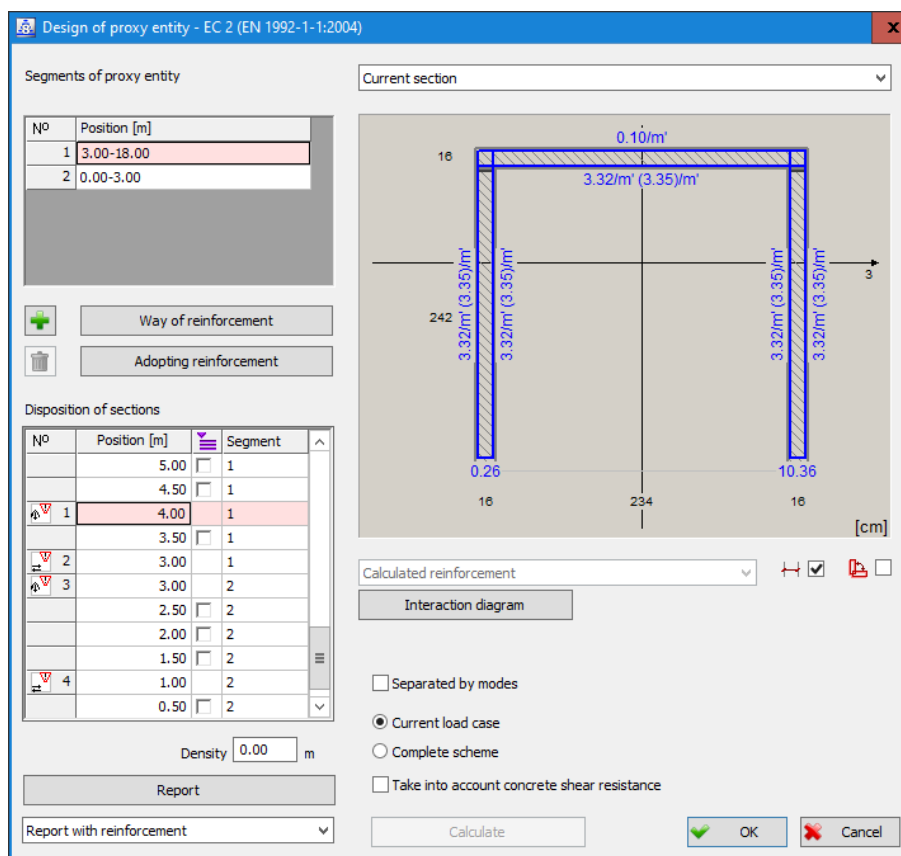
**Area of adopted reinforcement**

**Control of retained reinforcement** - The difference between the area of the adopted reinforcement and the calculated reinforcement for a given cross-section line. If the area of the adopted reinforcement is smaller than the area required, the difference will be printed in red. If a sufficient amount of reinforcement is adopted, the difference will be green.




An option is selected to display the image in the dialogue "Control of retained reinforcement"

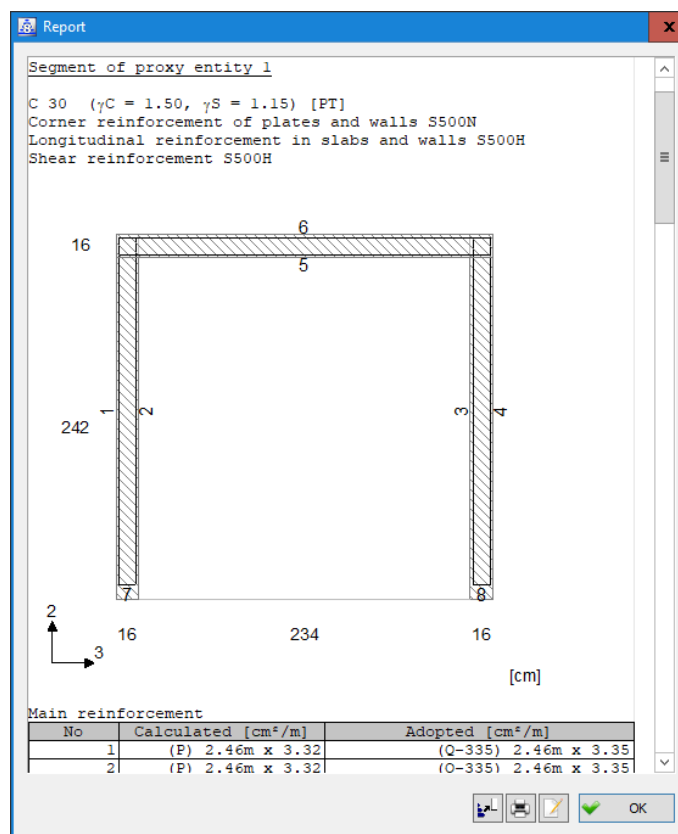
As the reinforcement is adopted for each proxy entity segment individually, based on the relevant cross sections, these options will only be available when the "Current Segment" option is selected from the list above the figure. If the option "Current section" is selected, the area next to each line will show the calculated reinforcement required for the currently selected cross section, and after that value the area of the known reinforcement will be printed, if given in the dialog box before dimensioning "Way of reinforcement".



Displays the required reinforcement of the selected section when it is above the image from the list "Current section" selected

**Interaction diagram** - Activating the command field "Interaction Diagram" opens a dialogue for generating the Interaction Diagram of the proxy entity in the section that is set for the current one in the "Disposition of sections" list. Working with this dialogue is exactly the same as with the command for generating beam interaction diagrams (Section 9.4.10).

**Report** - Selecting this button generates a report of the completed design of proxy entity, in the relevant sections and sections selected in the column  of the section "Disposition of sections", for each segment of the proxy entity.



Design of proxy entity Report Generated

**Report type** - From the list below the report generation button, select its content:

**Report with reinforcement** - The report contains complete concrete design results and adopted reinforcement in all sections for which it is generated.

**Report with influences** - The report contains only influence tables in all sections for which it is generated.

**Report with reinforcement and influences** - A report containing both.

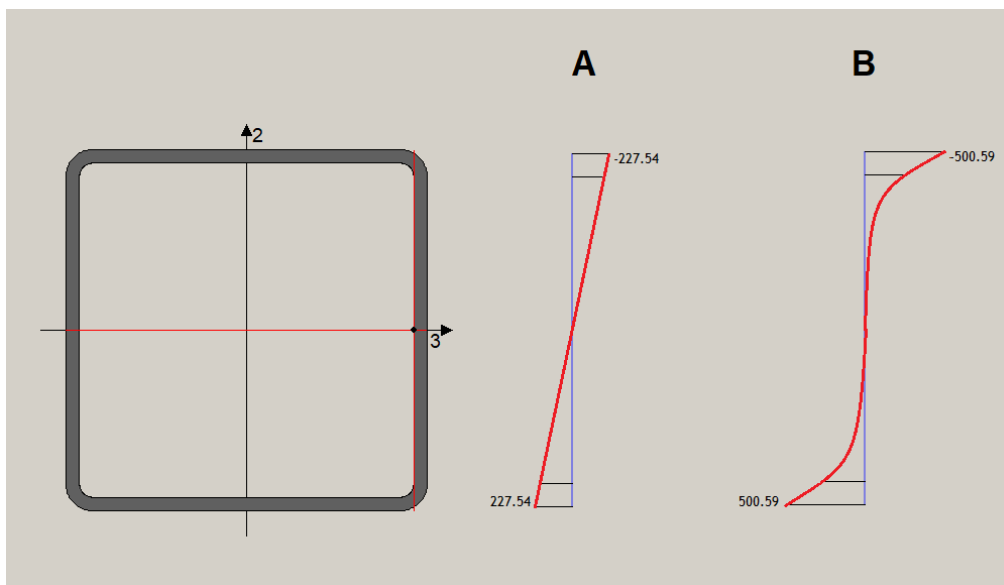
#### IMPORTANT NOTICE:

One of the assumptions on which the sizing of the proxy entity (reductor) is based is that the set of elements that make up the reductor acts as a line girder. The term "line girder" is considered to be a rigid body with one dominant dimension, such that during its deformation due to the action of external forces, there is no out of plane deformation of the cross section.

In order for the cross-sectional bending effect to be as small as possible, the cross-section itself should be as compact as possible.

The figure below shows the stress diagrams in one cross section exposed to pure bending. Diagram "A" of the stress distribution corresponds to the ideal state, without out of plane bending the cross section. Diagram "B" corresponds to a real state of stress distribution of non-compact cross section.

In both cases the bending moment is the same. The areas of the diagrams are the same in both cases, but the distribution and intensity of the stresses in the elements (rib and flange) are different.



These are the stages of reducer dimensioning:

- 1) from the elements that make up the proxy entity, local influences are taken and reduced (translated and summed) in relation to the axis of the reducer. Unique influences are obtained in the reducer.
- 2) a concrete cross-section is formed, which consists of all cross-sections of the reducer elements and is dimensioned on the effect of a unique influence in the reducer. Dimensioning is performed according to the idealized distribution of deformation, for a cross section that is not curved.

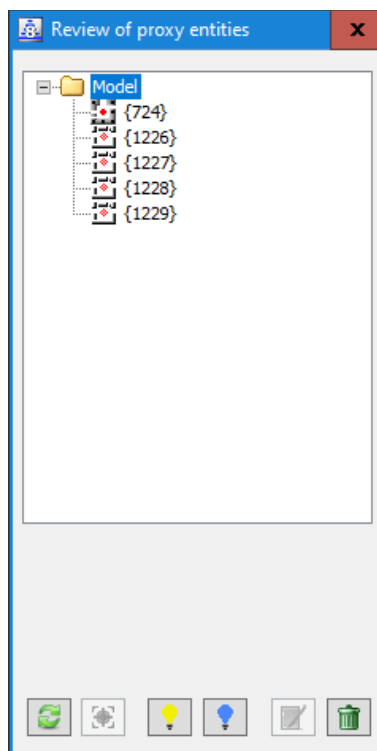
Dimensioning of seismic walls:

The wall is dimensioned according to its local influence, according to the real distribution.

It is clear that in this second case, the seismic wall (flange of this section) is dimensioned for far greater forces. The more disjointed and non-compact the cross section, the more significant the stress distribution within the cross section will deviate from the linear one. Therefore, there will be a deviation in the concrete dimensioning results of the proxy entity compared to the reinforcement calculation performed by command "Seismic walls".

## 9.12 Review of proxy entities

Selecting the “**Review of proxy entities**” command, which in the module for processing the results of calculations in the menu “Design ► Concrete”, opens a dialog of the following appearance:



The dialog shows a list of all proxy entities in the model. Since the list shows proxy entities for dimensioning as well as proxy entities that are not dimensioned, in the dialog, different symbols indicate proxy entities:



A symbol indicating that the proxy entity is designed.



Symbol indicating that the proxy entity is not designed.



Adding, deleting and design of proxy entity may cause the list of proxy entity in the dialog to no longer match the model state. Clicking this button refreshes the list, that is, aligns it with the current state of the proxy entity in the model.



Since the proxy entity cannot belong to any assembly, the given button is inactive in the proxy entity inspection dialog.



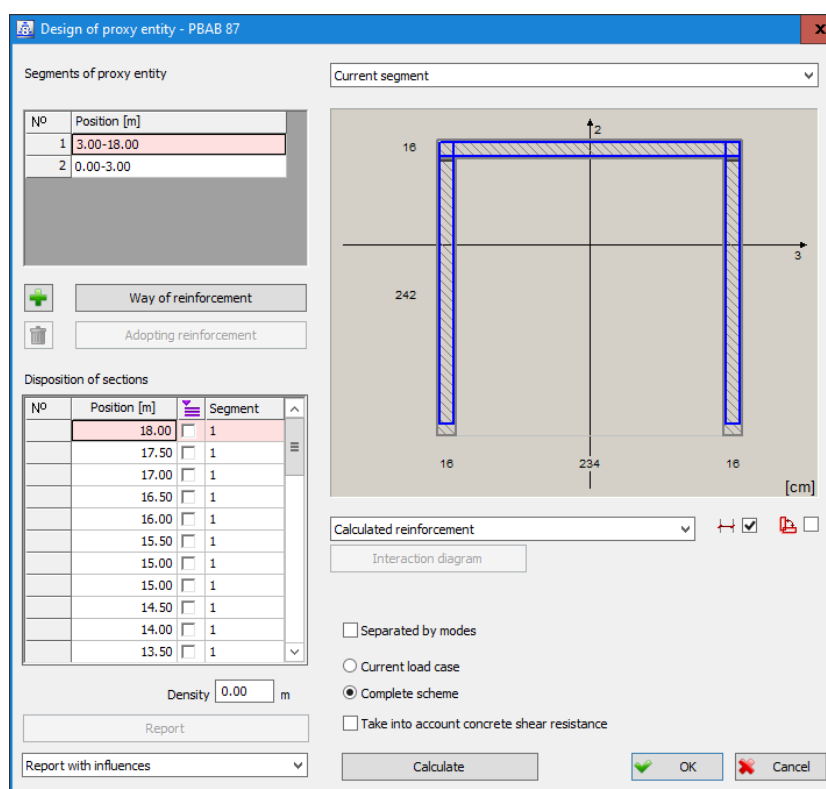
Clicking on this button turns on the visibility of the selected proxy entities, if they were previously turned off. If the “**Model**” branch is selected in the trunk, the visibility of all proxy entities in the model will be turned on. The visibility of the proxy entity can also be switched on by selecting the option “**Visible**” from the drop-down menu, which is opened by right-clicking on the name of the proxy entity, or through the branch “**Model**”.



Clicking on this button excludes the visibility of the selected proxy entities. If the "Model" branch is selected in the trunk, the visibility of all proxy entities in the model will be disabled. The visibility of the proxy entities can also be switched off by selecting the option "Not visible" from the drop-down menu, which is opened by right-clicking on the name of the Proxy entity, or through the branch "Model".



A button used to design of proxy entity currently selected in the list. Clicking on this button opens the following layout dialog:



Design of proxy entity dialogue

This dialog is the same as the dialog that opens when the proxy entity is designed, so we will not explain it again (see Chapter 9.11).



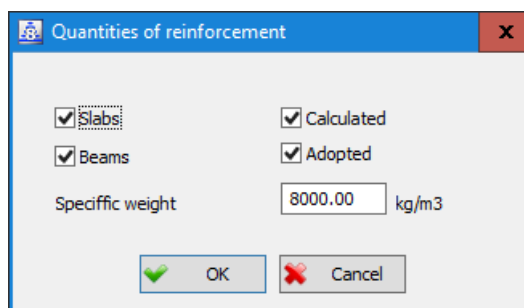
Button whose actuation removes the currently selected proxy entity from the drawing. The command to delete the proxy entity is also in the menu that opens by right-clicking on the name of the proxy entity in the list. Right-clicking on the name "Model" opens a menu with the option "Delete", which is used to delete all proxy entities from the model.

## 9.13 Quantities of reinforcement

Using the command "Quantities of reinforcement", a report of an orientation pre-measurement can be generated of the quantities of reinforcement required and adopted in the selected slabs and beams of the model. The command is in the "Design ► Concrete" drop-down menu, and its startup enters the standard selection procedure:

<0 sel.> Quantities of reinforcement - Select (All / Window / Polygon / Section / eXtras / selection Groups / Recent selection / Deselect / <End> / Layout):

Selecting the "**Layout**" sub-option from the command line opens a dialog where all the parameters necessary for generating the quantities of reinforcement are set:

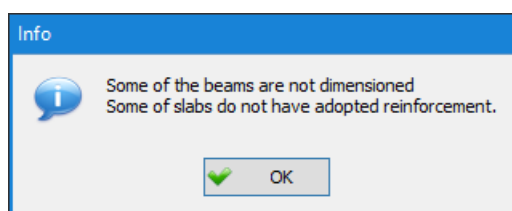


The check boxes "**Slabs**" and "**Beams**" determine whether the quantities of reinforcement is generated only for slabs, only for beams or for both.

The check boxes "**Calculated**" and "**Adopted**" select whether the report is generated only for the required, only for the adopted or for both types of reinforcement.

**Specific weight** - edit field specifying the specific weight of the adopted reinforcement.

After selection, the program checks that all beams and slabs have been designed and that all reinforcement has been adopted, and if no appropriate notification is issued. For example, if all selected beams are not designed and no reinforcement has been adopted in all selected slabs, the following message will be displayed:






These messages serve only as a notification of a possible failure that needs to be corrected and whether or not they are displayed, the program will generate a quantities of reinforcement report for those selected beams and slabs that meet all the necessary conditions and display it in a new dialog:

Slabs - quantities of reinforcement					
Material	$\gamma$ [kg/m <sup>3</sup> ]	Calculated		Adopted	
		Aa [kg]	Aa,br [kg]	Aa [kg]	Aa,br [kg]
S500H	8000.0	15710.2		18855.0	

Beams - quantities of reinforcement					
Material	$\gamma$ [kg/m <sup>3</sup> ]	Calculated		Adopted	
		Aa [kg]	Aa,br [kg]	Aa [kg]	Aa,br [kg]
S500H	8000.0	3094.9	655.5	4632.2	2193.1

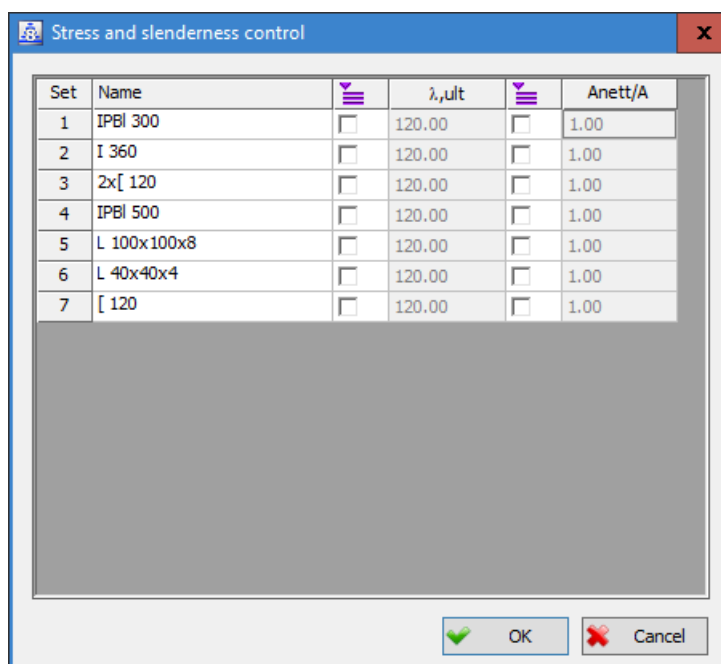
The layout of the dialog with the quantities of reinforcement report

Using the command fields located at the bottom right of the dialog, the created report of the quantities of reinforcement can be exported to the project documentation (  ), print (  ) or text file (  ).

# 10. Steel Section Design

## 10.3 Stress control

The program allows controlling the slenderness of the pressed bars as well as the stress control with the calculated weakening of the section from the external assets will be taken into account if the bar is subjected to tension. After activating the "**Stress control**" command, the following dialog opens:



**Set** Column with beam sets

**Name** Column with the names of profiles

**$\lambda_{ult}$**  Column for setting the limiting slenderness of the bars. If it is necessary that all beam sets have the same margin slenderness, it is enough to assign it to a single set, and then select the "**Apply to all**" command from the drop-down menu, which opens with the right mouse click over the input value. In this way, all the sets will be assigned the same value for the margin slenderness. To the left of this column there is a column with checkboxes, whose state determines which set of beams will be made to the slenderness analysis. Right-click on this column opens the drop-down menu with the "**All on**" and "**All off**" options, which you have the option to turn on or off check boxes for all sets.

**Anett/A** Column for setting the coefficient of the sectional surface area. If it is necessary that all beam sets have the same coefficient, it is enough to assign it to a single set, and then select the "**Apply to all**" command from the drop-down menu, which opens with the right mouse click over the entered value. In this way all the sets will be assigned the same value for the coefficient of the net sectional area. To the left of this column there is a column with check boxes, whose status determines which set of beams will be done to control the weakened section tension. Right-clicking on this column opens the drop-down menu with the "**All on**" and "**All off**" options, which you have the option to turn on or off check boxes for all sets.

By going to the "OK" button from this dialog, the program will perform slenderness control, if requested by the user, as well as stress control. All the bars in the model will be painted either green or red, while the cursor will get the shape of the select box, and the following message will appear on the command line:

<0 sel.> Stress control - Selecting (Report/ <End>):

The user is now expected to select the beam for which he wants to get a detailed overview of the conducted stress control.

After selecting the desired beam, the program will open a dialog with a detailed overview of the implemented stress control and slenderness control:

Nº	σmax [kN/cm²]	τmax [kN/cm²]	σumax [kN/cm²]
9	0.587	0.025	0.589
10	0.996	0.046	0.996
11	0.717	0.026	0.718
12	0.795	0.031	0.796
13	0.838	0.030	0.839
14	1.370	0.031	1.371
15	1.499	0.032	1.500
16	0.717	0.026	0.718

Exceeded slenderness  
(λy=110.14) (λz=215.82>120.00)

Layout of the dialog for displaying stress control and slenderness control

In the lower part of the dialog, the results of the checked slenderness of the selected beam are shown.

For the performed stress control and the slenderness control of the selected beam, a report can also be generated. For this purpose, the command field "Report" is foreseen, which activation opens a dialog of the following layout:

Stress control - EUROCODE 3 (EN 1993-1-1:2005)				
Desc.	LC	$\sigma$ [kN/cm <sup>2</sup> ]	$\tau$ [kN/cm <sup>2</sup> ]	$\sigma_u$ [kN/cm <sup>2</sup> ]
Set 5: L 100x100x8				
(169 - 282)	9	0.587	0.025	0.589
	10	0.996	<b>0.046</b>	0.996
	11	0.717	0.026	0.718
	12	0.795	0.031	0.796
	13	0.838	0.030	0.839
	14	1.370	0.031	1.371
	15	<b>1.499</b>	0.032	<b>1.500</b>
	16	0.717	0.026	0.718
Exceeded slenderness ( $\lambda_y=110.14$ ) ( $\lambda_z=215.82>120.00$ )				

Dialog with the generated report of the stress control and slenderness control

The last row shows the control of the slenderness of the selected beam.

We will note that the program will only control slenderness if the user has requested it by including the appropriate check boxes in the "Stress and slenderness control" dialog.

## 10.4 Input Data

### EUROCODE

#### Ignore lateral torsional buckling effects

Since some sections, e.g. tubular sections and square boxes, insensitive to lateral-torsional buckling, and for them there is no need to do the check, the program allows the omission of this check. Status of the check box "**Ignore lateral torsional buckling effects**" determines whether or not the load bearing test will be carried out on lateral-torsional buckling. Note that this check box appears only if the EUROCODE 3 (EN 1993-1-1: 2005) is set in the regulatory dialog (see chapter 10.1).

**Side budget calculation procedure lateral torso bending** - Closed list from which one of the offered procedures for calculation of lateral-torsional bending is made:

- **Global case (6.3.2.2)**
- **Rolled and Welded And Girders (6.3.2.3)**

#### Load shape dependant parameters

**Mcr** When the "Arbitrary" check box is set to on, the "**Mcr**" check box can be turned on and set a critical torque value for lateral-torsional twisting in the appropriate edit field. By turning on the check box "Mcr", the edit fields for setting the coefficients "C1", "C2", "C3", and " $\beta_{M.LT}$ " become inactive.

Edit field for setting the critical torque for lateral torsional twisting

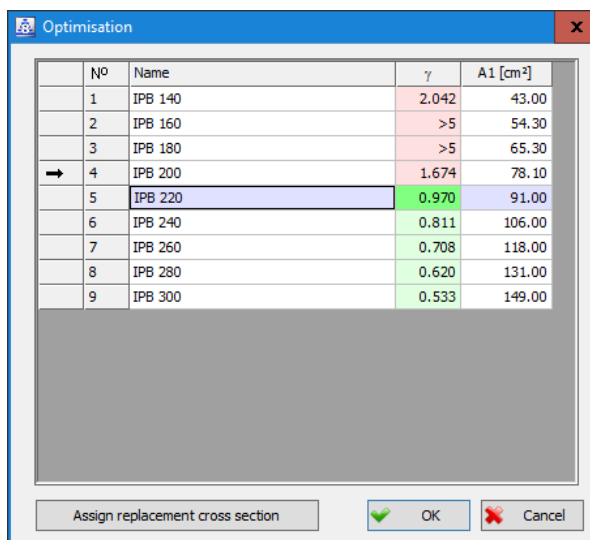
## 10.6 Viewing results

After running the command and selecting an arbitrary stick from the drawing, a dialog of the following layout opens:

"Optimisation" button in the dialog for a detailed view of the designing results

## Optimisation

Selecting this button opens a dialog where, based on the results of cross-section optimization, it is possible to place the currently selected rod instead of the current beam set by replacing the cross-section.



Dialogue for selecting a replacement section  
selected bar

**N°** Column with ordinal numbers of replacement cross sections of the selected bar, for which stability control was performed in the process of cross-section optimization. In front of the original section number is an arrow (→).

**Name** Column with descriptions of sets of replacement cross sections of the selected bar.

**γ** Column with utilization factors of replacement sections for which stability control was performed in the process of cross-section optimization. Depending on whether or not the replacement sections satisfy or do not satisfy, the cells in the given column will be colored differently:

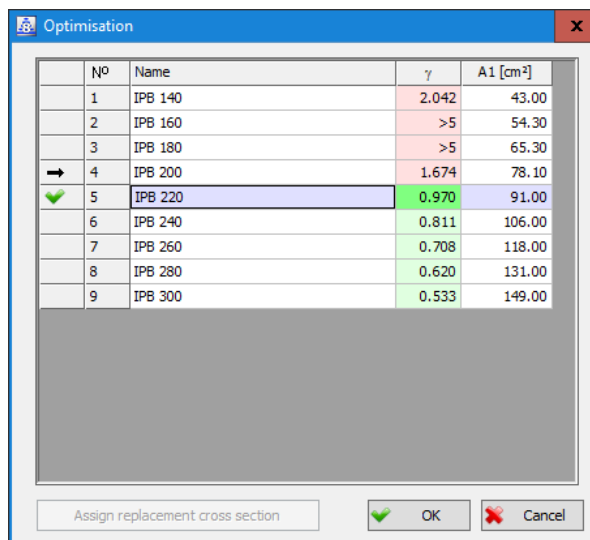
Cells in which the utilization factor is greater than the upper utilization limit are indicated in red.

Cells with a utilization factor less than the upper utilization limit are indicated in green, with the cell with the most optimal stability control results clearly indicated.

**A1 [cm2]** Column showing surfaces of alternating cross sections.

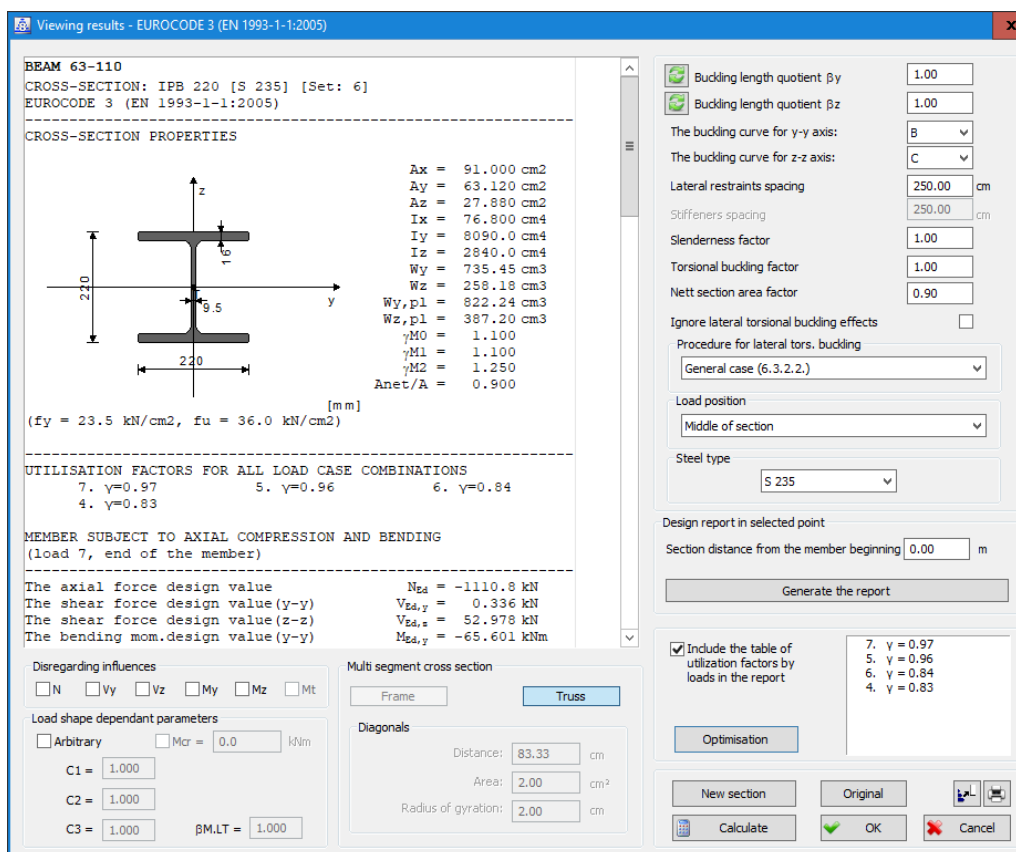
### Assign replacement cross section

By selecting the desired section from the list and activating this button, the selected rod will be assigned a replacement section, and a symbol "✔" will appear in the column in front of its ordinal number.



The selected beam is assigned a replacement cross section

By activating the "OK" button, the dialogue will be closed, and the program will return to the dialog to display the results, which will now show the sizing results for the replacement section:




The dialog shows the results of designing the replacement cross section

## 10.9 Stability control - calculator

The layout of the dialog for controlling the stability of an individual bar

The closed list located in the lower left corner of the dialog is intended for selecting some of the previously saved configurations. Each regulation has its own list of configurations. Selecting the appropriate regulation, the list will show all the configurations saved with the assigned regulation.

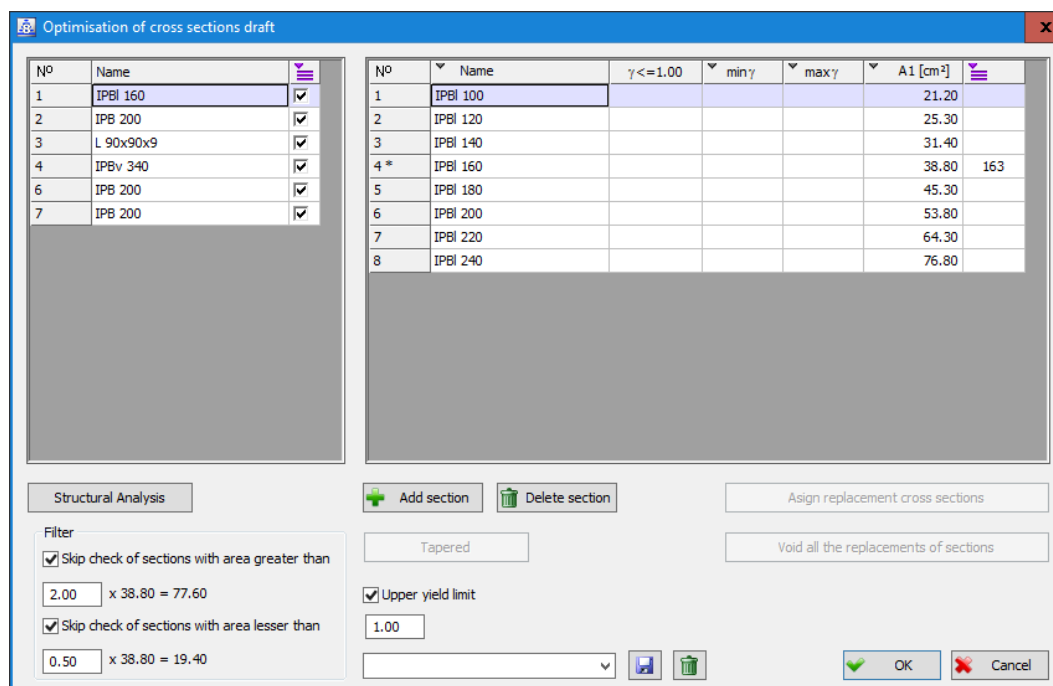
 **Save** Selecting this button, the current state of the parameters in the dialog is saved.

 **Delete** Selecting this button, the current configuration is deleted.

## 10.10 Optimisation of cross sections

Cross-section optimization in the Tower program enables checking the dimensioning of adopted cross-sections and provides the user with information on whether it can use another profile instead of one profile used in the construction. The optimization process is based on the sizing of all the bars of one cross-section with its smaller or larger profiles. As a result of cross-section optimization, the number and location of bars that do not satisfy the stability control are obtained, as well as the number and location of bars that are oversized and could be replaced by smaller bars, thus making changes that could lead to savings.

Running the **"Optimisation of cross sections"** command opens the following layout dialog:



The layout of the cross-section optimization dialogue

The left part of the dialog shows a list of all the steel cross sections that are in the model and for which it is possible to optimize the cross sections:

**Nº** Column with ordinal numbers of steel cross-section sets included in the model.

**Name** Column with descriptions of sets of steel sections. Behind this column is a column with check boxes which condition determines for which set of beams the cross-sections will be optimized. Right-clicking on this column opens the drop-down menu with the options "All on" and "All off", the choice of which you can enable or disable check boxes for all sets.

Optimizing the selection of cross sections, that is, finding a cross section that can be satisfied, can be done by checking all profiles from the appropriate family or by checking only a selected group of profiles. In the "Filter" section, the user is allowed to define which profiles will be checked during the optimization process to see if they can replace the adopted cross-section in the model:

#### Skip check of sections with area greater than

Included check box means that sections whose surface exceeds the corresponding value, which the user defines as a function of the area of the adopted cross-section in the model, will not be checked. The edit field specifies a factor that multiplies the cross-sectional area ("A1") used in the model, which means that cross-sectional areas larger than the obtained value will not be checked when optimizing the cross-sections.

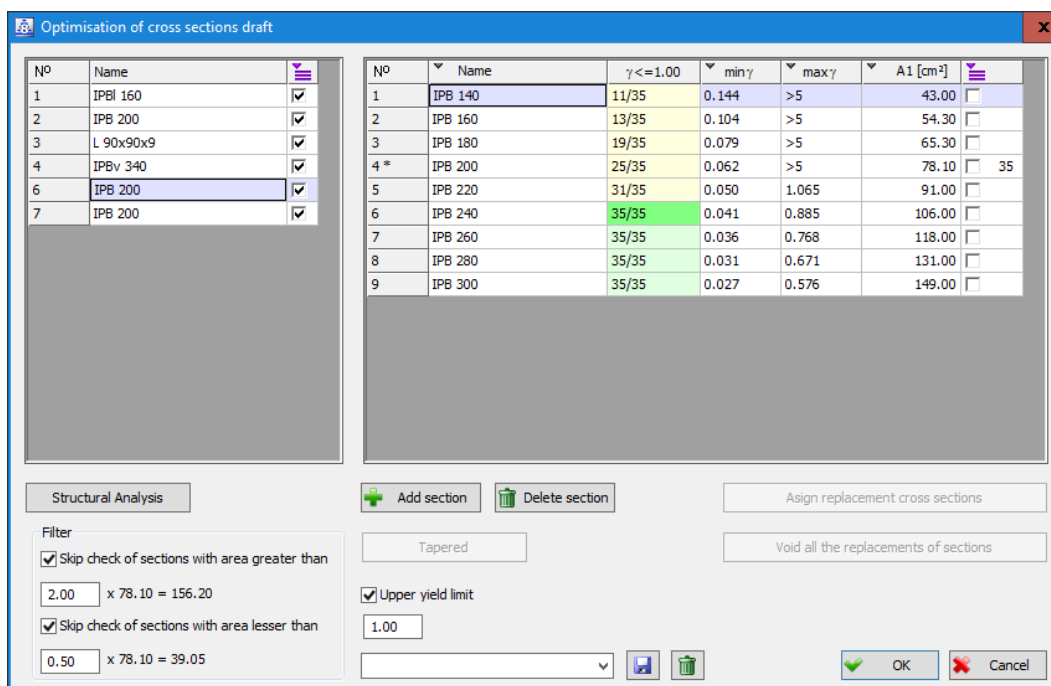
#### Skip check of sections with area lesser than

Included check box means that sections whose area was lesser then corresponding value, which the user defines as a function of the area of the adopted cross-section in the model, will not be checked. The edit field specifies a factor that multiplies the cross-sectional area ("A1") used in the model, which means that cross-sectional areas lesser than the obtained value will not be checked when optimizing the cross-sections.

## Structural Analysis

A button which activation starts the process of optimizing the selected sets of bars, that is, designing the steel sections defined by the user in the "Filter" dialog section.

In the right part of the dialogue, for each set of steel sections in the model, a list with all sections that are checked in the optimization process is displayed:



The right table shows the results of the steel section optimization

**N°** Column with ordinal numbers of steel sections checked for cross-section optimization. The symbol "\*" after the sequence number indicates that the given section is used in the model.

**Name** Column with descriptions of steel cross sections to be checked for cross section optimization.

**$\gamma \leq$**  The name of the column displays the limit value of the utilization coefficient that the user can set himself. It is necessary to first activate the check box "**Upper yield limit**" and then enter the desired value in the appropriate edit field. A check box off indicates that the value defined in the regulation dialog will be taken as the upper utilization limit (see section 10.1). After optimizing the cross sections, this column shows how many bars of the total number of bars of a given set have a utilization factor less than the upper utilization limit, or how many bars are over-dimensioned and can be replaced by smaller profiles, and how many bars have an utilization factor greater than the upper limit utilization. Depending on the results of the cross-section optimization, the cells in this column are marked with special colors:

Cells in which no bar of a given set has a yield factor less than the upper yield limit are marked in red.

Yellow indicates cells in which only some bars of the total number of bars of a given set have a utilization factor less than the upper utilization limit ("**n1/n**", where "**n1**" is the number of bars that meets the upper utilization limit, and "**n**" the total number of bars).

The cells in which all the bars of a given set have a yield factor less than the upper yield limit are marked in green, with the cell with the most optimal stability control results clearly indicated.


**min  $\gamma$**  Column showing the minimum values of utilization factors that the bars of the corresponding set have.



**max  $\gamma$**  Column showing the maximum values of utilization factors that the bars of a given set have.

**A1 [cm<sup>2</sup>]** Column showing the cross-sectional areas of the corresponding set.

Including check boxes in the last column and activating the "**Assign replacement cross sections**" button allows the user to replace the adopted cross section in the model with the desired cross section.

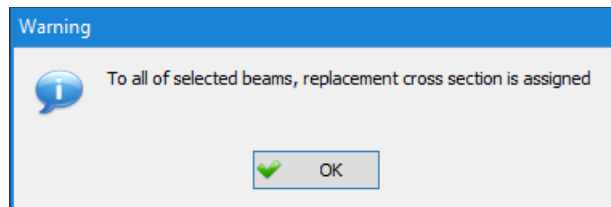
### Sorting Columns

Sorting can be done for all columns that have a symbol  in the name cell. Right-clicking on the name of the desired column opens the drop-down menu with options:

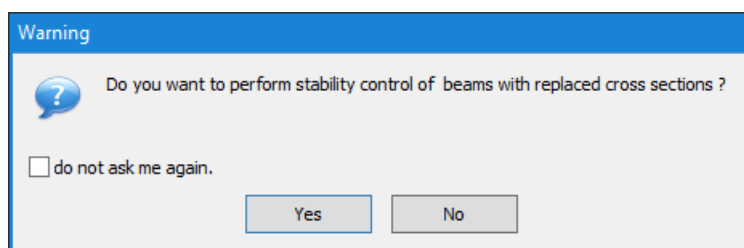
- **Sort (+)** -selecting this option sorts the sections shown in the table by the data from the selected column in ascending order. After sorting in a cell with a column name, a character will be printed below the sort symbol. 
- **Sort (-)** - selecting this option sorts the sections shown in the table by the data from the selected column in descending order. After sorting in a cell with a column name, a character will be printed below the sort symbol .

### Assign replacement cross sections

Activating this button will assign all the sticks in the current set to replace the sections with the check box included in the last column, with the program throwing a corresponding message:



In addition to the check boxes included, it will now print how many rods have been replaced with the selected sections. By selecting the "**OK**" button, a new alert appears:



By selecting the "**OK**" button, the program will control the stability of all bars that have been replaced with a cross-section.

### Void all the replacements of sections

By activating this button, all the bars of the current set that have been assigned to replace the cross sections will re-set the cross section adopted in the model.

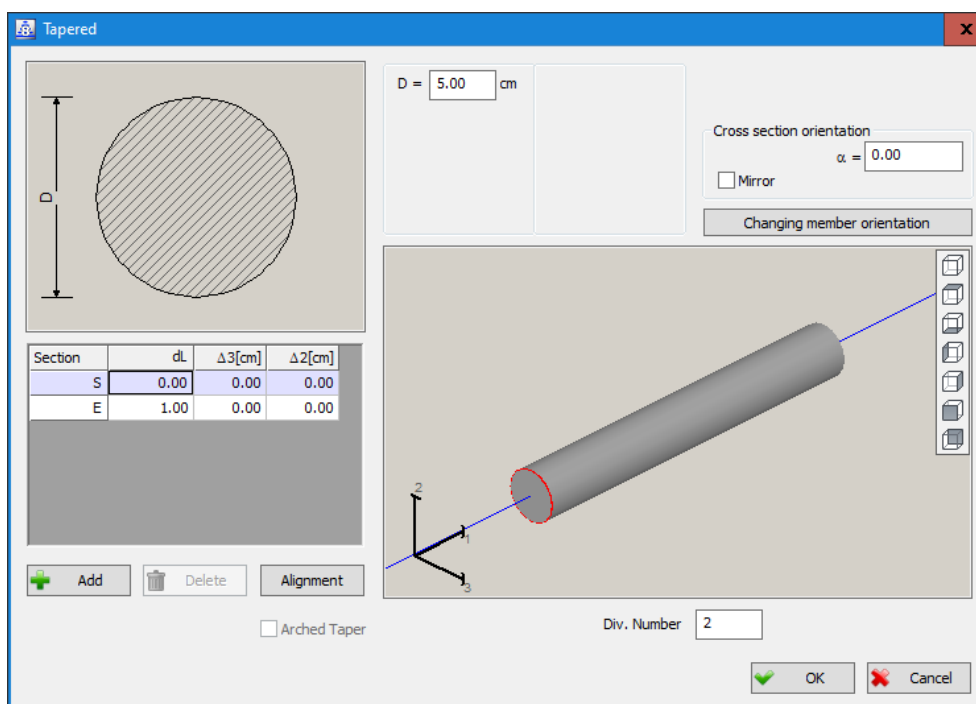
### Add section

The button to add a new section to the list of sections that are checked in the optimization process. The newly added section will be placed at the end of the list.

### Delete section

A button that can be used to remove arbitrarily selected sections from the list of sections to be checked in the optimization process.

**Tapered** Cross-sections can be optimized not only for sections defined by selection from the profile library, but also for sections defined by specifying their dimensions (rectangular, tubular, circular, T-section, I-section, ...). Since the optimization process is based on the designing of all the bars of one cross-section with its close smaller or larger cross-sections, if a cross-section is defined from the list defined by specifying its dimensions, the button "**Tapered**" becomes active. By activating the given button, the program enables modeling of a variable cross section beam, while by downloading a cross section from the beam, new cross sections will be inserted into the list for optimization:

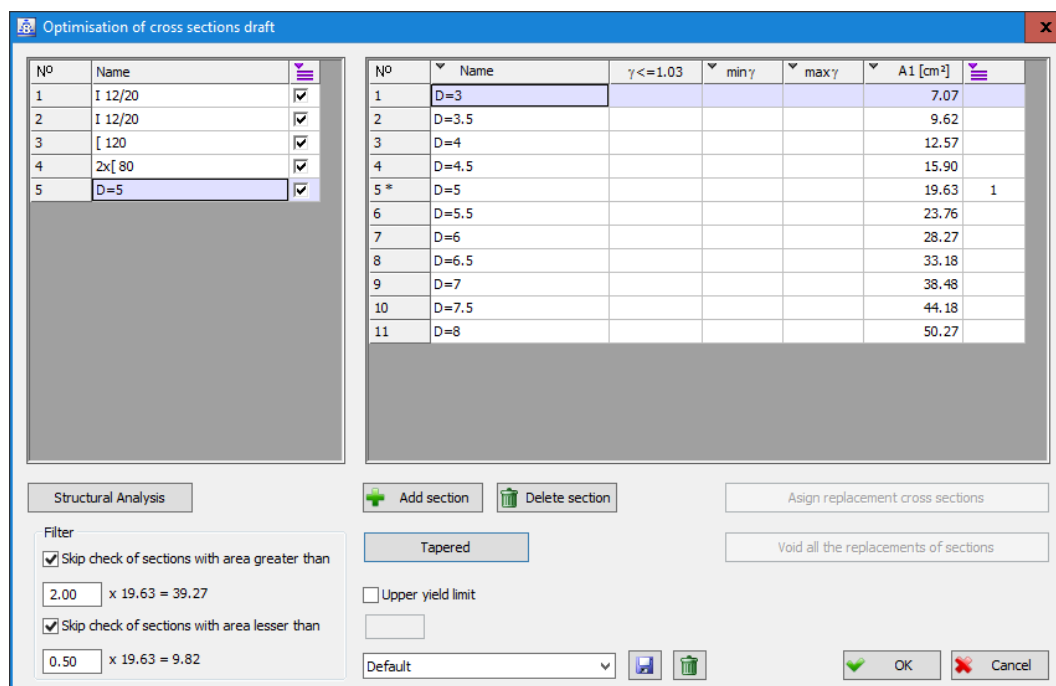


Layout dialoue for defining a tapered beam cross-section

The definition of variable cross-section beams has been explained earlier and will not be re-explained (see section 3.1.4).

Note that in this way the variable cross section bars are not dimensioned, but the variable cross section bar is used to define a series of cross sections with related geometry that will be considered in the optimization process.

**Div. Number** - After defining a variable section beam, in this edit field, the user specifies the number of sections that will automatically be placed in the list of sections after leaving the dialog for optimization.



Cross sections obtained using the "Tapered" option have been added to the list



Selecting this button saves the current state of the section with cross sections that are checked during the optimization process.



Selecting this button deletes the current configuration.

Any of the previously recorded configurations can be selected from the list at any time with the program and set to the current one, with the exception that for each profile family only the corresponding configuration can be selected.

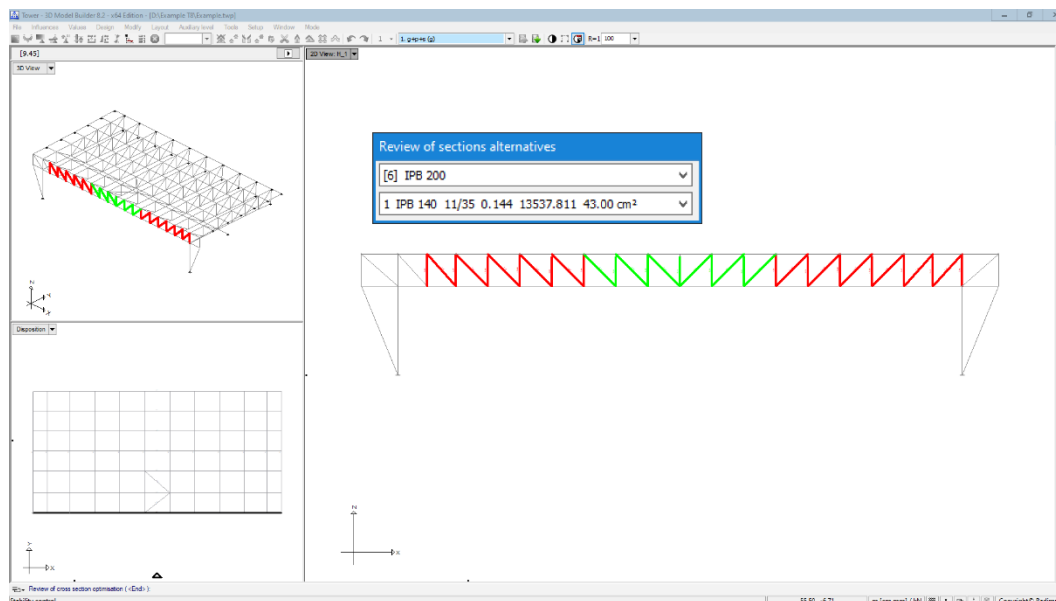
## 10.11 Review of cross section optimisation

By selecting this command, the program displays all the bars of the current set, for which cross-section optimization has been carried out, in green and red, respectively, while the line command will take the following form:

**Review of cross section optimisation (<End>):**

Green will paint all the bars that satisfy, and red all the rods that do not meet stability control.

At the time of invocation of the command, a dialogue will appear on the screen in which the user can assign replacement sections to the currently selected set of beams, thus reviewing the results:

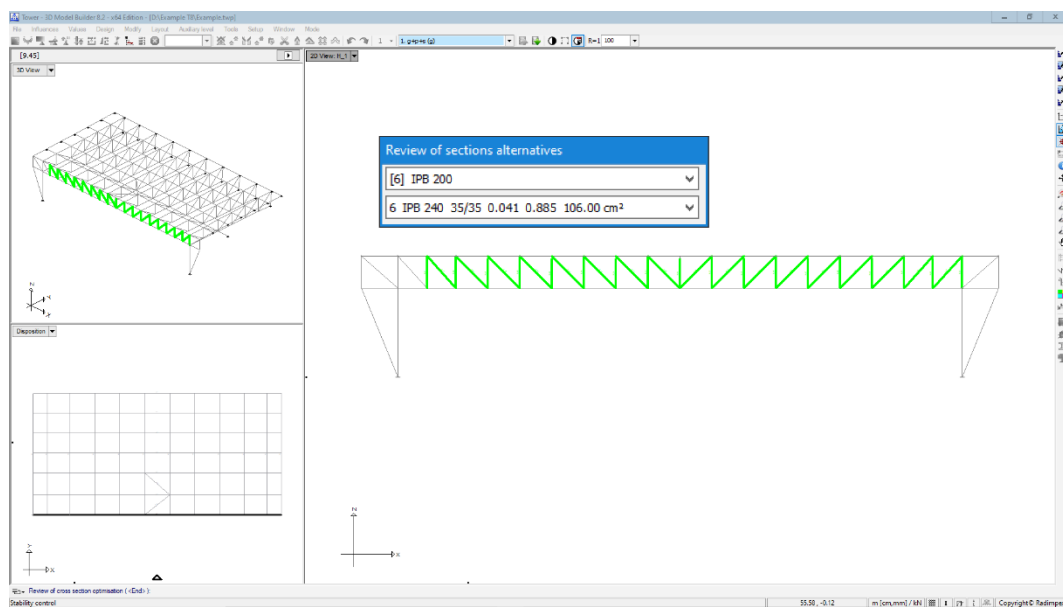


Section	Number of bars	Utilization coefficient	Area (cm <sup>2</sup> )
[6] IPB 200			
1 IPB 140	11/35	0.144	43.00
2 IPB 160	13/35	0.104	54.30
3 IPB 180	19/35	0.079	65.30
4 * IPB 200	25/35	0.062	78.10
5 IPB 220	31/35	0.050	91.00
6 IPB 240	35/35	0.041	106.00
7 IPB 260	35/35	0.036	118.00
8 IPB 280	35/35	0.031	131.00
9 IPB 300	35/35	0.027	149.00

The layout of the replacement cross sections

From the list above in the dialog, the user selects the current set of beams used in the model, while from the list below, selects the replacement section with which to replace the bars of the selected set. For easier selection of the replacement cross section, the list also shows the results of optimization, ie how many bars out of the total number of bars it meets, as well as the minimum and maximum utilization coefficient and cross-sectional area.

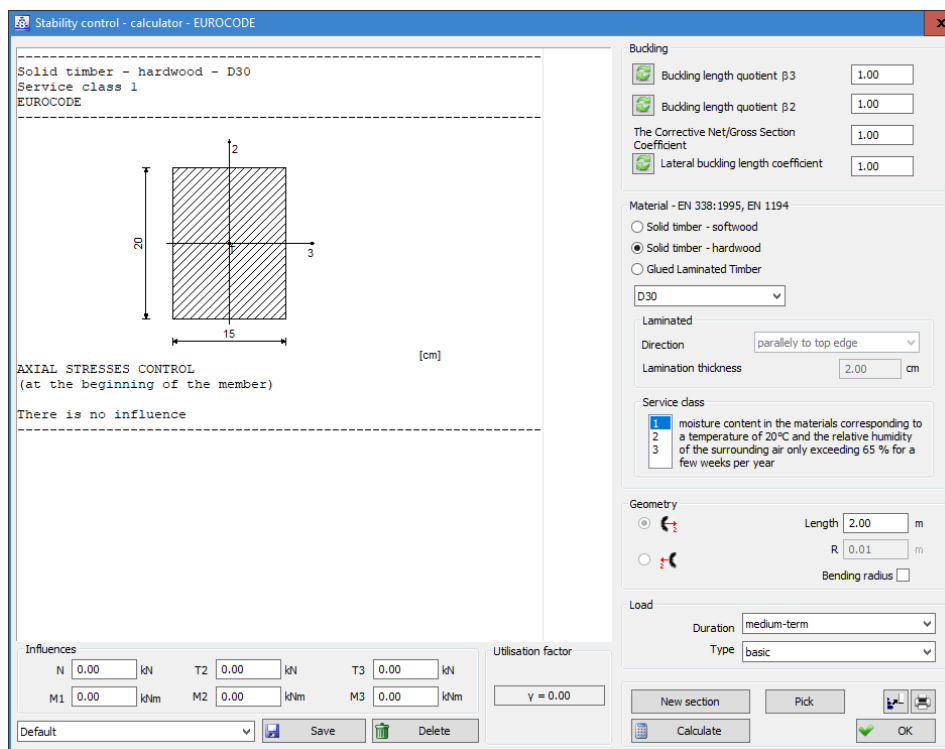
By setting the desired sections in the lists, all the sticks of the current set will be assigned to the replacing sections, whereby the program will stain all the sticks of the current set in green and red respectively:



After installing the replacement section, the program is all bars current set satisfying, painted green


# 11. Wood Constructions Dimensioning process


## 11.8 Stability control - calculator



The layout of the dialog for controlling the stability of an individual bar

The closed list located in the lower left corner of the dialog is intended for selecting some of the previously recorded configurations. Each regulation has its own list of configurations. By choosing the appropriate regulation, the list will show all the configurations saved with the assigned regulation.

 **Save** Selecting this button, the current state of the parameters in the dialog is saved.

 **Delete** Selecting this button, the current configuration is deleted.