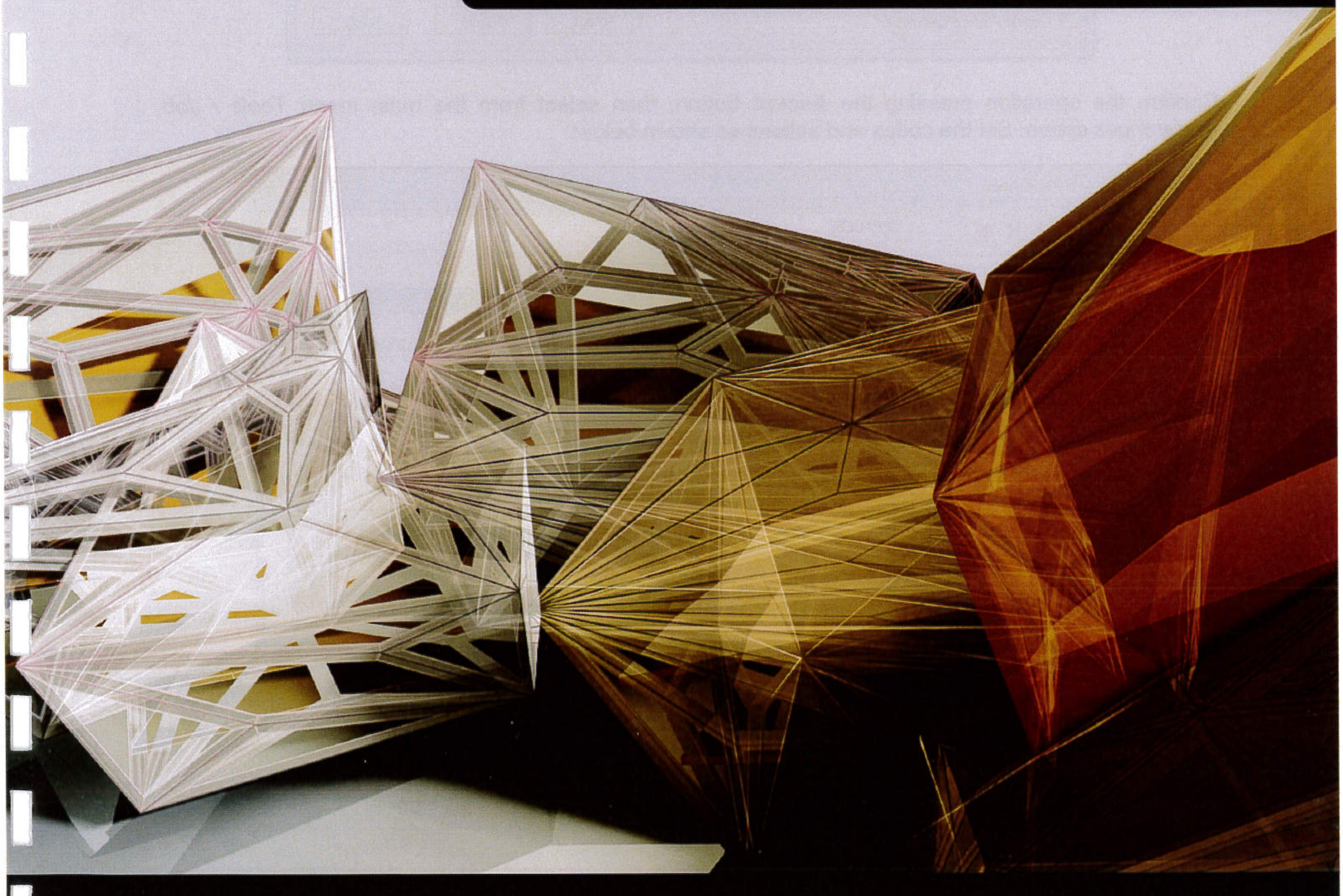
Robobat®

CONSTRUCTIVE SPIRIT

AUTODESK®
Authorized Training Center

AUTODESK® ROBOT™ STRUCTURAL ANALYSIS PROFESSIONAL

Training manual



AUTODESK.
Gold Partner

AUTODESK.
Authorized Training Center
Authorized Certification Center

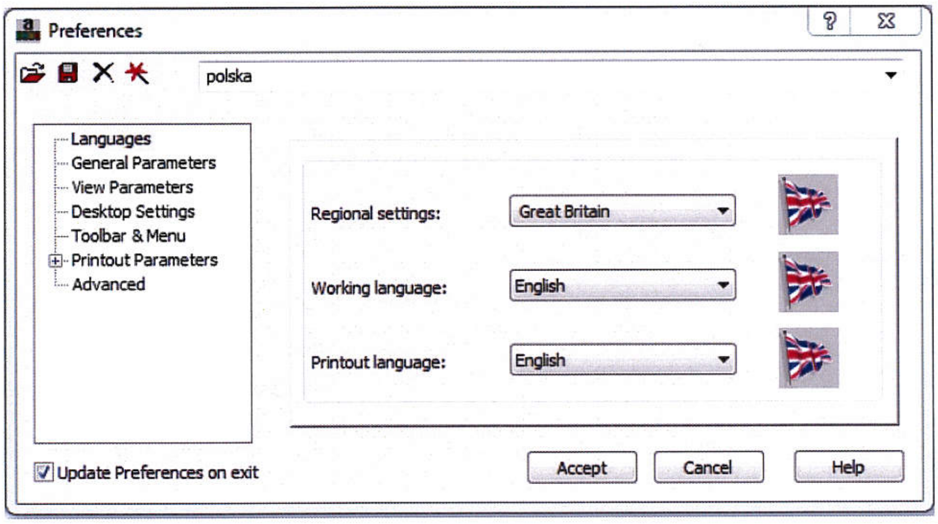
AUTODESK.
Consulting Specialized
Product Support Specialized

hp Gold Partner

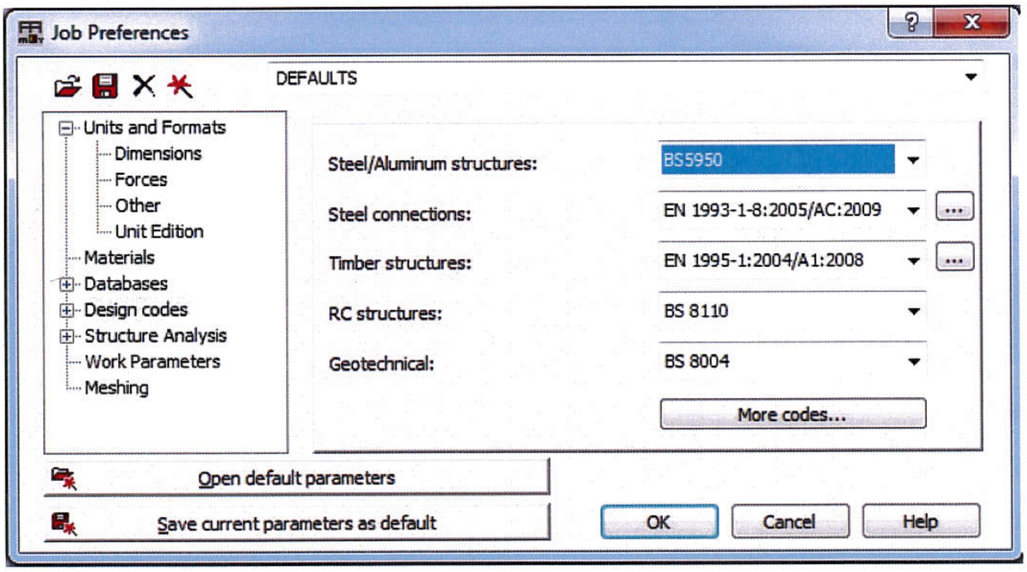
AUTOFDESK ROBOT STRUCTURAL ANALYSIS SYSTEM CONFIGURATION AND GENERAL INTERFACE OPTIONS

Setting working language and codes

Before commencing structure definition, one should set the working language and codes to be applied in the project. On purpose to do that select *Tools / Preferences* option from the main menu and apply the setting as shown in the picture below:



Confirm the operation pressing the **Accept** button, then select from the main menu *Tools / Job Preferences* option. Set the codes and actions as shown below:

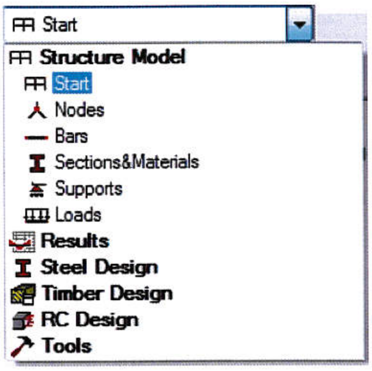


Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Edit options that are useful in the process of structure definition

Layout selection

It is necessary to select appropriate layouts in the process of structure definition. The layouts are accessible by clicking the list box in the top right corner of the main window, which opens the layout list shown in the figure below:



Selection

- By means of the **mouse** – if the selection with the mouse cursor is performed from the upper left corner to the lower right corner, only the bars (objects) that enter entirely within the defined rectangle will be selected; if the selection with the cursor is performed from the lower right corner to the upper left corner, all the bars (objects) that enter, however partially, into the window will be selected;

- By means of the *list boxes in the top menu* – it is possible to select elements (bars or nodes) by typing the relevant numbers or selecting all the elements.

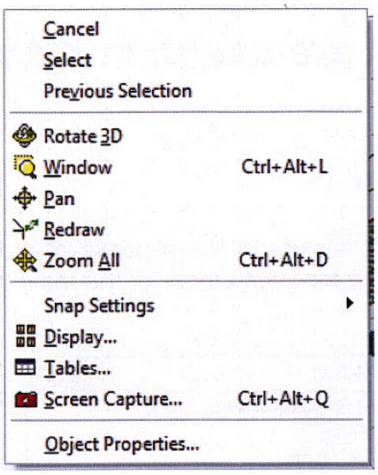
- By means of icons: *Bar selection* , *Node selection* (beside the list boxes) – this manner of selection allows one to filter the elements to be selected.

The selected elements may be edited in a new window. To do so, one should press the *Edit in New Window* icon - after selecting the required elements. There will appear a new window (viewer) with the selected element. It can be closed by means of the *Exit* icon located in the recently open window, or by clicking the *End of Edit* icon.

Context menu

While working in the graphical viewer, one may activate the context menu (shown below) by pressing the right-hand mouse button.

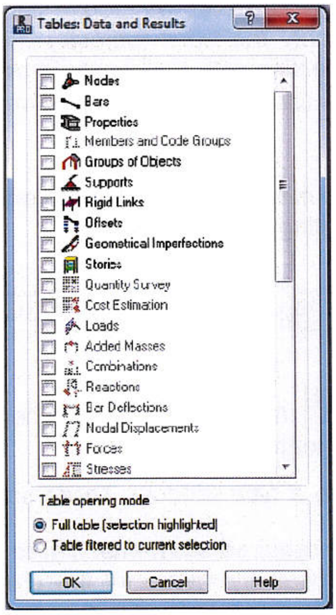
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



The menu allows one to perform many useful (and frequently used) operations while the program is carrying out the formerly issued commands.

Tables

Structure parameters may be modified by means of the relevant tables. The tables relevant to the current layout become visible, when one enters the layout. In order to be able to perform global edit operations, one should use the *View / Tables* option from the main menu. There will appear the ***Tables: Data and Results*** dialog box.

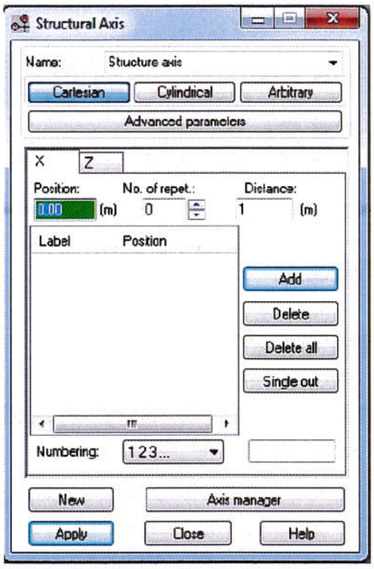


In this dialog box, one should indicate the required items and press the **OK** button. A table containing data will be generated for each of the indicated items. Once the *Edit* tab is activated in the bottom left corner of a given table, one may perform the operation of modifying structure parameters.

Structural axes

It is possible to modify the position of structural axes during the design process. Once the *Select* option is chosen (except when one is in the **START** layout), one should double-click the symbol (numbering) referring to the required axis, which results in the appearance of the below-presented dialog box used for structural axis modification.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

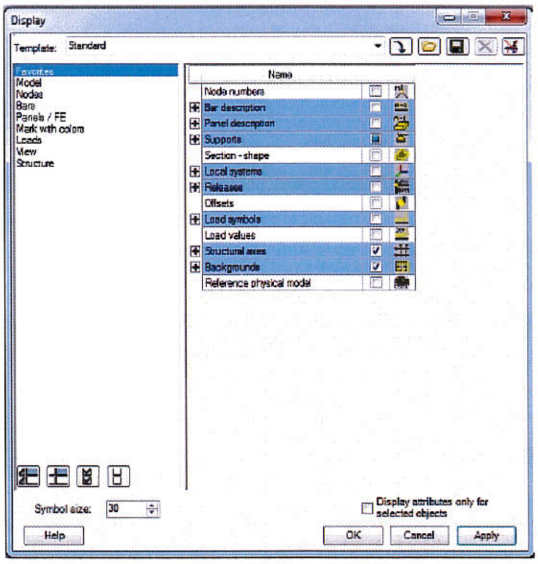


Snap settings

They become available once the  icon is pressed (the first one icon located in the bottom left corner of the screen), which results in the appearance of the ***Snap Settings*** dialog box. Test the operation of particular options.

Display of attributes

The option becomes accessible once the  icon (the second one icon in the bottom left corner of the screen) is pressed; as a result, there appears the ***Display*** dialog box shown below.



The available tabs allow one to get access to the data on structure parameters. This option is also available from the main menu by means of selecting the *View / Display* command.

Default options

Once the  icon (the third one icon in the bottom left corner of the screen) is pressed, the program restores the default attributes presented on screen.

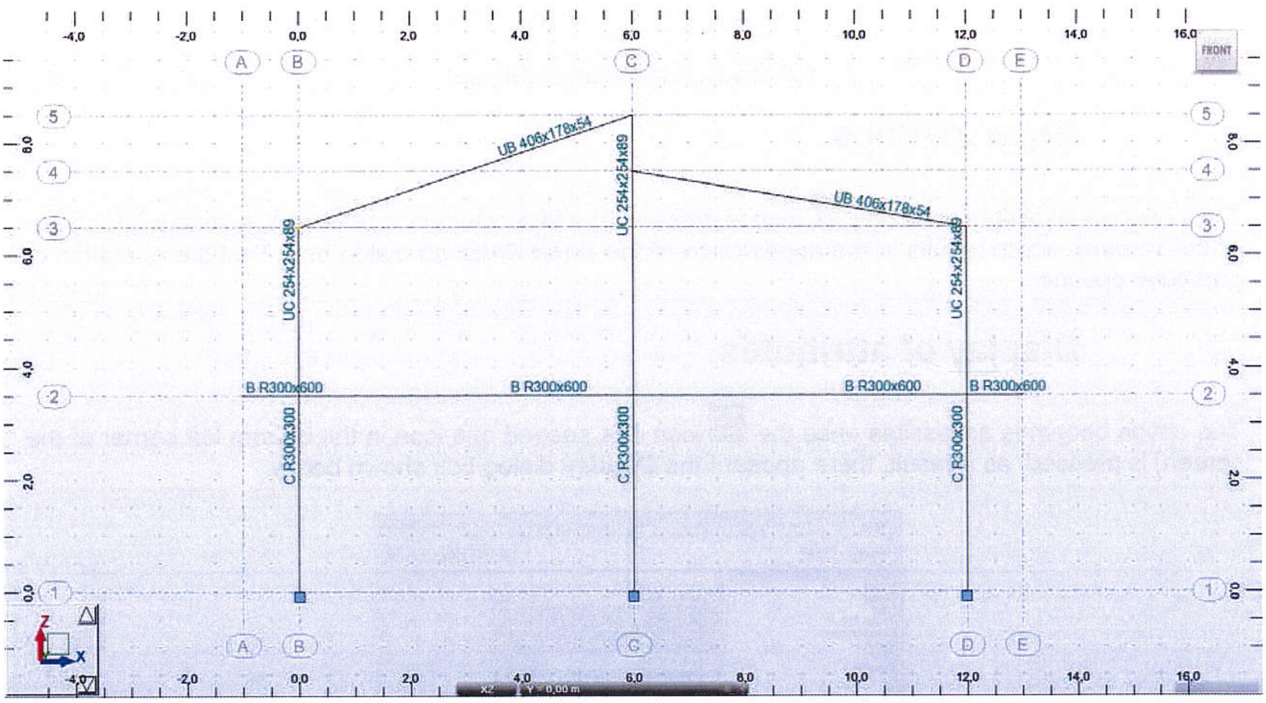
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1. EXERCISE NO. 1 "DEFINITION OF A 2D STRUCTURE – MIXED RC AND STEEL"

1.1. Structure geometry

1.1.1. Proposed model of a 2D frame

Proposed scheme of the frame (RC beam 300x600, columns 300x300).



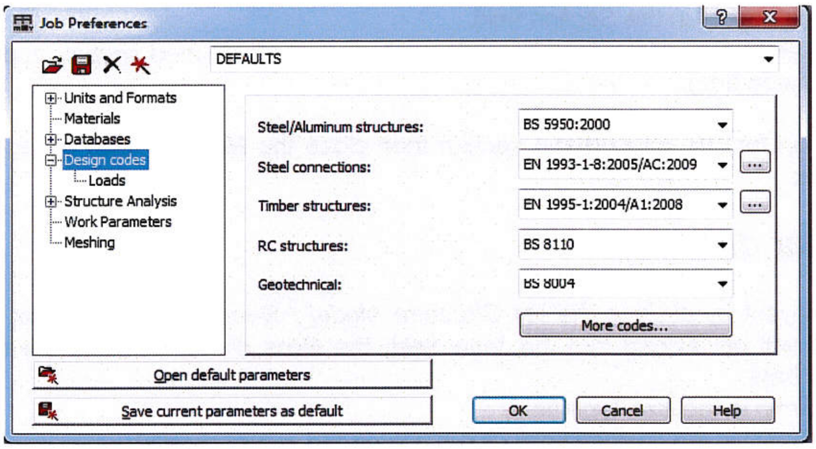
1.1.2. Opening a project

Open ***Autodesk Robot Structural Analysis*** and select structure type *Frame 2D Design* icon  or, if ***Robot*** is already running, select the *File / New Project / Frame 2D Design* command from the main menu.

1.1.3. Setting the codes

One should set the codes to be implemented in the project before commencing the process of structure design. To do so, one should select the *Tools / Job Preferences / Codes* command from the main menu and define the settings shown in the figure below:

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



1.1.4.Structural axes definition

Click the *Axis Definition* icon located in the right-hand toolbar (or use the *Geometry / Axis Definition* command from the main menu), which opens the ***Structural Axis*** dialog box and define structural axes. Introduce the required value into the *Position* field and set it into the *Set of Created Axes* field by pressing **Insert** button. Perform the operation for the following values:

- Set *Numbering* to: A, B, C… *Tab X*: -1; 0; 6; 12; 13;
- Set *Numbering* to: 1, 2, and 3… *Tab Z*: 0; 3.5; 6.5; 7.5; 8.5;

Confirm by clicking *Apply / Close*.

1.1.5.Definition of section applied in the structure

Select the **SECTIONS&MATERIALS** layout by clicking the list box in the top right corner (or click the *Bar Sections* icon located in the right-hand toolbar), which opens the ***Sections*** dialog box and check presence of the following sections:

- C R300x300,
- B R300x600,
- UC 254x254x89,
- UB 406x178x54,

If the above sections are not present in the list of available sections one should define it.

To define a new section, one should press the *New section definition* icon, which results in opening the ***New Section*** dialog box.

To define the *C R300x300* section, one should:

- Set *RC Column* option located in the *Section Type* field,
- Set 300 in the *b* field,
- Set 300 in the *h* field,
- End the operation by pressing the **Add** button (the newly defined section appears in the ***Bar Sections*** dialog box).

To define the *B R300x600* section, one should:

- Set *RC Beam* option located in the *Section Type* field,
- Set 300 in the *b* field,
- Set 600 in the *h* field,
- End the operation by pressing the **Add** button (the newly defined section appears in the ***Bar Sections*** dialog box).

To define the UC 254x254x89 section, one should:

- Set *Steel* in the *Section Type* field,
- Set *Simple Catpro* in the *Database* field,
- Set *UC* in the *Family* field,

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

- Set UC 254x254x89 in the *Section* field,
- End the operation by pressing the **Add** button (the newly defined section appears in the ***Bar Sections*** dialog box).

Repeat the operation for UB 406x178x54 section then close the ***New Section*** dialog box. Close the ***Sections*** dialog box.

1.1.6. Bar definition

Select the **BARS** layout by clicking the list *Structure Model / Bars* from the list box in the top right corner (the screen will be divided into the *View* field, the ***Bars*** dialog box and the *Bars* table) and define the following bars:
- Set *RC Column* in the *Bar Type* field,
- Set *C R300x300* in the *Section* field (if it is not loaded automatically)
- Set the cursor in the *Node Coordinates / Beginning* field and then, in the graphic viewer *View*, select graphically the beginning and end of the column by means of the coordinates of the intersection point of defined axes:
  B1-B2,
  C1-C2,
  D1-D2,

Repeat the operation for the following bars:
- Bar type *RC Beam*,
- Section: *B R300x600*,
  A2-B2,
  B2-C2,
  C2-D2,
  D2-E2,

- Bar type: *Column*,
- Section: UC 254x254x89,
  B2-B3,
  C2-C5,
  D2-D3,

- Bar type: *Beam*,
- Section: UB 406x178x54,
  B3-C5,
  C4-D3,

1.1.7.Support definition

Select the **SUPPORTS** layout by clicking the list *Structure Model / Supports* from the list box in the top right corner (the screen will be divided into the *View* viewer, the ***Supports*** dialog box and the *Supports* table) and define the following bars:
- Highlight *Fixed* in the list of supports
- Select *Point/Node* in the field *Current Selection* and set the cursor in the dialog filed,
- Go to the graphical viewer *View*, select with window the following nodes: 1, 3, and 5, (bottom nodes of columns located on the axis 1). One may also type the relevant numbers into the *Point/Node* field.
- Press the **Apply** button.

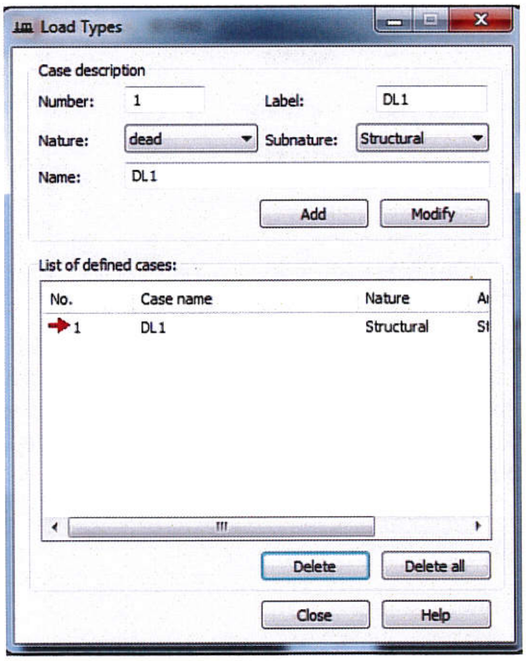
The program will apply supports to the three selected nodes, which will be displayed in the *View* viewer.

Robobat Polska Sp. z o.o., ul. Radzikowskiego 47D, 31-315 Kraków, Poland
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1.1.8.Load definition

Go to the **LOADS** layout selecting *Structure Model / Loads* option from the menu. The screen will be divided into three parts: *graphical viewer*, *Loads* table and ***Load Types*** dialog box. Define the load cases proposed above:

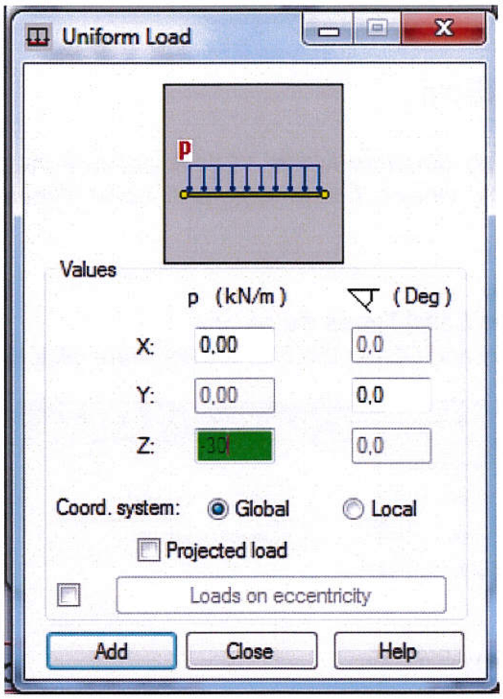
- DL1
  - select *Nature*: *dead* in the ***Load Types*** dialog box
  - type 1 in the *Number* field and *DL1* in the *Name* field; then, click the **New** button



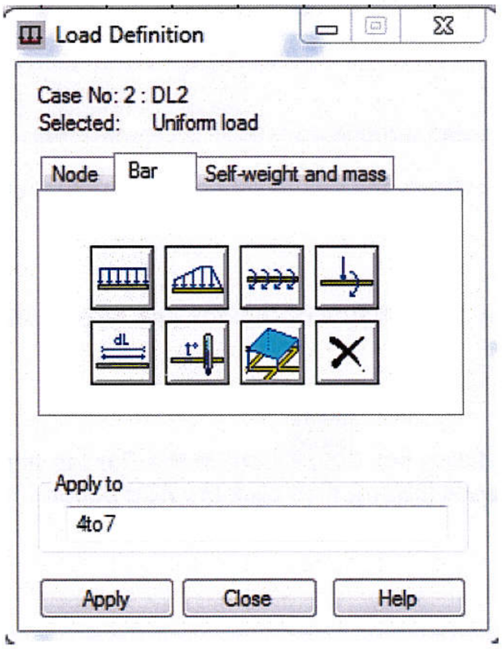
The program will define automatically the self-weight on all structure bars.

- DL2
  - select *Nature*: *dead* in the ***Load Types*** dialog box and type *Number* 2 and *Name*: *DL2*, press the **Add** button
  - select the *Load Definition* icon from the right toolbar
  - open the ***Uniform Load*** dialog box (icon) in the *Bar* tab and type pz = -30 [kN/m] in the edit field *Z* (column of p loads [kN/m]) then click the **Add** button

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

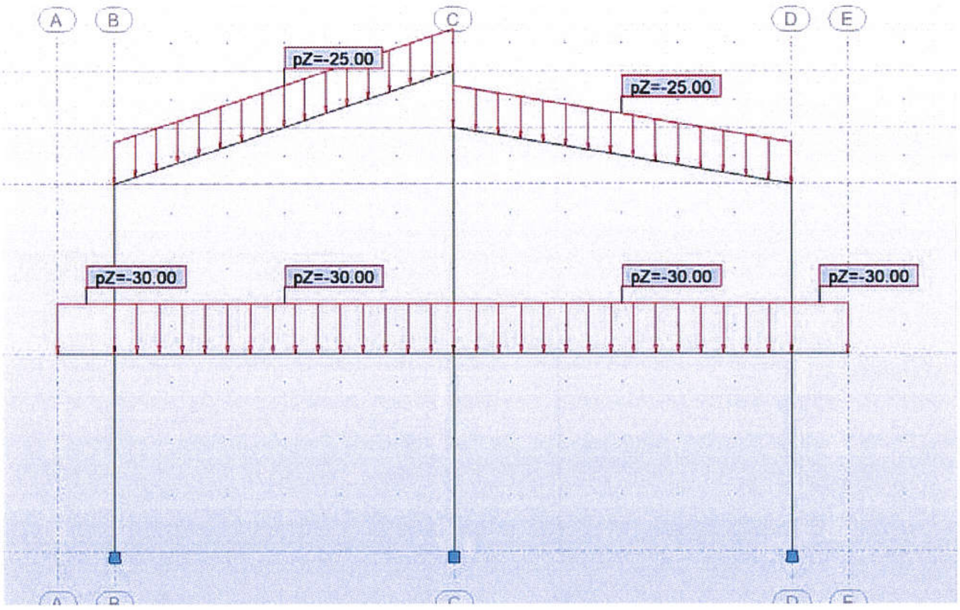


– apply defined uniform load to structure spans (bars 4,5,6,7,). Go to the *Apply To* field in the ***Load Definition*** dialog box; the mouse cursor will change from the *choice* mode into the *selection* mode. Then, indicate beams 4,5,6,7 in the graphical viewer (with the **Ctrl** button pressed, click on the bars). The bar numbers will appear in the *Apply To* field. Confirm the selection by pressing the **Apply** button

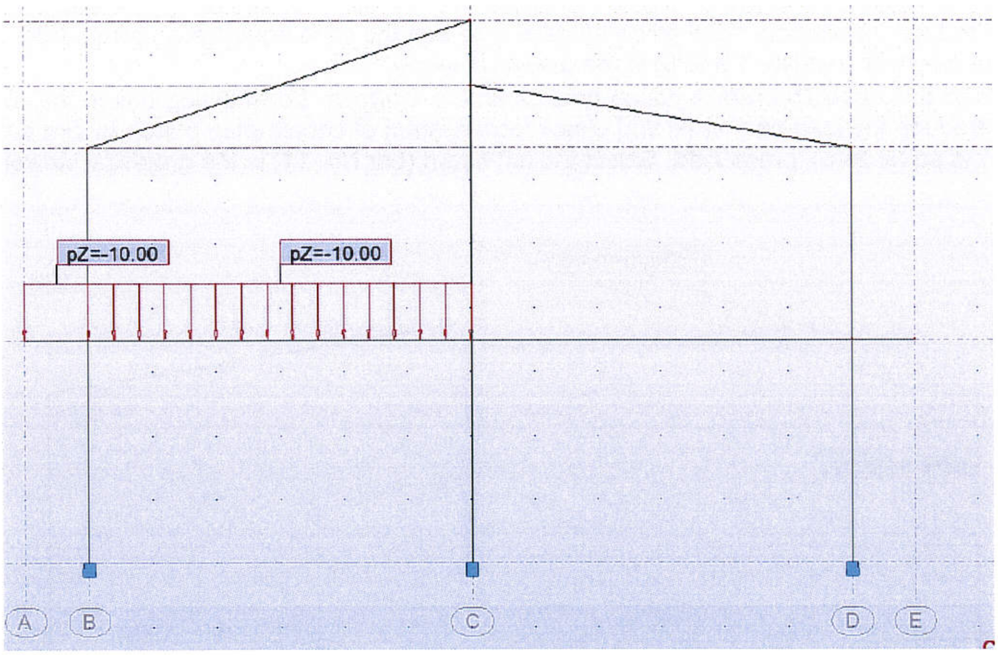


Define uniform loads pz=-25 kN/m for roof beams 11, 12

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

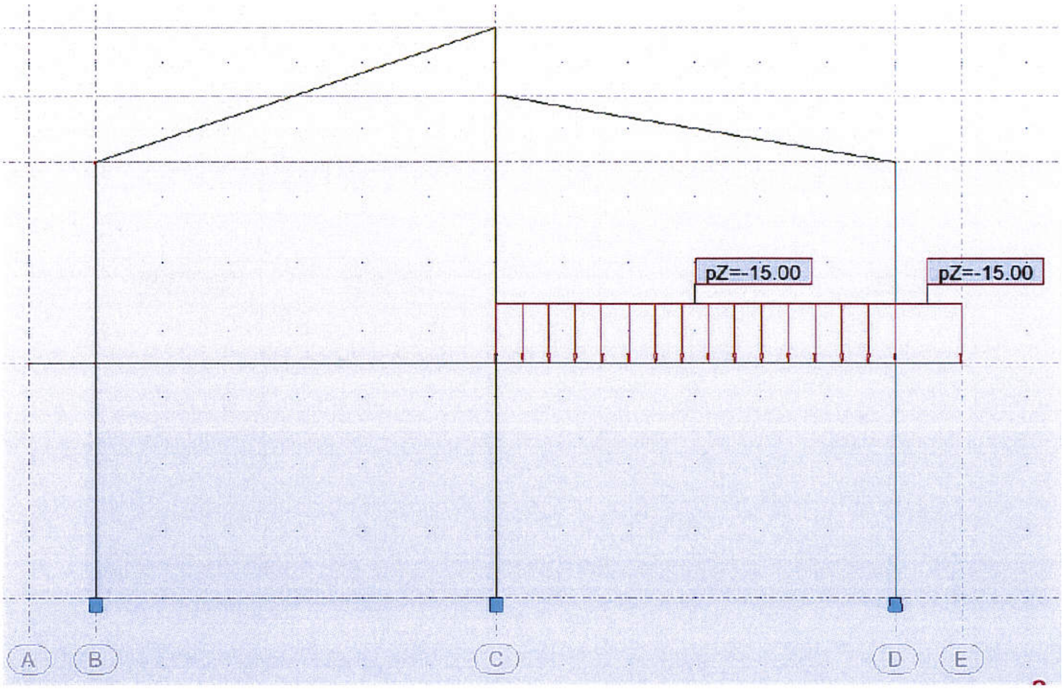


- LL1
  - introduce *Nature: live*, *Number: 3* and *Name: LL1* in the ***Load Types*** dialog box and click the **Add** button
  - in the ***Bar*** tab of the ***Load Definition*** dialog box, enter the definition of uniform loads on bars. Introduce load pz = -10 [kN] then **Add**. Apply this load to elements 4 and 5

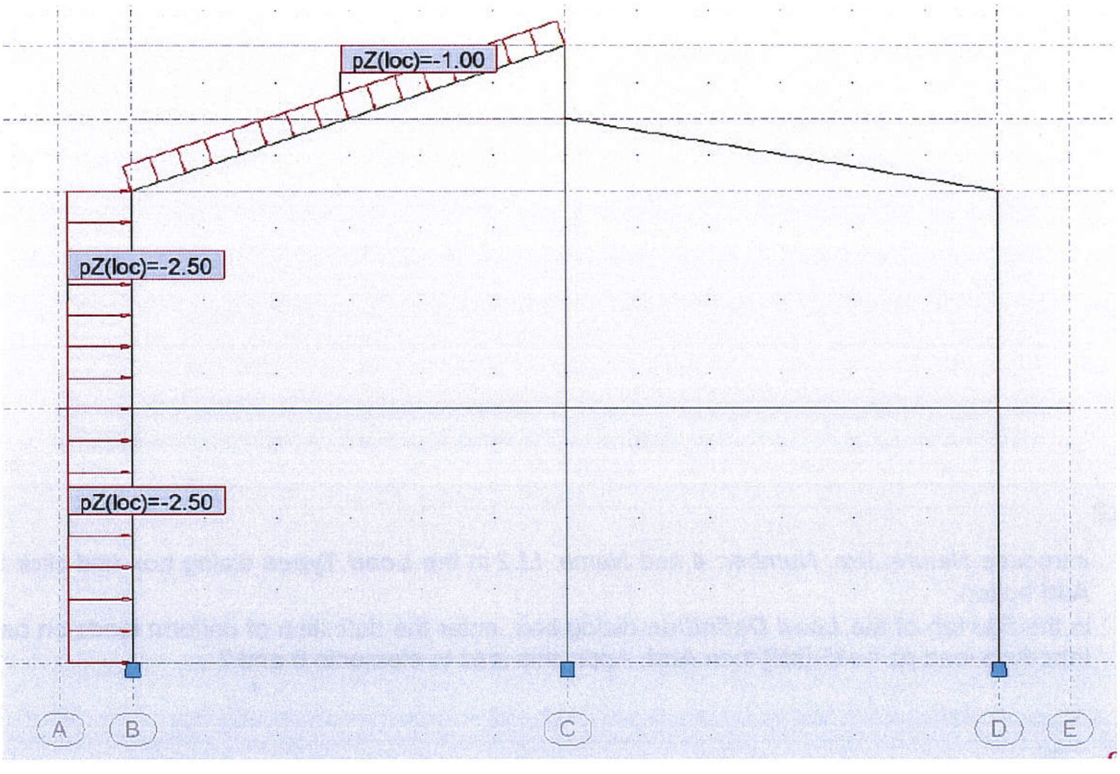


- LL2
  - Introduce *Nature*: *live*, *Number: 4* and *Name*: *LL2* in the ***Load Types*** dialog box and click the **Add** button.
  - in the ***Bar*** tab of the ***Load Definition*** dialog box, enter the definition of uniform loads on bars. Introduce load pz = -15 [kN] then **Add**. Apply this load to elements 6 and 7

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



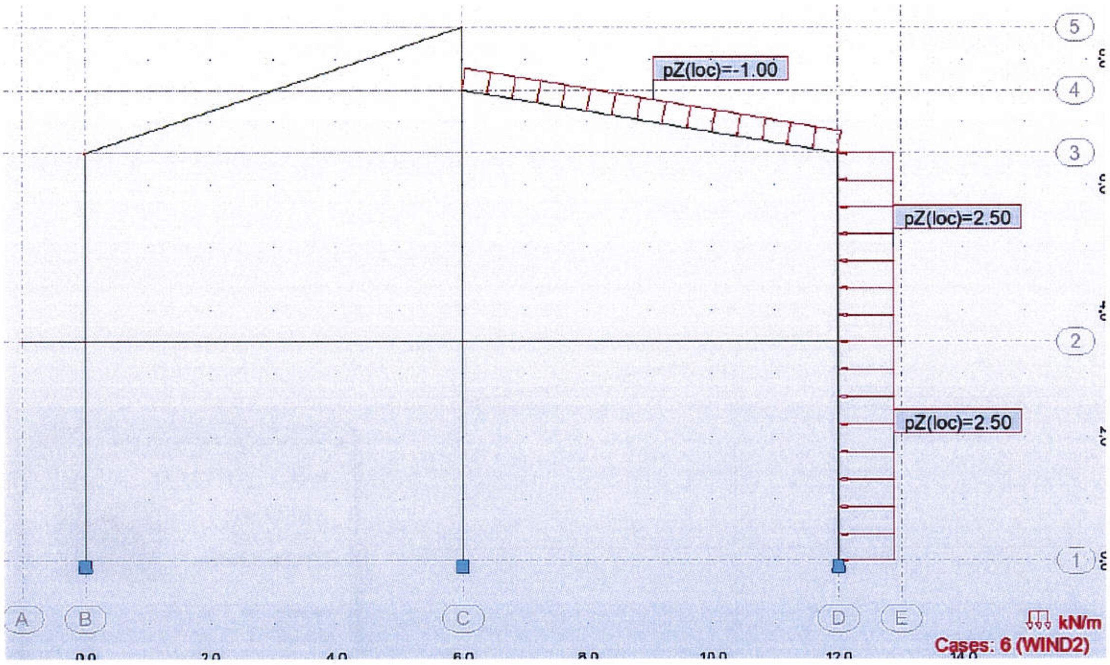
- wind 1
  - introduce *Nature*: *wind*, *Number: 5* and *Name*: *wind 1* in the ***Load Types*** dialog box, confirm with the **New** button
  - go to the ***Load Definition*** dialog box. Open the ***Uniform Load*** dialog box in the *Bar* tab and introduce the load px = 2,5 [kN/m] (make sure that the pz is equal zero), press **Add.** Select the left columns (bar No. 1 and 8) in the graphical viewer
  - go to the ***Load Definition*** dialog box. Open the ***Uniform Load*** dialog box in the *Bar* tab and introduce the load pz = -1 [kN/m] check local system of coordinates button (make sure that the px is equal zero), press **Add**. Select the left beam (bar No. 11) in the graphical viewer



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

- wind 2

  - introduce *Nature*: *wind*, *Number*: *6* and *Name*: *wind 2* in the ***Load Types*** dialog box, confirm with the **New** button
  - go to the ***Load Definition*** dialog box. Open the ***Uniform Load*** dialog box in the *Bar* tab and introduce the load px = -2,5 [kN/m] (make sure that the pz is equal zero), press **Add**. Select the right columns (bar No. 3 and 10) in the graphical viewer
  - go to the ***Load Definition*** dialog box. Open the ***Uniform Load*** dialog box in the *Bar* tab and introduce the load pz = -1 [kN/m] check local system of coordinates button (make sure that the px is equal zero), press **Add**. Select the right beam (bar No. 12) in the graphical viewer



Loads can be modified graphically or in load table. Go to the *Loads* table. The *Edit* tab allows one to modify the existing loads and define new ones.

Loads

Case	Load type	List									
1:DL1	self-weight	1to12	Whole structu	-Z	Factor=1,00	MEMO:					
2:DL2	uniform load	4to7	PX=0,0	PZ=-30,00	global	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
4:LL2	uniform load	6 7	PX=0,0	PZ=-15,00	global	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
3:LL1	uniform load	4 5	PX=0,0	PZ=-10,00	global	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
5:WIND1	uniform load	1 8	PX=0,0	PZ=-2,50	local	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
5:WIND1	uniform load	11	PX=0,0	PZ=-1,00	local	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
6:WIND2	uniform load	12	PX=0,0	PZ=-1,00	local	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
6:WIND2	uniform load	3 10	PX=0,0	PZ=2,50	local	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
2:DL2	uniform load	11 12	PX=0,0	PZ=-25,00	global	not project.	absolute	BE=0,0	DZ=0,0	MEMO:	
*											

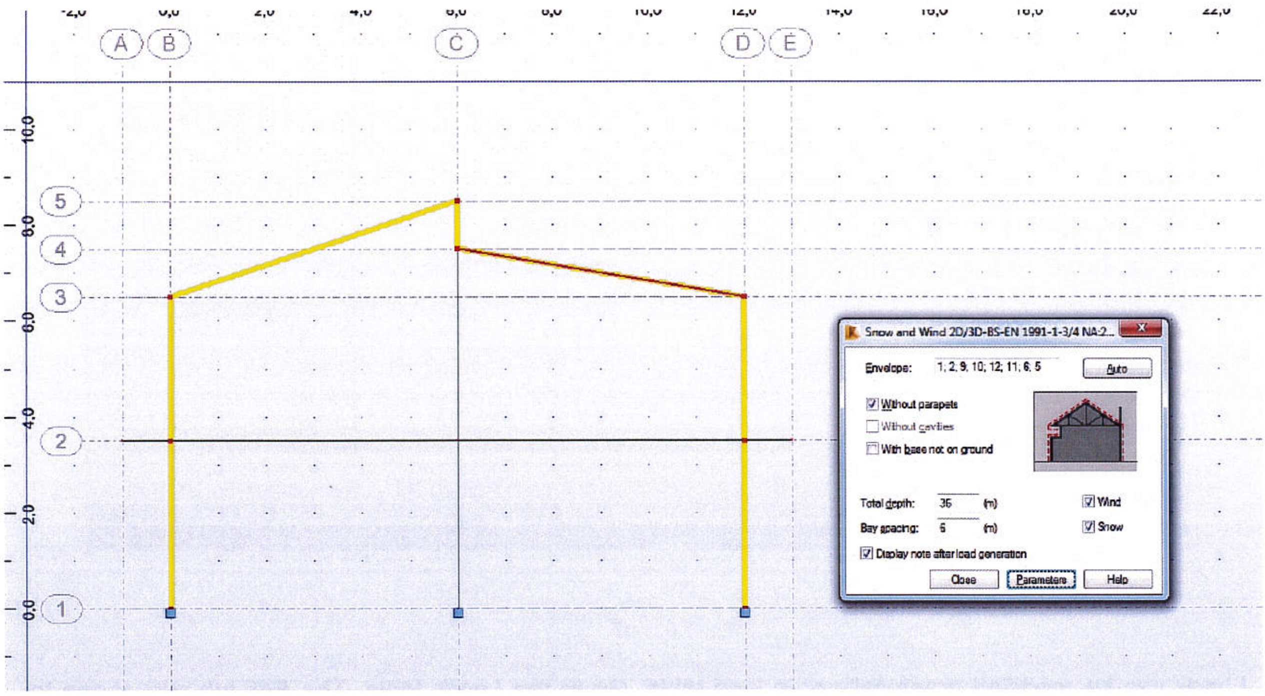
Save structure under the name 3-loads

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1.1.9.Climatic load definition

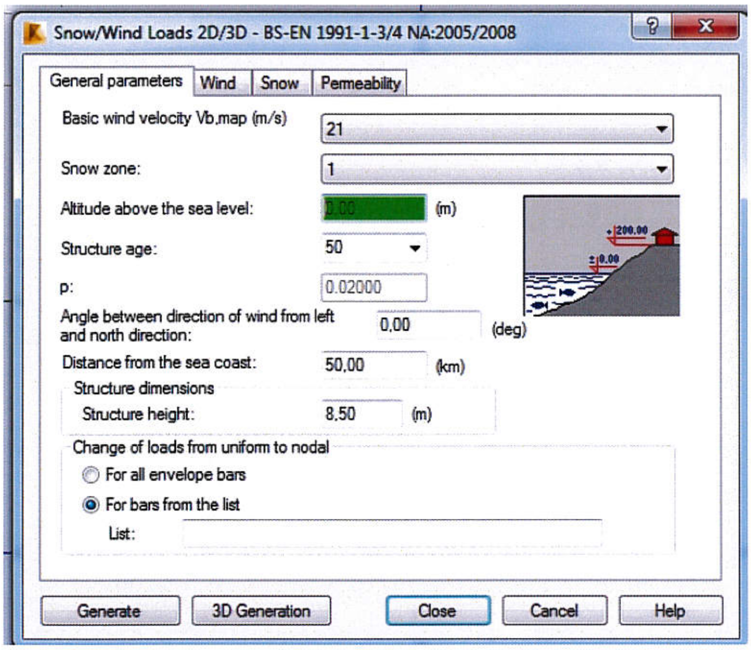
Select the **LOADS** layout by selecting the command *Structure Model / Loads* from the list box in the top right corner (the screen will be divided into the graphic viewer ***View***, the ***Load Types*** dialog box and the *Loads* table)

Select Snow/Wind Loads 2D/3D  form right vertical toolbar. In the following window
- click Auto
- check Without parapets option
- set Total depth to 36 m
- set Bay Spacing 6 m



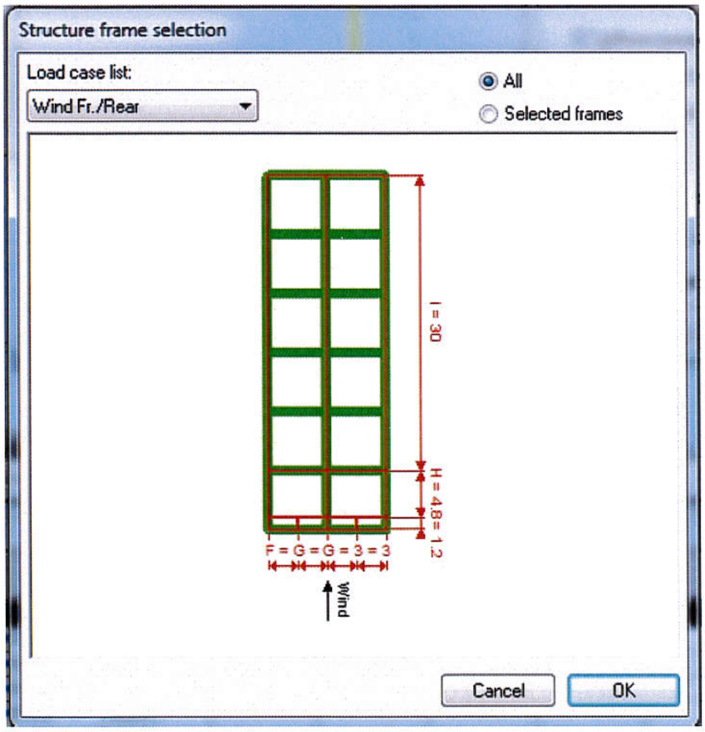
- click Parameters

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



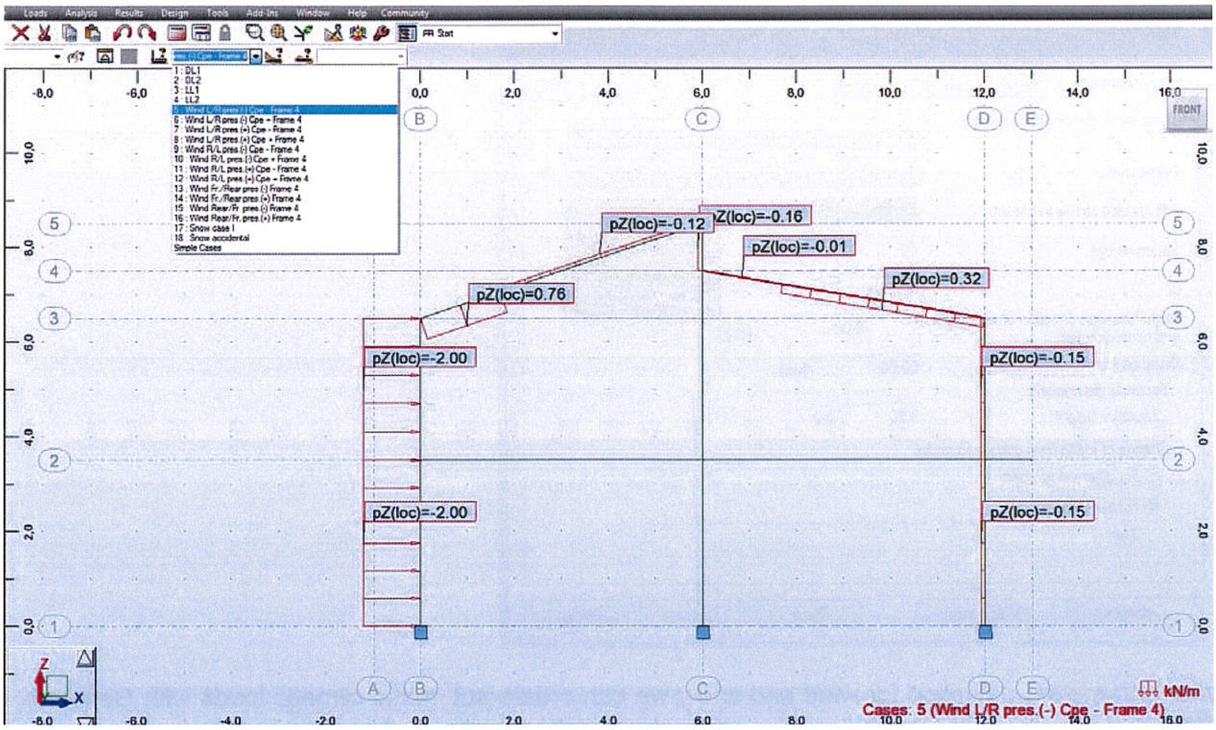
- after parameters definition for wind and snow we can create automatic climatic loads with Generate option

Then program ask for which zone (according to EC) loads will be generated



Following load cases will be created

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Save structure as 3- climatic loads

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

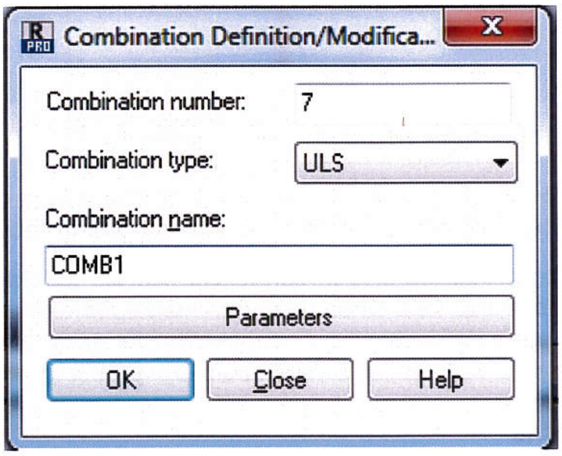
1.1.10. Load Combinations defined manually

Open 3-loads file

We can create load combinations using two methods. It can be done in manual mode or with use of Automatic Combination option

While being in the LOADS layout enter the load combination definition dialog box:

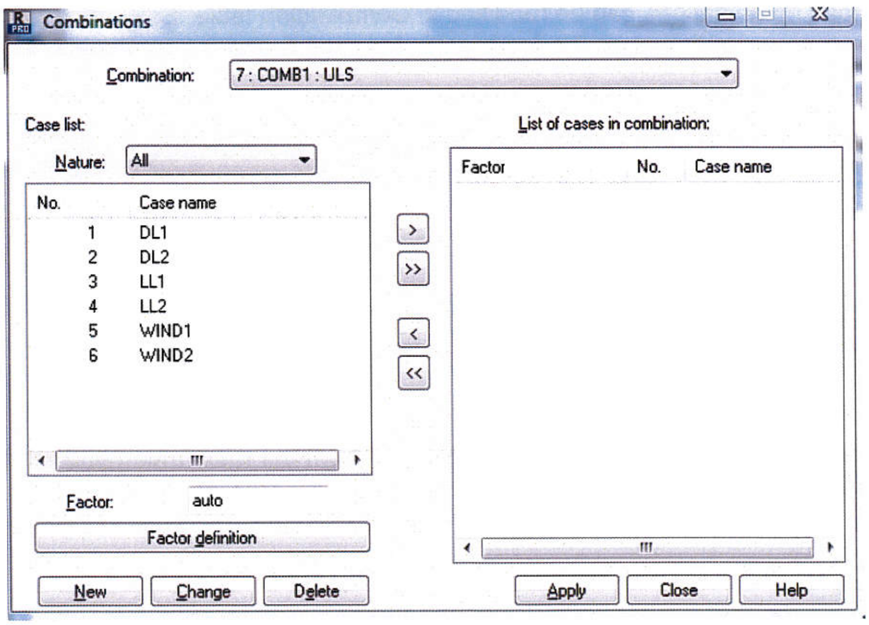
- from the toolbar by pressing the *Combinations* icon or
- by selecting the *Loads / Combinations* command from the main menu



Create a new load combination in the dialog box, i.e.:

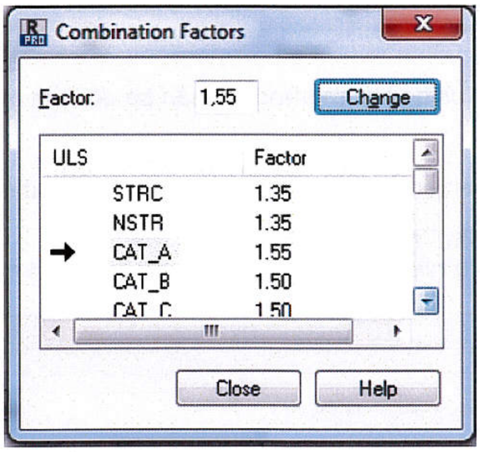
- introduce *Combination Name*: COMB1
- check out *Combination Type*: ULS (ultimate limit state)
- set *Nature*: dead
- click the **OK** button

There appears a new ***Combinations*** dialog box that allows one to define and modify the factors of a given combination.



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Press the **Factor Definition** button and set the arrow to point the *CAT A* label. In the *Factor* edit field introduce the new value 1.55 and confirm by clicking **Change** button. Close the dialog box by pressing the **Close** button.



Move the 1 to 5 load cases to the ***List of Cases in Combination*** dialog box. By means of the > button, one can move each load case separately, or one can choose to move all the cases by means of the >> button. The first solution allows one to control the factors for each separate case i.e. one may define the factor in the *Factor* edit field when moving a load case to the right. If the *Auto* option is set, the factor will assume the value defined in the ***Combination Factors*** dialog box

Confirm the creation of the COMB1 load combination by pressing **Apply** button. Then New combinations can be defined

Define the next SLS combinations

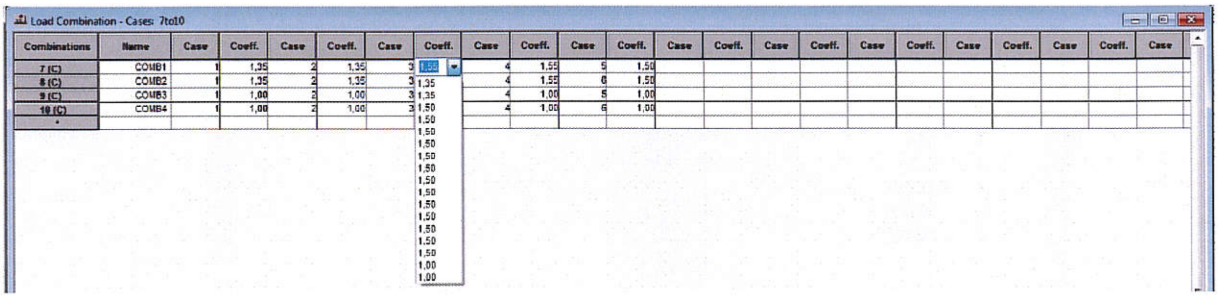
COMB 2 (ULS 1*1.35+2*1.35+3*1.55+4*1.55+6*1.5)

And following SLS:QPR combinations

COMB 3 (SLS:QPR) 1*1.0+2*1.0+3*1.0+4*1.0+5*1.0)
COMB 4 (SLS:QPR) 1*1.0+2*1.0+3*1.0+4*1.0+6*1.0)

Do not forget to press Apply after last combination definition

Manually defined load combinations can be modified in Combination table. Select *Loads / Combination Table* from the main menu and change the load factor for case 5 and 6 I combinations 1 and 2 from 1.5 to 1.60 in the **Edit** tab.



In order to save the project select *File / Save As* command from the main menu and set the file name as 5-manuall combination.

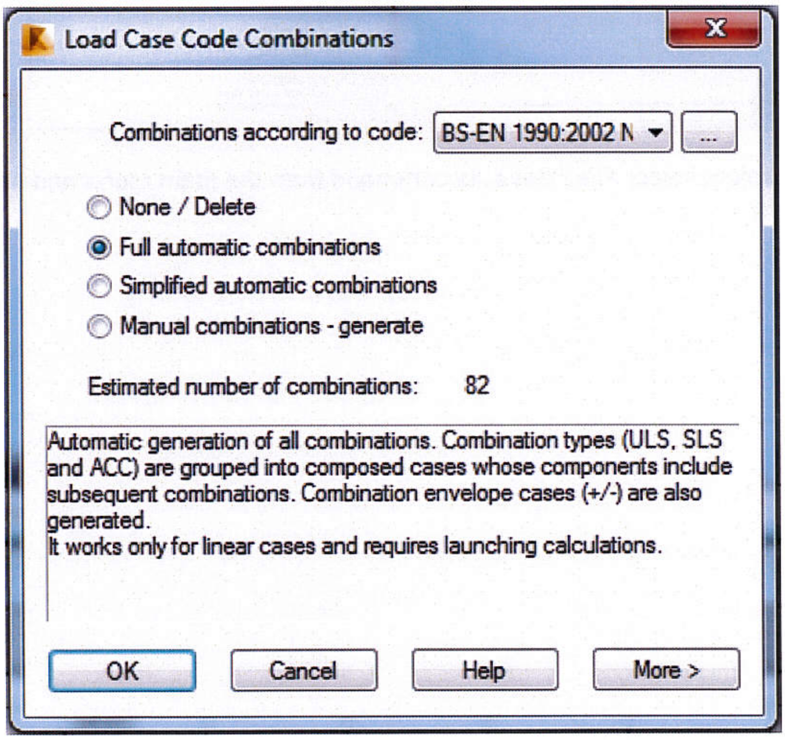
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1.1.11. Load Combinations defined automatically

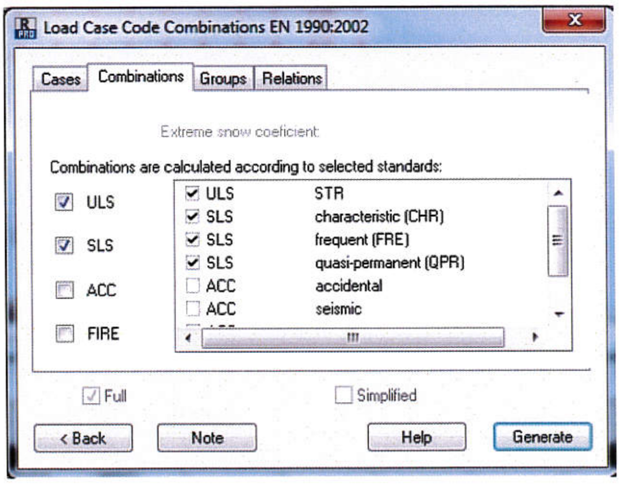
Open 3-loads file

While being in the LOADS layout enter the code combination definition dialog box :

- by selecting the *Loads / Code Combinations* command from the main menu
- select *Full automatic combinations* then *More*



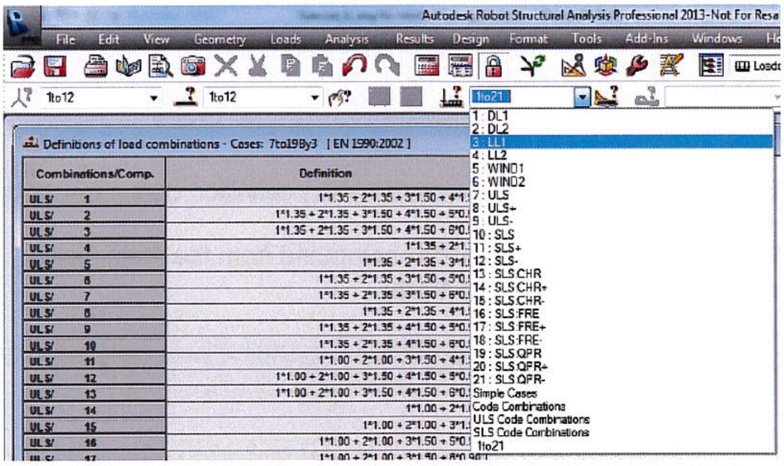
- select required combination types from Combination tab



- by use of Generate create Code combinations

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Code combinations will be created after structure analysis and can be controlled in load combination table (Loads/ Combination table option in main menu)



In order to save the project select *File / Save As* command from the main menu and set the file name as 6-code combination

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1.2. Results analysis

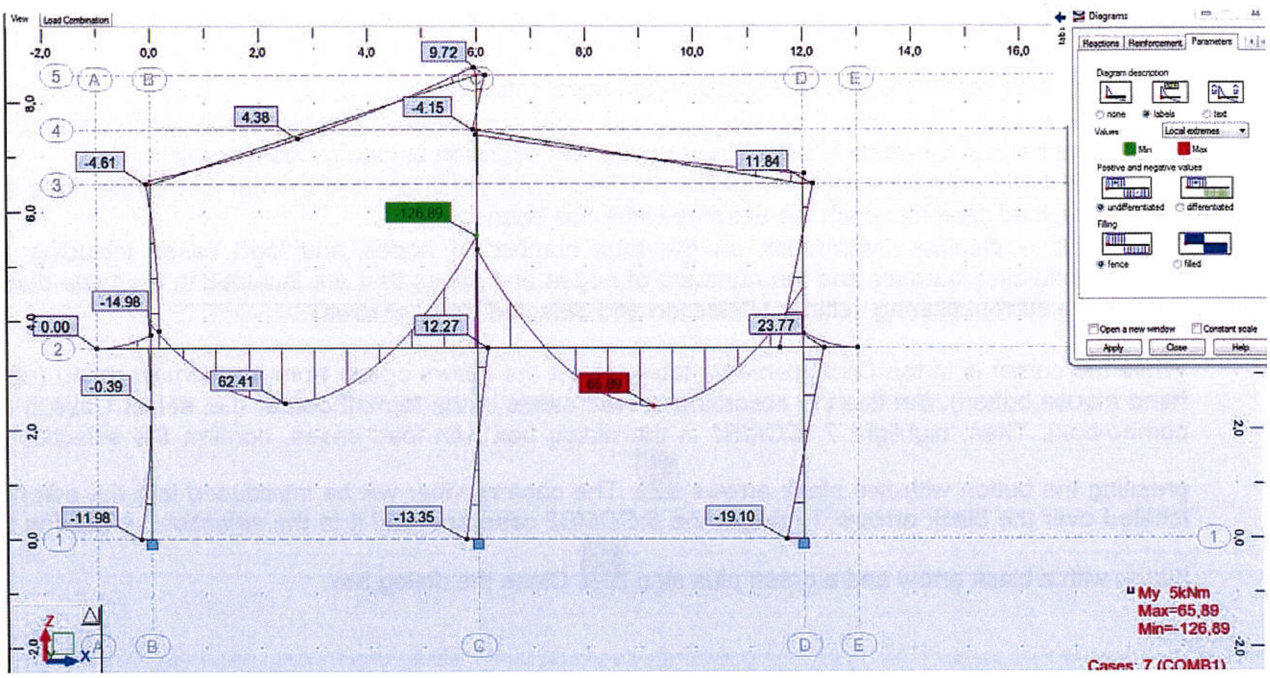
Open 5-manuall combination file

Run calculations for the designed structure by means of the *Analysis / Calculations* command from the main menu or by pressing the *Calculations* icon.

1.2.1. Analysis of results in graphical form

Open the *Results / Results* layout. The screen will be divided into three parts: the graphical viewer *View*, the *Reactions* table and the ***Diagrams*** dialog box.
Restore the default attributes of the structure view by clicking the icon in the bottom left corner of the screen. Supports, sections, numbers of nodes and bars and the dimension lines should disappear.
The ***Diagrams*** dialog box contains six tabs:

- *NTM* tab allows one to display the internal forces and moments
- *Deformation* tab allows one to visualize displacements and deformations
- *Stresses* tab allows one to select a type of stresses
- *Reactions* tab allows one to select a type of reactions
- *Reinforcement* tab allows one to select the quantities the diagrams of which are to be presented for structure bars: theoretical reinforcement, real reinforcement, reinforcement ratio, stirrup spacing etc.
- *Parameters* tab allows one to set the parameters of diagram and description display



Additionally, the *NTM, Deformation* and *Stresses* tabs contain buttons used to change the scale of the relevant diagrams:

- button that increase/decrease discretely the scale: **Scale +** and **Scale –**
- the **Normalize** button that sets the scale to assure that the diagrams do not extend outside the graphical viewer and remain legible

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

The *Open a New Window* option – located at the bottom of the ***Diagrams*** dialog box – allows one to view the graphical results in a new window. If one selects only some of the structure elements in the *View* window, then, only these elements will be visible in the new window.
Check out the *Deformation* option in the *Deformation* tab. Confirm the selection by clicking **Apply**. Introduce the cursor into the *Cases* list box (upper toolbar) and change the load cases by means of keyboard cursors (up and down arrows) and observe the results in the *View* window. If necessary, re-scale the drawing by means of the **Normalize** button.
Prepare a screen capture for load case 7: *Comb 1* – select the *Screen Capture* option from the context menu (right-hand mouse button). Leave the default name and confirm by clicking **OK**.
Disable the display of displacements by switching off the *Deformation* checkbox in the *Deformation* tab. Confirm by clicking **Apply**.
In the *View* viewer: select spans – indicate beams (with the left mouse button pressed) by dragging the window-cursor from the top left corner to the bottom right one. Check out the *Horizontal* option in the *Diagram Description* field of the *Parameters* tab. Then, click the *Open a New Window* option at the bottom of the dialog box. Switch on display of *MY* moments in the *NTM* tab. Confirm by clicking **Apply**.
As before, look through the results for all cases – the ***Case Selection*** dialog box. Prepare screen captures for the cases with combinations *7: Comb 1*. Leave the default name. Close the window with the diagram by means of the **Exit** button.

1.2.2. Analysis of results in tables

The following tables of results are available from the top menu, item *Results*:

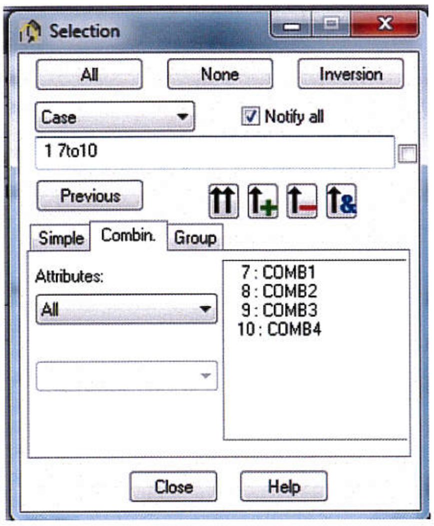
- Reactions
- Displacements
- Deflections
- Forces
- Stresses

Open any of the tables, e.g. the table of displacements – *Results / Displacements* command from the main menu. There are four tabs at the table bottom (the meaning of the tabs is analogous in the remaining tables):

- *Values* – overview of displacements and nodal rotations
- *Envelope* – presents the maximum and minimum value of displacements and rotations for each node; the value is select from the current selection of load cases (grey fields)
- *Global Extremes* – presents global maximum and minimum and the number of the node and the load case for which the extreme value has been found
- *Info* – displays information on the total number of nodes and load cases including the node/case number and the numbers of nodes and cases that are included in the table due to the current filtering settings (Selection and Selected Number rows)

While the cursor is in the *Displacements* table, select the *Filters* option from the context menu (right-hand mouse button). Set filtering according to load cases in the top left corner (i.e. select *Case* in the combo-box). Then, highlight *7: COMB1* in the dialog box with load cases, confirm the selection by pressing the button with two black arrows . The case number will be introduced into the edit field located over the black arrows. Highlight the *8:COMB2* case and add it to the selection by clicking the button with a black arrow and a green plus sign . Close the dialog box.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



1.2.3. Detailed analysis of bars in graphical form

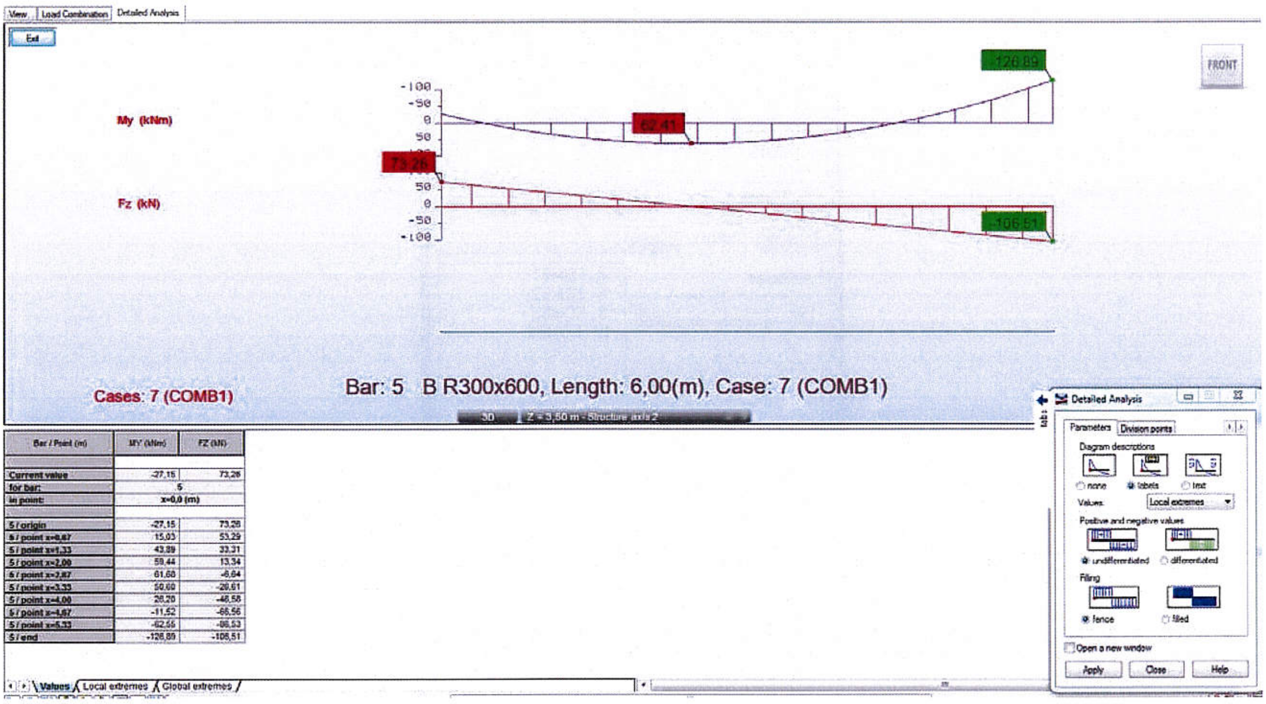
Select the *Table Columns* option from the context menu. Check out the display of *RY* rotations in the *Displacements* tab. Go to the *General* tab and click *Coordinates* in the *Element Data Selection* field. Confirm by clicking **OK**.
While the cursor is located in the Values tab, prepare a screen capture – select the *Screen Capture* option from the context menu. Leave the default name and press **OK**.

Go to the **DETAILED ANALYSIS** layout (*Results / Detailed Analysis*). The screen will be divided into two parts: the *View* graphical viewer and the ***Detailed Analysis*** dialog box.
The ***Detailed Analysis*** dialog box consists of five tabs:

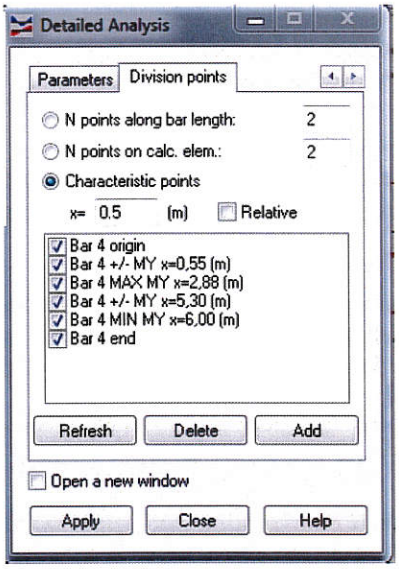
- *NTM* – displays internal forces, moments and displacements
- *Stresses* – displays stresses
- *Reinforcement* – displays reinforcement
- *Parameters* – sets the parameter of diagram and description display
- *Division points* – that tab allows one to select the characteristic points on a bar for which the results will be presented

Indicate the left span in the *View* window. Check out the *Open a new window* checkbox at the bottom of the dialog box. Press the **Apply** button. The currently, opened window is divided into two parts: the top one contains results in graphical form, while the bottom one displays the results in the form of a table.
Click the *maximum – S max* option in the *Stresses* tab. Activate the display of horizontal descriptions in the *Parameters* tab (click *Horizontal* in the *Diagram Description* field). Confirm the selection by clicking **Apply**.
Go to the *Division Points* tab. Click the *N points along bar length* option and type 10 in the field beside. Press **Apply**. The window with results will display results (S max stress values) for 10 points evenly distributed over bar length.
Check out the *characteristic points* option in the same tab. Pressing the **Refresh** button will result in introducing into the dialog box the coordinates of the points for which the maximum and minimum values of *S max* is obtained (*MIN S max x = ... (m)* and *MAX S max x = ... (m)*, as well as the beginning and end points of the bar (*origin* and *end*). Once the **Apply** button is pressed, the results for the four characteristic points will be introduced into the results window.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



It is also possible to display the results for the user-defined points. To do so one should click the *relative* checkbox in the *Division points* tab. Introduce x = 0.4 in the edit field. Press the **Add** button. The *user x = 0,4* row will be added in the dialog box below. By analogy, introduce the points with the following coordinates: x = 0.5 and x = 0.6. Confirm by clicking **Apply**.



Select case *7:COMB1* in the *Case Selection* combo-box. Go to the *Detailed Analysis* combo-box and press the **Refresh** button and then the **Apply** button.
Go to the window with graphical results and prepare a screen capture with the default name. Close the window by pressing **Exit**.
In order to save the project select *File / Save As* command from the main menu and set the file name as **7-results**

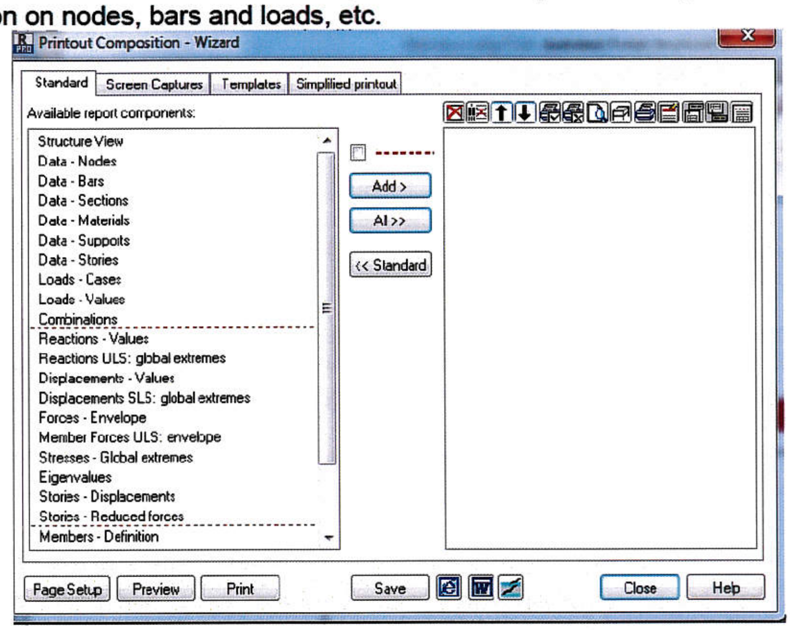
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

1.3. Preparation of a document for printing

1.3.1. Printout composition

Select the *File / Printout Composition* command from the main menu. The ***Printout Composition – Wizard*** dialog box consists of four tabs:

- *Standard* – allows one to add standard positions to the printout composition, i.e. structure view, information on nodes, bars and loads, etc.



- *Screen Captures* – allows one to add user-defined screen captures to project documentation
- *Templates* – allows one to use ready-made templates containing certain standard elements; the tab allows one also to save a user-defined template under an arbitrary name; the template must be created in the Simplified printout tab
- *Simplified printout* – allows one to create a template by selecting and filtering the relevant components

Go to the *Simplified printout* tab. The left-hand panel of the dialog box contains elements that may be introduced into the final printout. Each item is accompanied by a checkbox that may assume one of the following three states:

– means that the element will not be introduced into the final printout

– means that the element will be introduced into the final printout with all its components (all the components of this category will be displayed – filtering will be disregarded)

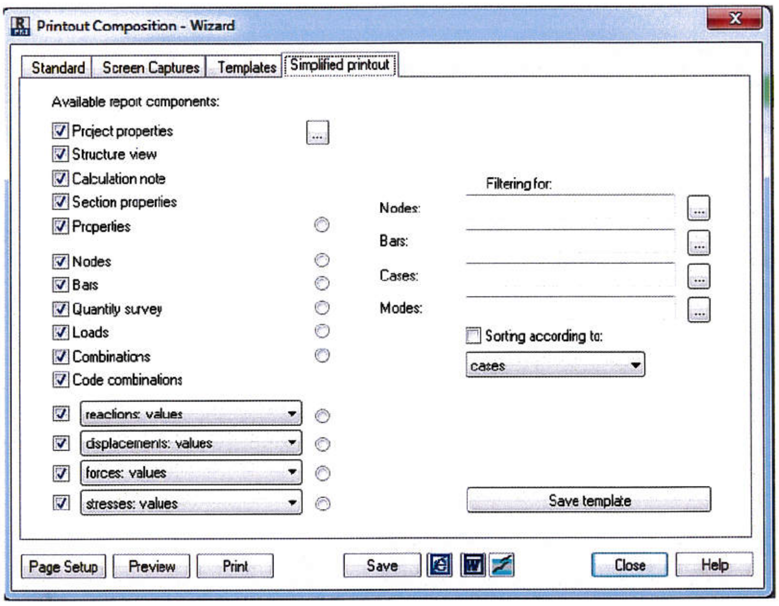
– means that the element will be introduced into the final printout but only the filtered components will be recognized

1.3.2. Filtering of the components

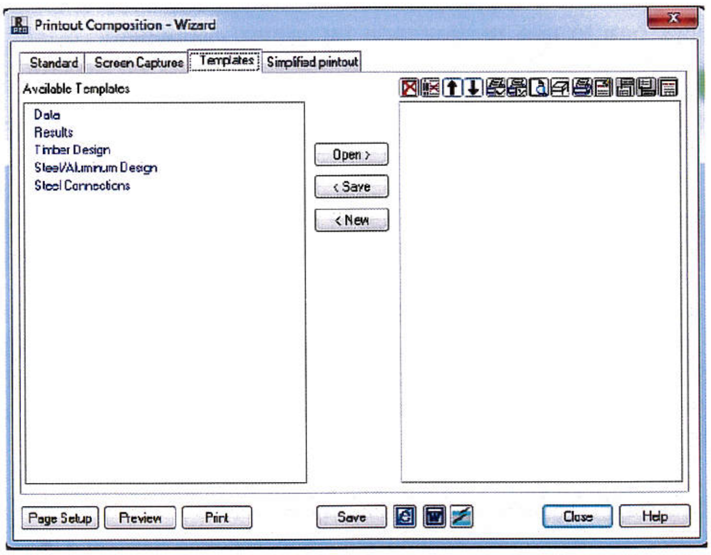
A radio button located to the right (*Filtering for*) accompanies each of the elements in the left-hand part of the dialog box (*Available report components*). Clicking a radio button activates one of the four fields: *Nodes*, *Bars*, *Cases*, and *Modes*. For instance, to filter the components *"forces: values"*, one should click the radio button to the right of the component. These results in activation of the following fields: *Bars*, *Cases* and *Modes*.
Uncheck all the check boxes, leaving only the following components active: *Bars*, *Loads* and *forces: values*. Before filtering the components *Loads* and *forces: values* grey them out, or the program will not recognize our selection:

- filtering of *Loads* components – click the radio button and introduce 1to7 in the *Cases* field
- filtering of *forces: values* components – click the radio button and introduce 1to4 in the *Bars* field and 1to3 in the *Cases* field

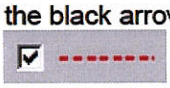
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Once the **Save template** button is pressed, the program automatically goes to the *Templates* tab. The tab is divided into two panels. The left one contains the names of standard templates. The right one contains the components that will be included in the final printout, i.e. – in our case – the components defined in the *Simplified printout* tab.
Create a new template by clicking the **New** button. Introduce the name *"My template"* in the left-hand panel and confirm by pressing **Enter**. Highlight the newly created template. Save the template by clicking the **Save** button.
The top right corner of the dialog box contains a toolbar with icons. Press the *Delete all components*  icon. Then, click the "My template" template in the left-hand panel and press the **Open** button. Thus, one can check if the template has been created correctly.

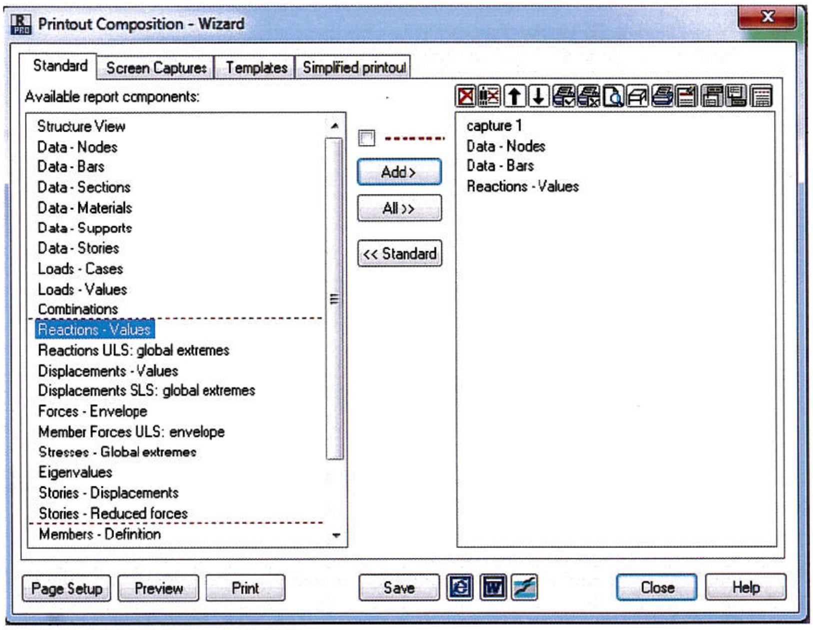


Go to the *Screen Captures* tab. Indicate the screen capture named "Capture 1" and move it to the right-hand panel by clicking the **Add** button. Then, move the screen capture to the top of the printout composition list by means of the black arrows from the toolbar.

Comment: *There is the*  *checkbox over the **Add** button. It is checked out by default. It means that the elements added to printout composition will be located in separate pages.*

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

While the right-hand panel is active, one may activate the context menu. Indicate *Loads* component and activate the context menu by means of the right-hand mouse button. Uncheck the *Start on a new page*. Thus, the *Loads* component will be placed on the same page as the *Bars* component.
Go to the left-hand panel and indicate all screen captures apart from "Capture 1" (click the selected elements with the *Ctrl* button pressed). Uncheck the checkbox imposing location on separate pages (the one beside the red dashed line). Move the screen captures to the right by means of the **Add** button.
Indicate the component *"View – Deformation; Cases: 7 (Comb1)"*. Select the *Start on a new page* option from the context menu. There will appear a dashed red line indicating that the given elements start at a new page. Reopen the context menu and select the *Note before* option. Introduce a text in the opened editor, for instance: "Results in the graphical form". Close the editor by confirming the message on saving the changes.
Indicate the first components in the list, i.e. "Capture 1", and select the **Title** option from the context menu. Change the drawing name in the opened editor to "Structure – Frame 2D". Save changes by means of the *File / Save* command from the main menu of the editor. Close the editor.



Print the created document. Press the **Print** button at the bottom of the dialog box. Close the ***Printout Composition*** dialog box.

Using the *File / Save As* command from the main menu save the file under the name **Exercise 1-printout composition**.

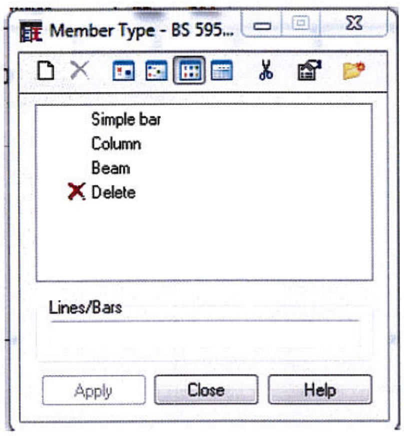
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

2. EXERCISE NO. 2 "DESIGN OF STEEL MEMBERS"

2.1. Open the 7-results file

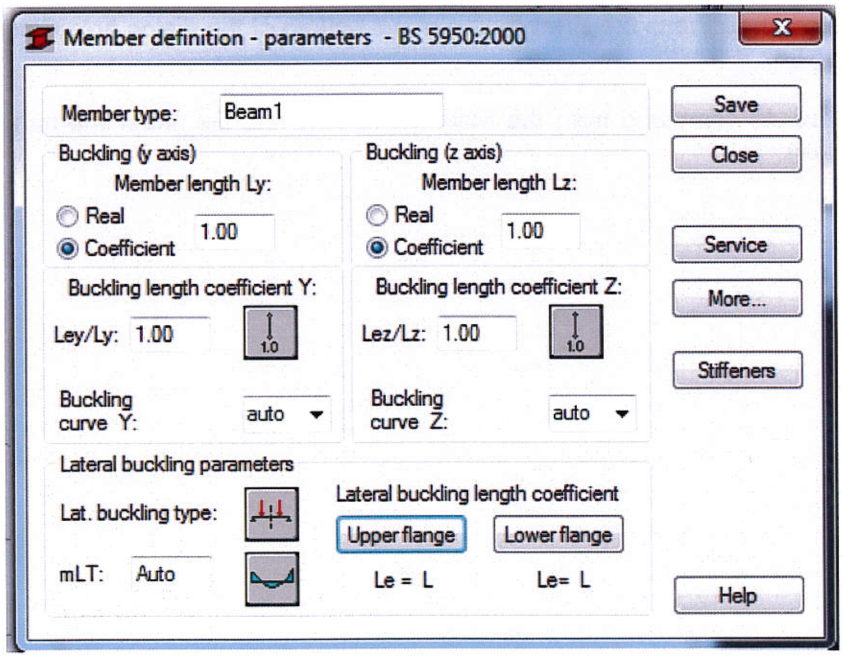
2.2. Definition of code parameters for particular structure members

In order to define code parameters, one should select the *Design / Steel members design option / Code parameters* command from the main menu. Once the dialog box is open (see the picture below), click the *New* icon.

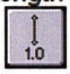


The operation results in opening the ***Member Definition – Parameters*** dialog box (see the picture below) where one should define the following settings:

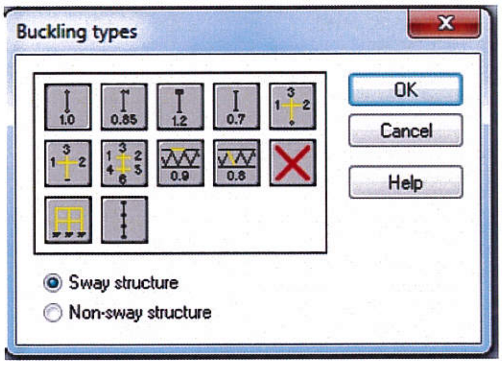
- type *Beam 1* in the *Member Type* field
- check out the *Coefficient* option in the *Buckling (Y-axis) Member Length l_y* and *Buckling (Z-axis) Member Length l_z* fields



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

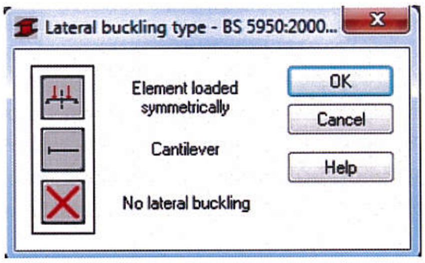
In the *Buckling length coefficient* field select the coefficient for directions **Y** and **Z**. To achieve this, one should click the 

icon, which results in the appearance of the dialog box (see the picture below) that allows one to select the buckling length coefficient.



By clicking the appropriate icon, one should select the coefficient 1.0 (first icon from the left in the first row). Once the selection is done, close the dialog box by clicking **OK** button. Then, perform the same operation for direction **Z**.
Define the parameters of lateral buckling type in the ***Lateral Buckling Parameters*** dialog box:

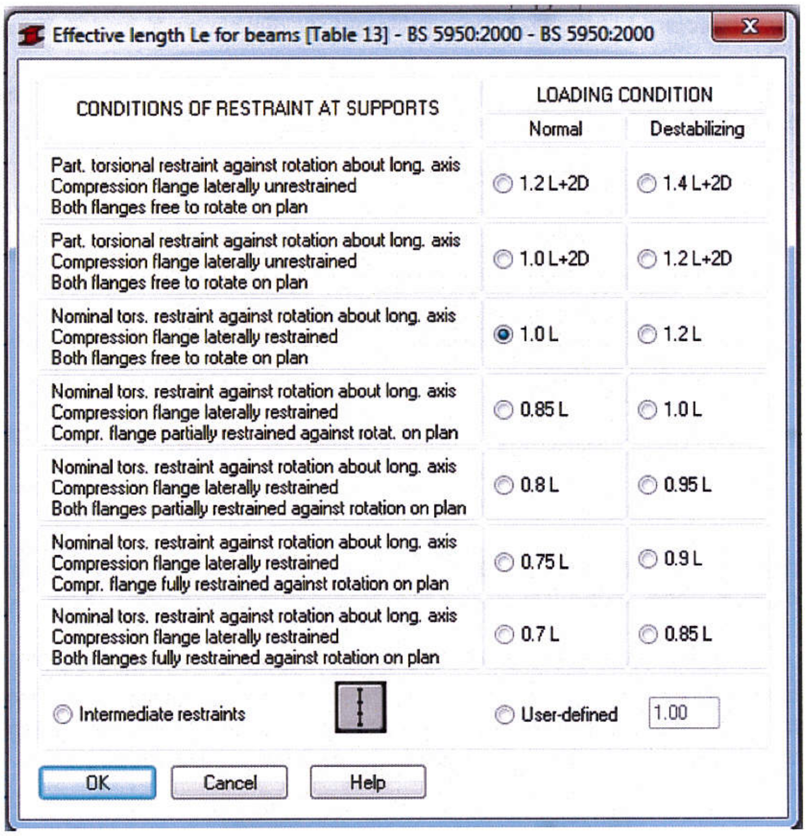
- Define the lateral buckling type by clicking the icon *Lat. Buckling Type* within the *Lateral Buckling Parameters* field. It results in the appearance on the screen the dialog box (presented below) where one should select the *Element Loaded Symmetrically* option



Once the selection is done, close the dialog box by clicking **OK** button.

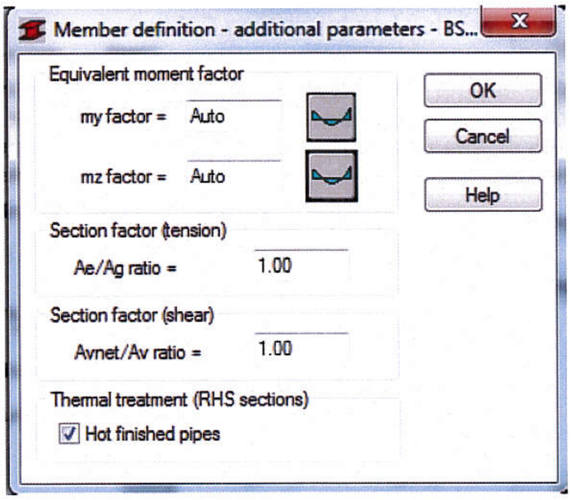
Define the effective length coefficient for beam by clicking the icon: *Upper Flange* (do the same with *Lower Flange*). Once this is done, there appears the dialog box where one should select the third option in the first column (see the picture below).

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Once the selection is done, close the dialog box by clicking **OK** button.
Once the above parameters are defined, one should select the **More** button in the ***Member Definition – Parameters*** dialog box in order to determine additional parameters. There appears the ***Member Definition – Additional Parameters*** dialog box where one should define the following settings:

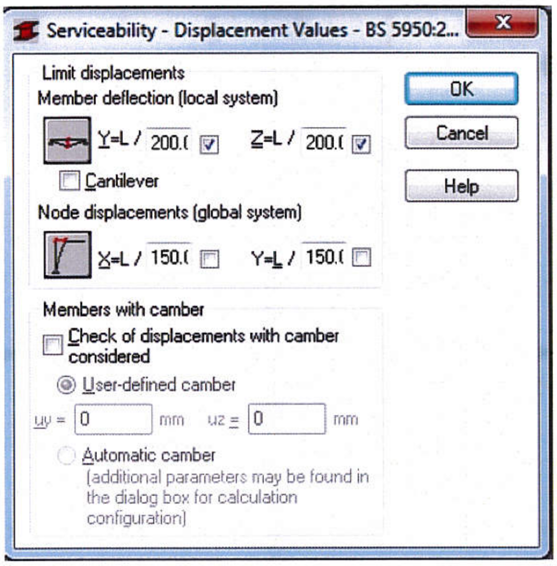
- In the *Calculation Parameters* field: leave the default values of parameters as shown in the picture below



- Define the *Ae/Ag r Ratio = 1,0* in the *Sections Parameters* field

Once the selection is done, close the dialog box by clicking **OK**. Press the **Service** button, which opens the ***Serviceability – Displacement Values*** dialog box (see the picture below). Check out the check boxes referring to the *Member deflection* options in the *Limit Displacements* field.

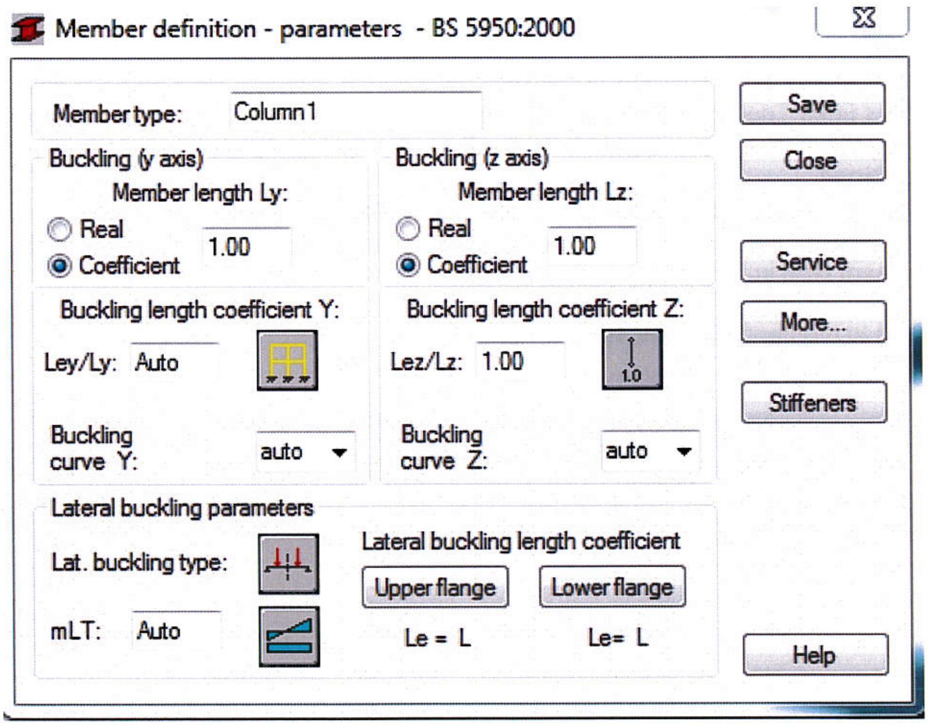
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Confirm the above settings by clicking **OK**. Save the above settings by clicking **Save** in the ***Member Definition – Parameters*** dialog box. Then, close the dialog box by clicking the **Close** button.
In the ***Member Type*** dialog box, there will appear a new type of element: "*Beam1*".

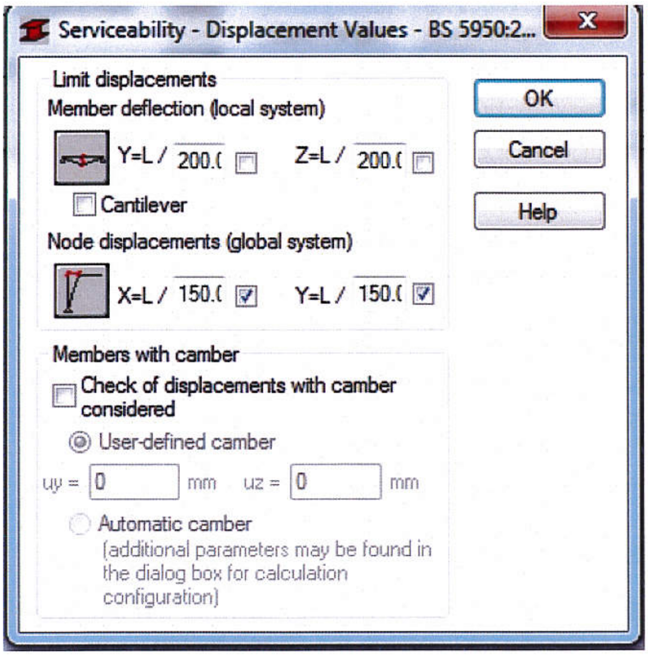
In exactly the same way we will define parameters for columns in "Column 1 " set of parameters

For buckling length we will use automatic procedure of buckling length calculation in Ly direction and 1 in Lz direction



Then for Service maximal node displacement will be defined

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



In the ***Member Type*** dialog box, there will appear a new type of element: "*Column 1*".

2.3. Assigning parameters to elements in a structure

Parameters may be assigned in:
- The ***Member Type*** dialog box: it allows one to indicate the defined element type and set the cursor in the *Lines/Bars* field. Then one should go to the graphical viewer and indicate the two structure beams, while the **Ctrl** button is pressed. Confirm by clicking **Apply** and close the dialog box by clicking **Close**.
- The table available from the main menu by means of the *View / Tables / Bars* command. Once the table is open, one should set the cursor in the appropriate field of the *Type* column and select the defined element type in the unfolded combo box.

"Column 1" should be assigned to elements 8 9 and 10
"Beam 1" should be assigned to elements 11 12

Definition tables

Member	Name	Components	Code group	Section	Type	Ly (m)	Lz (m)
1	RC Column_1	1	(N/A)	R300x300	RC Column	(N/A)	(N/A)
2	RC Column_2	2	(N/A)	R300x300	RC Column	(N/A)	(N/A)
3	RC Column_3	3	(N/A)	R300x300	RC Column	(N/A)	(N/A)
4	RC Beam_4	4	(N/A)	R300x600	RC Beam	(N/A)	(N/A)
5	RC Beam_5	5	(N/A)	R300x600	RC Beam	(N/A)	(N/A)
6	RC Beam_6	6	(N/A)	R300x600	RC Beam	(N/A)	(N/A)
7	RC Beam_7	7	(N/A)	R300x600	RC Beam	(N/A)	(N/A)
8	Column 1_8	8	(N/A)	4x254x89	Column 1	3,00	3,00
9	Column 1_9	9	(N/A)	4x254x89	Column 1	5,00	5,00
10	Column 1_10	10	(N/A)	4x254x89	Column 1	3,00	3,00
11	Beam1_11	11	(N/A)	6x178x54	Beam1	6,32	6,32
12	Beam1_12	12	(N/A)	6x178x54	Beam1	6,08	6,08

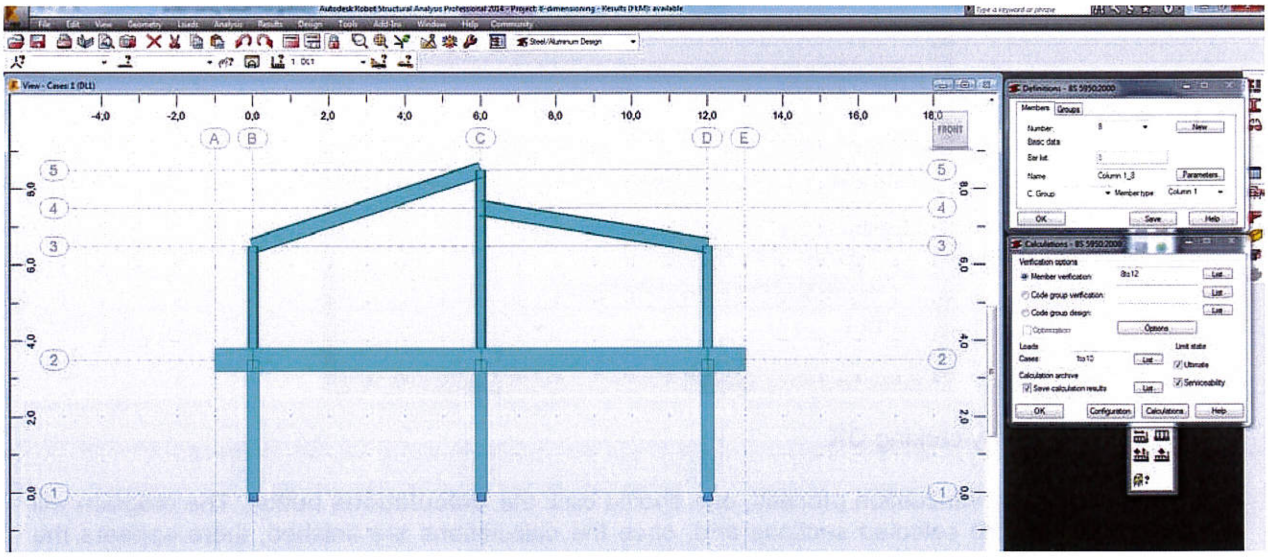
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

2.4. Structure calculations

In order to perform calculations of the structure, one should select the *Analysis / Calculations* command from the main menu or click the icon from the upper toolbar.

2.5. Entering the Steel Design layout

In order to enter the layout, one should unfold the combo-box in the top right-hand corner and select the *Structure Design / Steel /Aluminum Design* option. The screen will be divided into three parts: the *View* graphic viewer, the ***Definitions*** and ***Calculations*** dialog box (see the picture below).



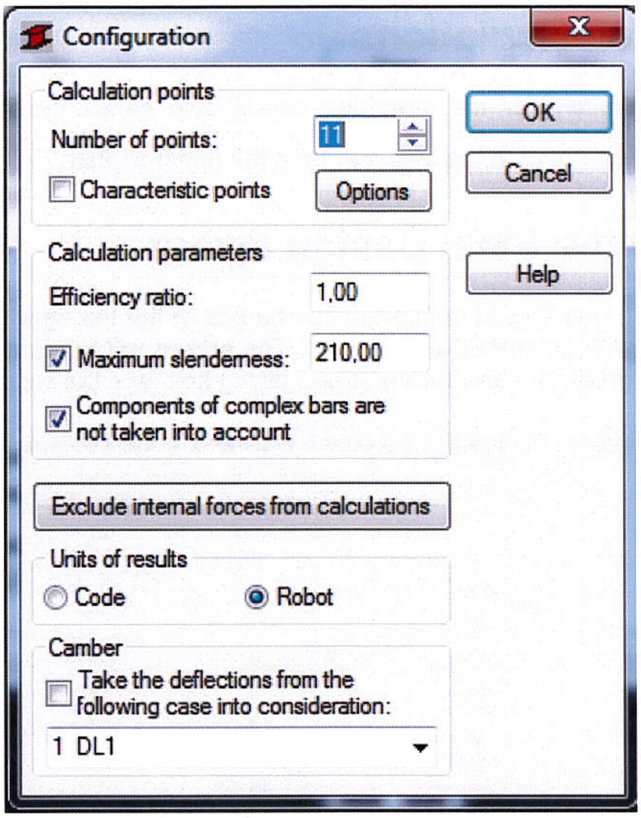
2.6. Member verification

Check out the *Member Verification* option in the ***Calculations*** dialog box. In order to select members, one should click the *List* relevant for a given option and, once the ***Member Selection*** dialog box is open, select all the bars by clicking the **All** button and then close the dialog box by clicking **Close**.
Click the **Load Case Selection** button in the *Loads* field and in the dialog box with the same name select all load cases by clicking the **All** button. Once the selection is done, close the dialog box by clicking **Close**.
Analysis will be carried out with respect to *ULS* (internal forces) and *SLS* (deflections). Therefore, one should select the *Ultimate* and *Service* options in the *Limit State* field.
In order to define the verification parameters, one should click the **Configuration** button. This results in opening the ***Configuration*** dialog box (see the picture below) that allows one to define the following calculation parameters:

- *Points Number* set for 11,
- *Efficiency Ratio* set for 1.0,
- *Maximum Slenderness* set for 210,

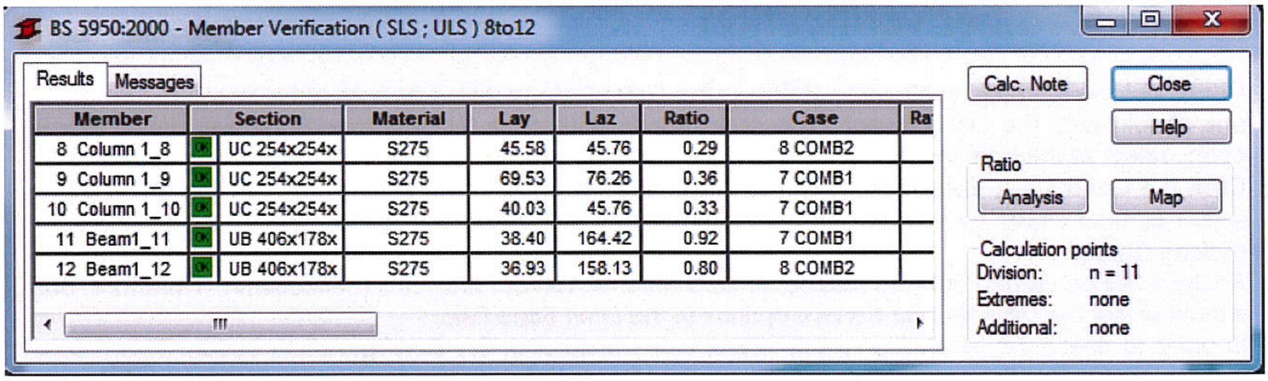
Leave the default values of all the parameters.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Close the dialog box by clicking **OK**.

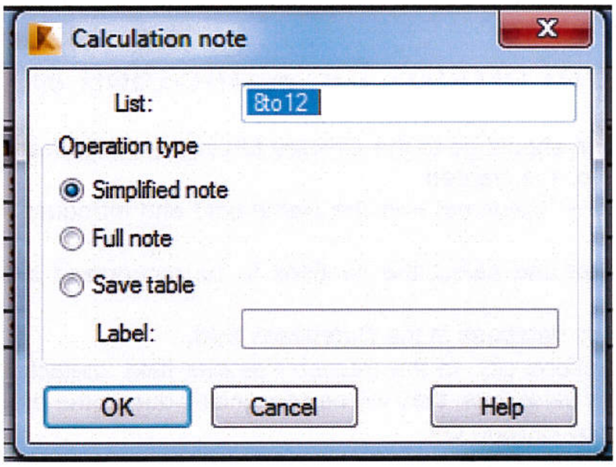
In order to launch the verification process, one should click the **Calculations** button. The program will automatically verify the selected sections and, once the calculations are finished, there appears the dialog box presenting the verified members (see below).



One may carry out editing of the results:

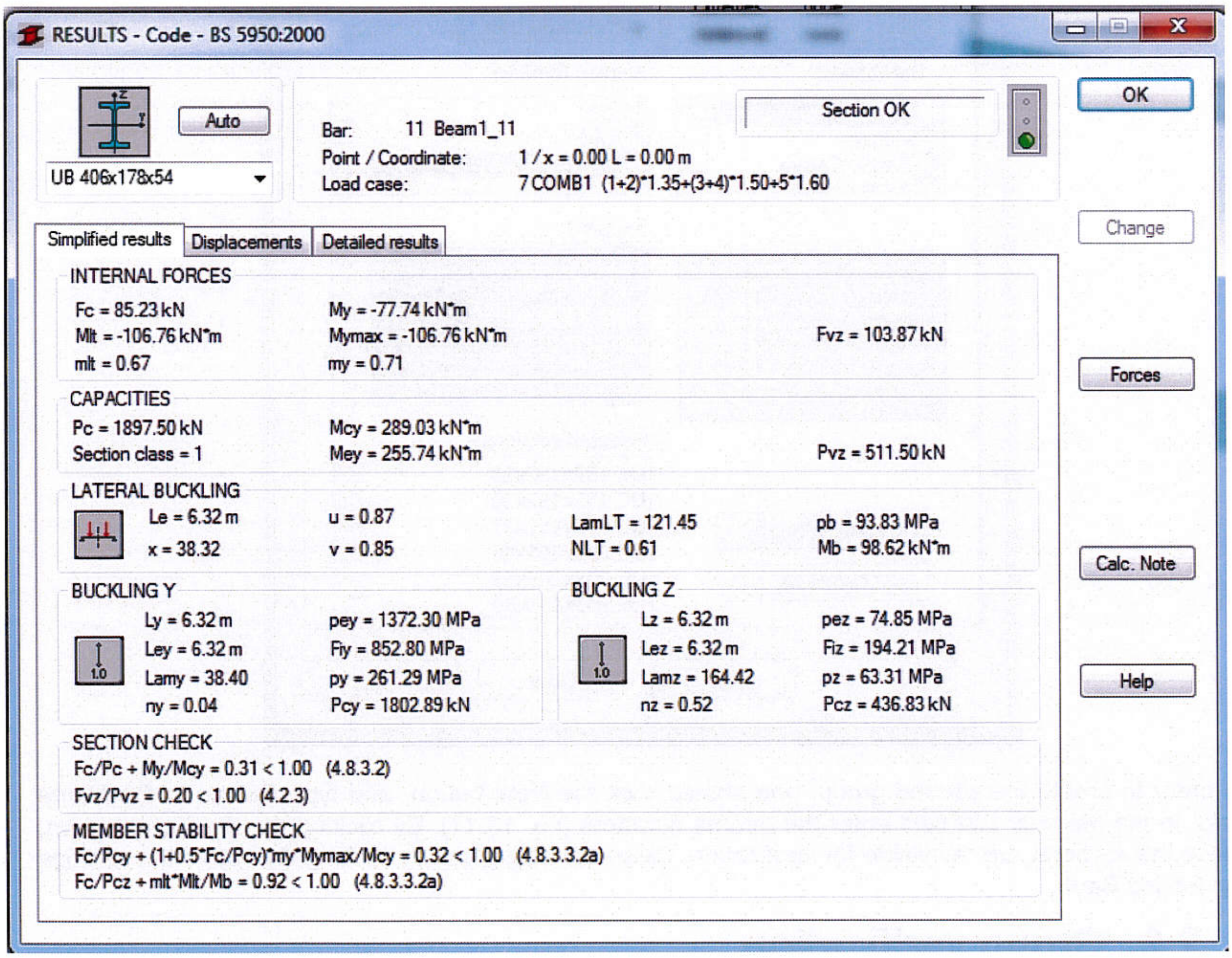
- For all the members simultaneously – by means of the *Calc. Note* option. There appears the ***Printout*** dialog box (see the picture below). Select here the *Table Printout* option. Thus, we obtain a table with all the verified elements.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



It is also possible to save all the detailed data to a file. This can be done by means of the *Full printout* option.
By means of the *Table screen capture* option, one may obtain a screen capture of the table of results. In this option, one has to define additionally the *Label* name.

– For single members – by clicking the row with the calculated member in the ***Member Verification*** dialog box. There will appear the ***Results*** dialog box presenting detailed results (see the picture below).



One may obtain detailed information concerning a given section on the tabs: *Simplified results* and *Detailed results* (and the *Displacements* tab in the case of beams where displacement is also calculated).

Save file as 8-verification

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

2.7. Definition of groups for verification and optimization

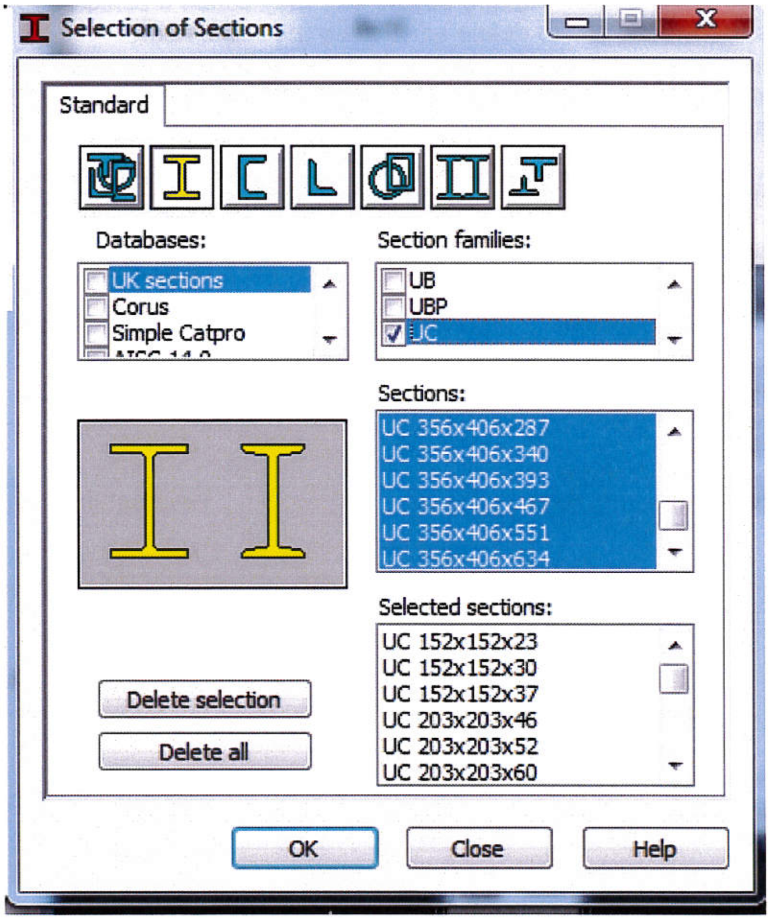
In order to define a group, one should go to the ***Groups*** tab in the ***Definitions*** dialog box and click the **New** button. Thus, a group No. 1 is created.
Type the name of the group – "*Columns*" – in the *Name* field and introduce members 8to10 into the *Member List* field.

Click the **Sections** button and select the sections to be recognized during the verification and design processes:

- Select the *UK sections* database in the *Databases* field,
- Select in turn the sections *UC* in the *Section Families* field, (selected sections are loaded to the *Selected Sections* field; thus, they will be recognized during the process of verification),
- Close the dialog box by clicking **OK**.

In the material field select the *S275* steel type

Save all the settings in the ***Definitions*** dialog box by clicking **Save**.



In order to create the second group, one should click the **New** button, and type *Beams* in the *Name* field. In the *Member List* field enter the beams numbers (i.e. 10,11). By clicking the **Sections** button, make the sections: *UB* available for verification. Close the dialog box by clicking **OK**. Save the settings by clicking **Save**.

2.8. Group verification

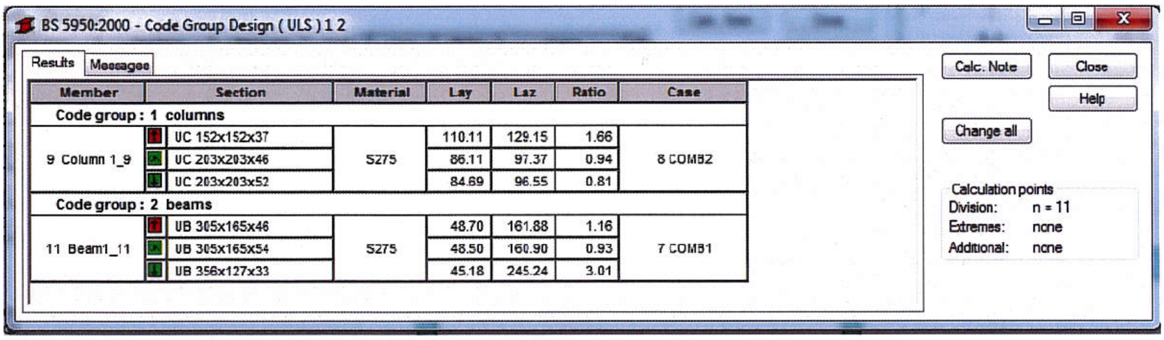
In order to verify the created groups, one should switch to the ***Calculations*** dialog box and select the *Code Group Verification* option. Group verification is carried out only for the *ULS*. Then, one should define numbers of groups that will be verified – in this case 1 and 2 and all load cases. Run *Calculations*. Group verification is then carried out.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

2.9. Group design

Indicate the *Code Group Design* option in the **Calculations** dialog box and type group numbers (No. 1 and 2). Click the **Calculations** button. In the ***Code Group Design*** dialog box, there appear the sections selected for each group, formerly made available for verification during the group definition. Once the results analysis is done, close the dialog box.

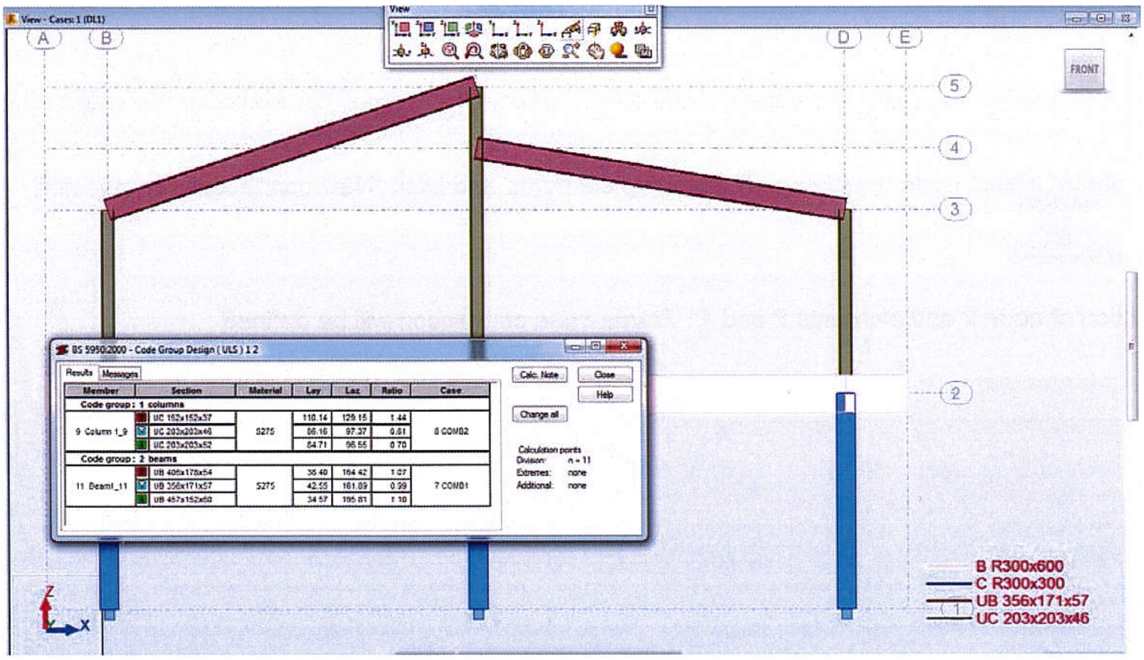
In order to carry out optimization, one should click the *Optimization* option and then press the **Optimization** button, which opens the ***Optimization Options*** dialog box. This dialog box is used to determine the parameters of calculations performed on member groups with the optimization options taken into account. From the available options select the *Weight* option. Switching this option on results in recognizing section weight in optimization, i.e. the program will look for the lightest section in the group from among the sections that meet the code-defined criteria. Confirm the operation by clicking the **OK** button.



Member	Section	Material	Lay	Laz	Ratio	Case
Code group : 1 columns						
9 Column 1_9	UC 152x152x37	S275	110.11	129.15	1.66	8 COMB2
	UC 203x203x46		86.11	97.37	0.94	
	UC 203x203x52		84.69	96.55	0.81	
Code group : 2 beams						
11 Beam1_11	UB 305x165x46	S275	48.70	161.88	1.16	7 COMB1
	UB 305x165x54		48.50	160.90	0.93	
	UB 356x127x33		45.18	245.24	3.01	

Launch calculations. The program will carry out the design calculations with section optimization. From a set of sections, the program selects the most optimal one with respect to weight. Clicking the *Change All* option results in exchanging the section applied during the design process for the optimized sections.

Close the design dialog box and run the *View / Tables / Bars* command from the main menu. Check the correctness of the operation of exchanging sections after optimization.



Save the structure under the name 9-**optimized**.

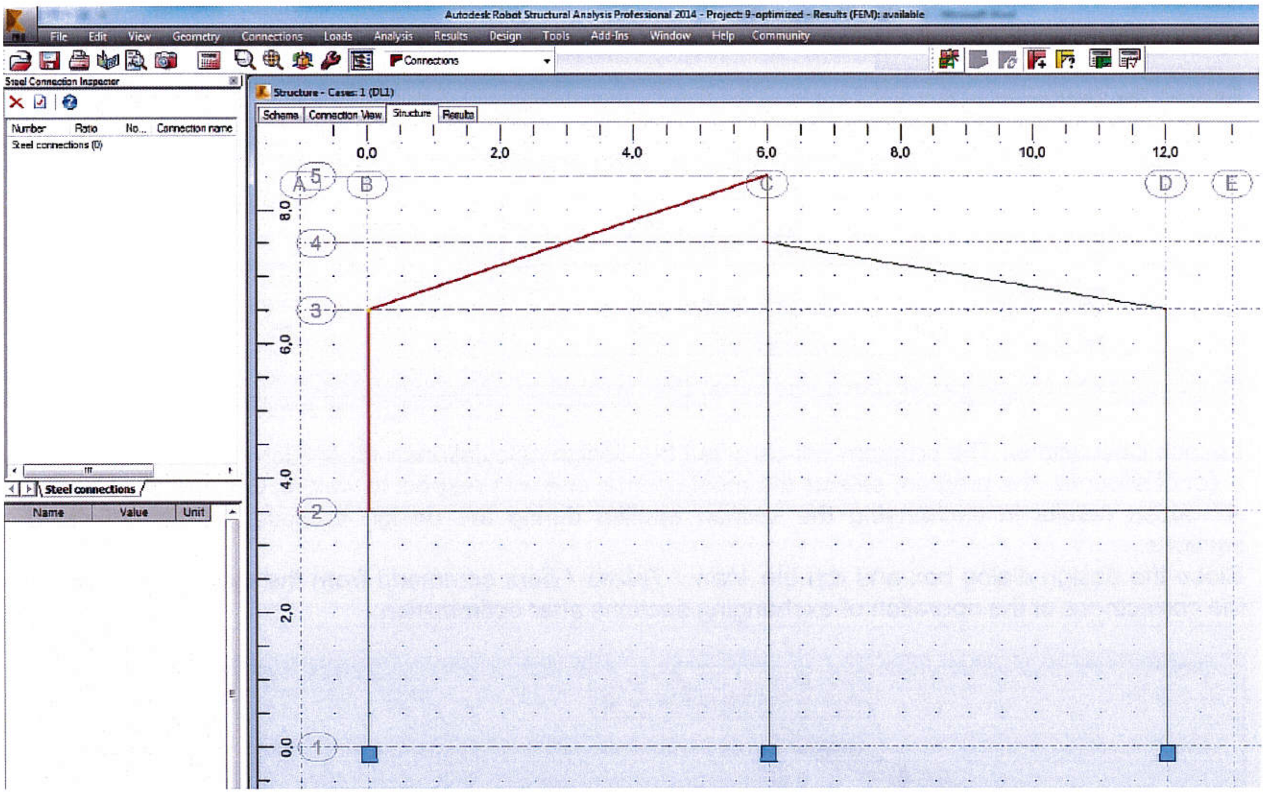
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

3. *EXERCISE NO. 3 "DESIGN OF STEEL CONNECTIONS"*

3.1. Open the 9-optimized file

3.2. Definition and verification of steel connection

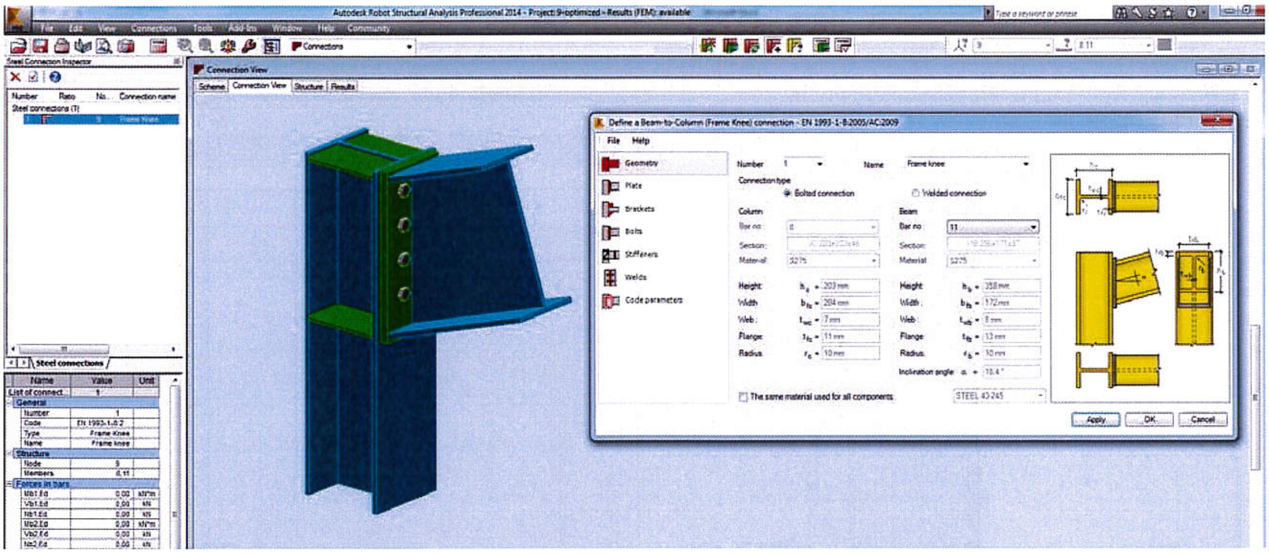
In order to define steel connection in model, one open the *Design / Steel connections design* command from the main menu. Once the option is called Steel connection layout will be opened.



Then one should select node together with adjoining elements and click "New connection for selected bars" option 

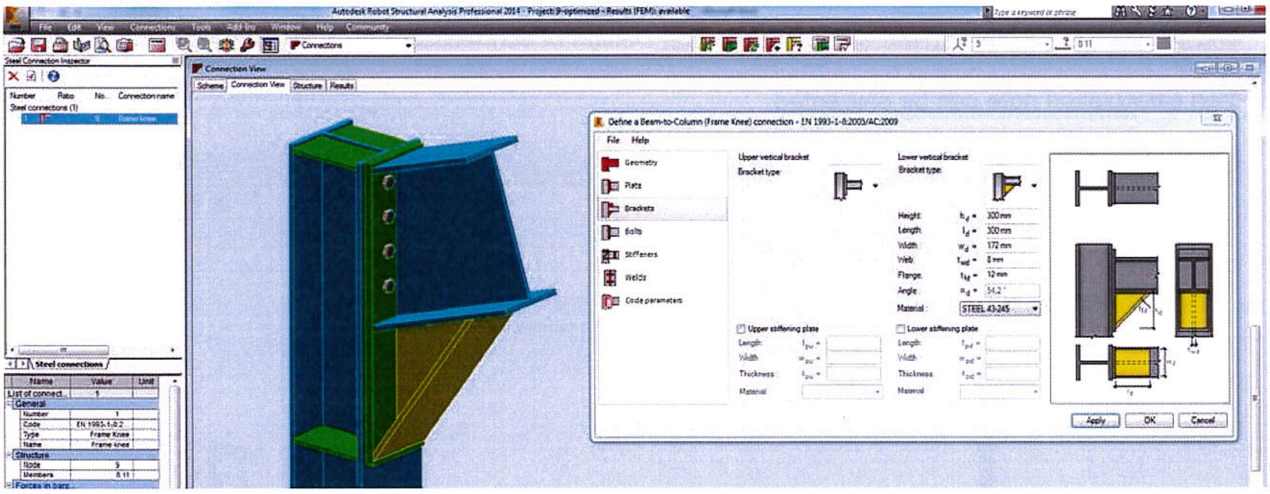
After selection of node 9 and elements 8 and 11 Frame knee connection will be defined

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

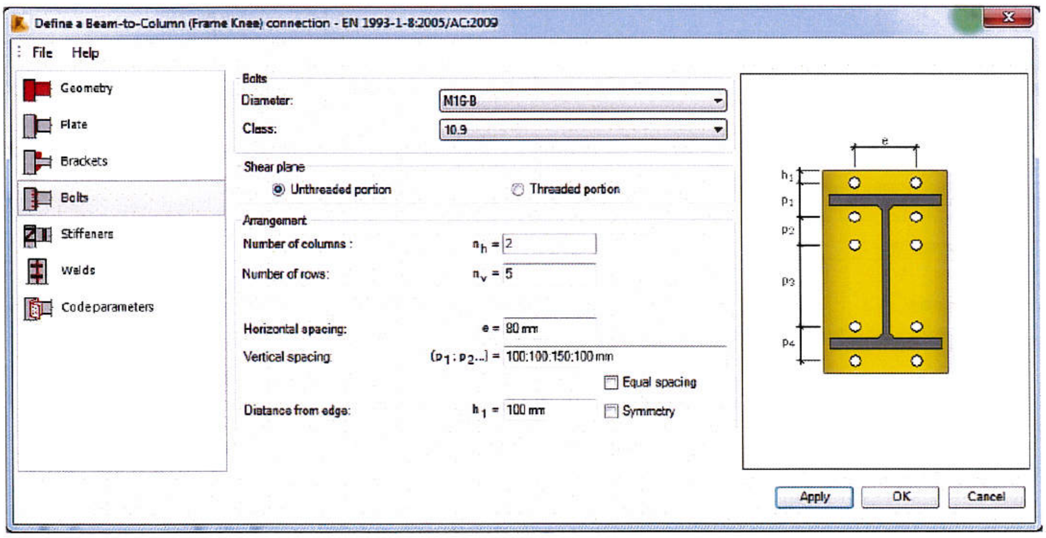


Steel connection module does not optimize connection. User can modify connection parameters defining bolts distibution, additional brackets and stiffeners then verify connection

We will define additional lower bracket (length and height 300 mm)

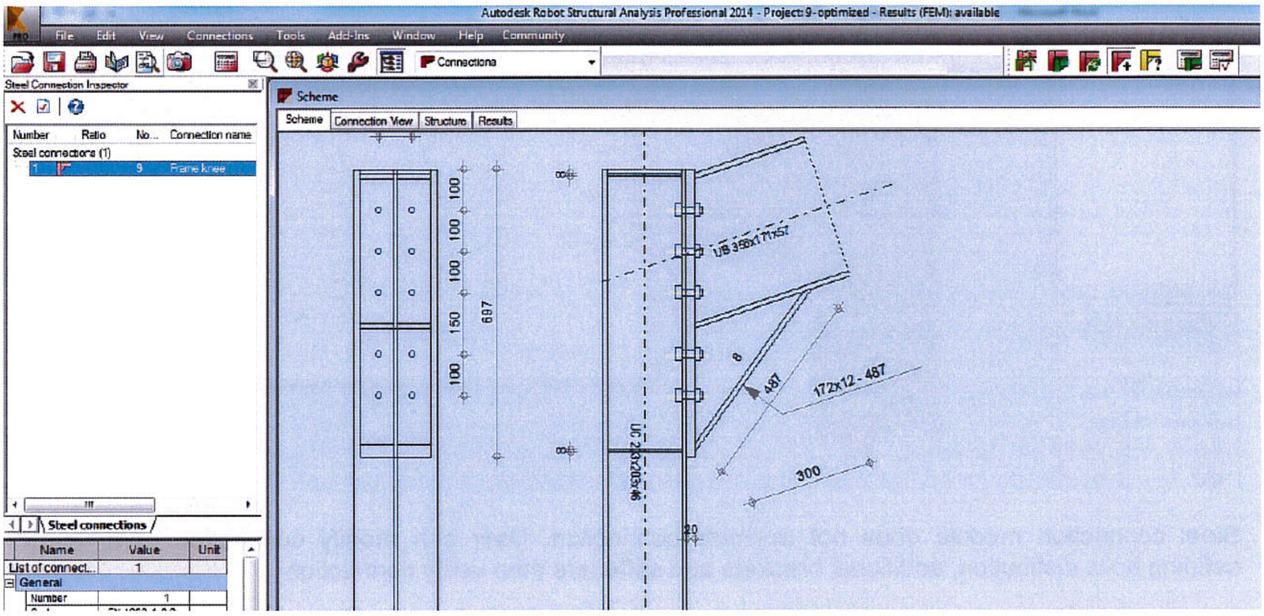


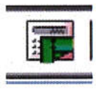
Then modify bolts distribution



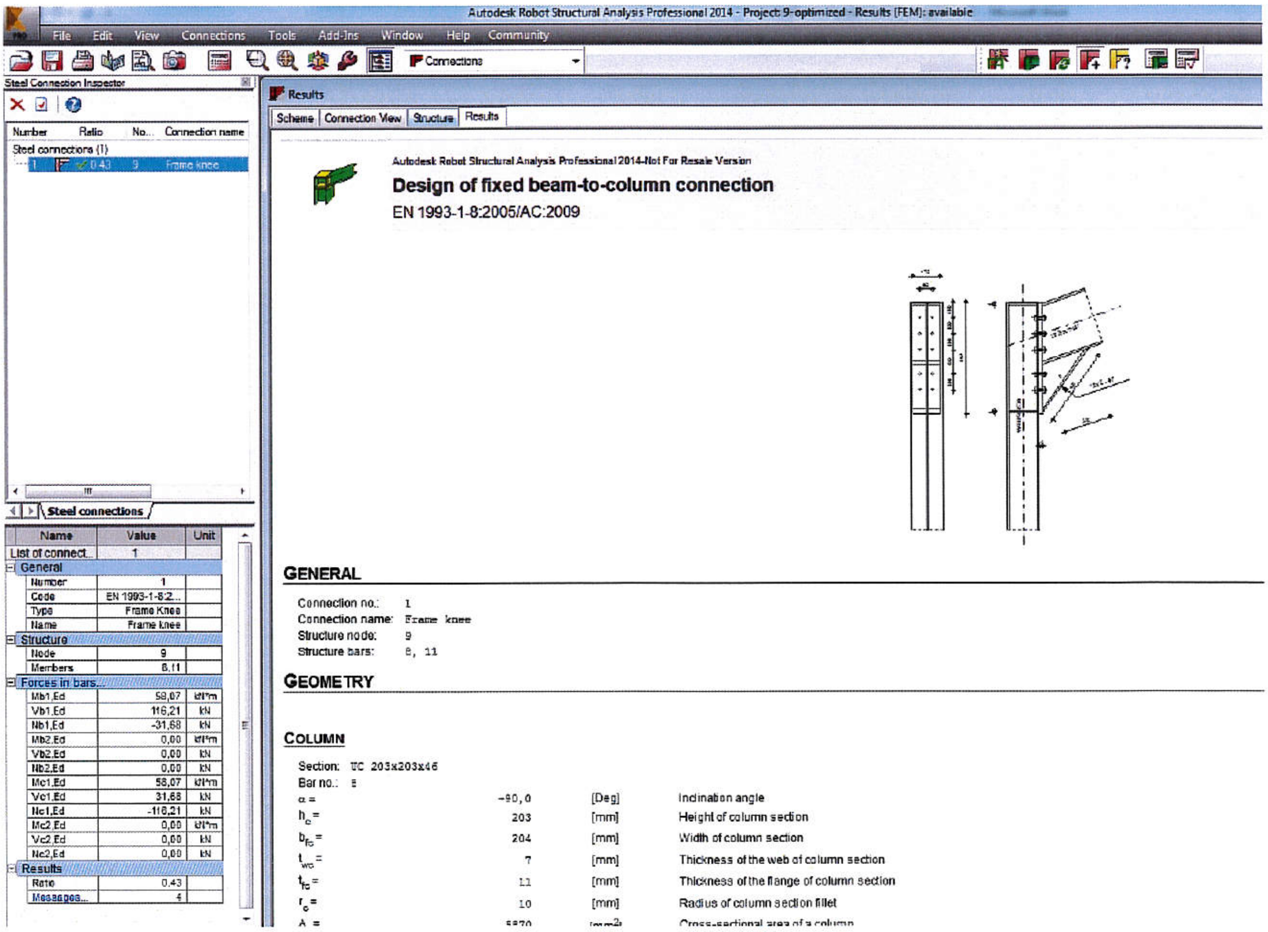
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Then after Apply the connection seme can be presented (Scheme tab)



and connection can be verified with use op option Design of connection from a structure 

then calcculation note can be presented



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Another connection can be defined by switching to Structure tab selecting connection node and adjoining elements and clicking “New connection for selected bars” option 

Save file as 10-connections

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

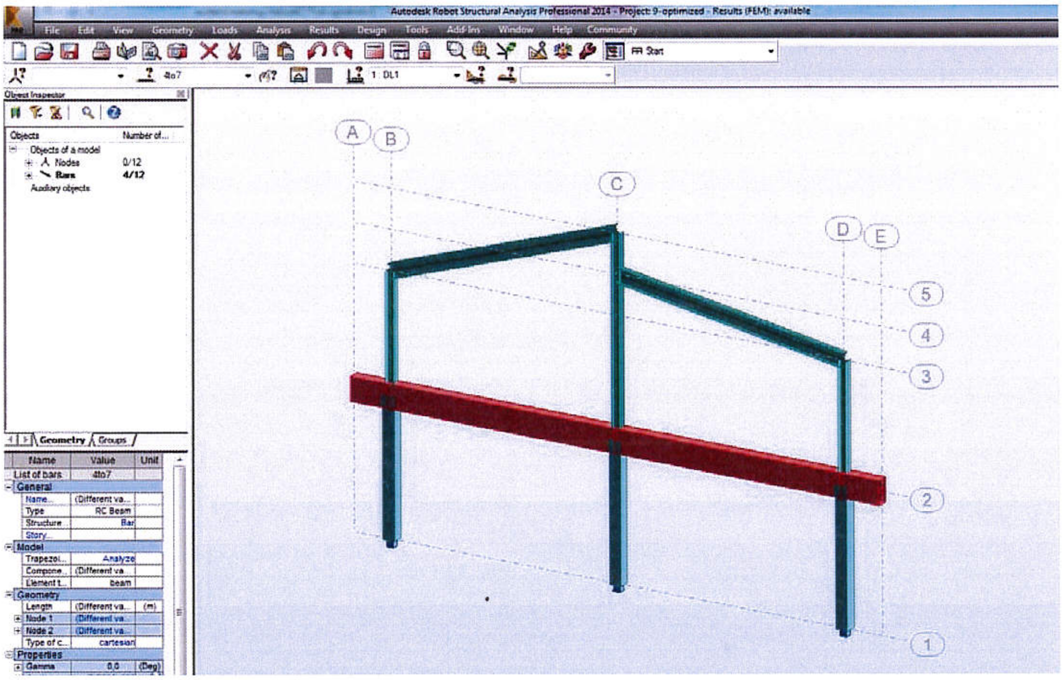
4. EXERCISE NO. 4 CONCRETE ELEMENTS DIMENSIONING (BEAMS, COLUMNS, SPREAD FOOTINGS)

4.1. Open the 9-optimized file

4.2. Provided reinforcement

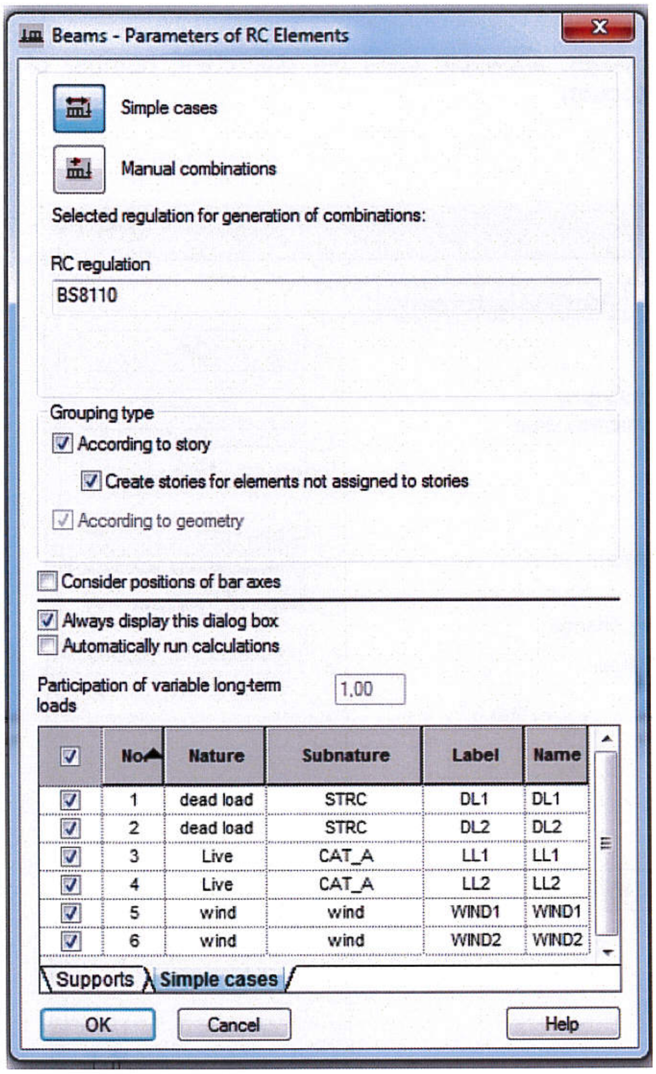
Since version 2013 different types of concrete elements can be transferred to Robot provided reinforcement module in one step. If user select slab and its supported nodes program will try to create slab and spread footing for all supported nodes of the slab in provided reinforcement module.

Selected elements will be transferred to provided reinforcement module

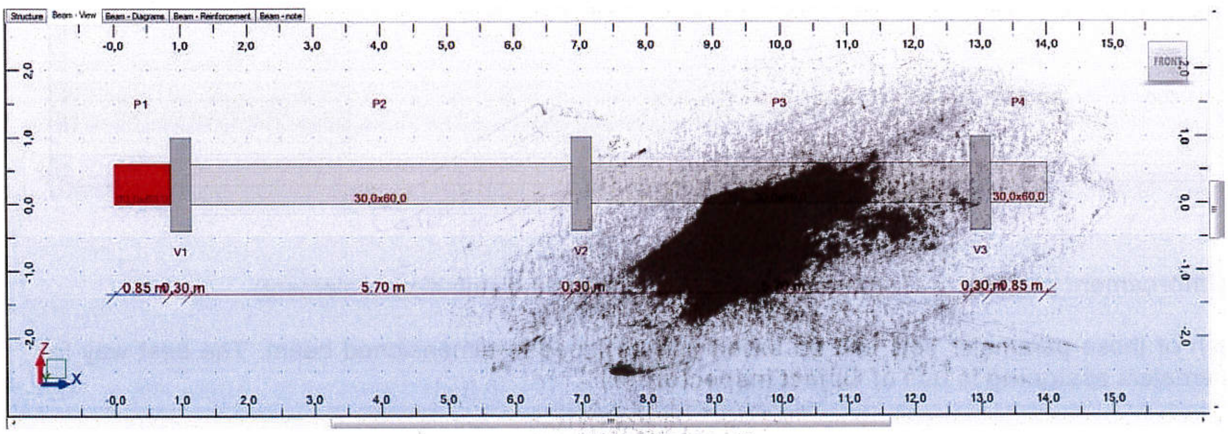


While information about elements are transferred program asks about loads transfer way (simple cases or manual combinations). Simple cases mode uses default code combination regulation for combination definition so user defined coefficients are not taken into account. In our model one should use manual combination option.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



While beam and column with selected loads are transferred to provided reinforcement module they can be selected for dimensioning in Object Inspector



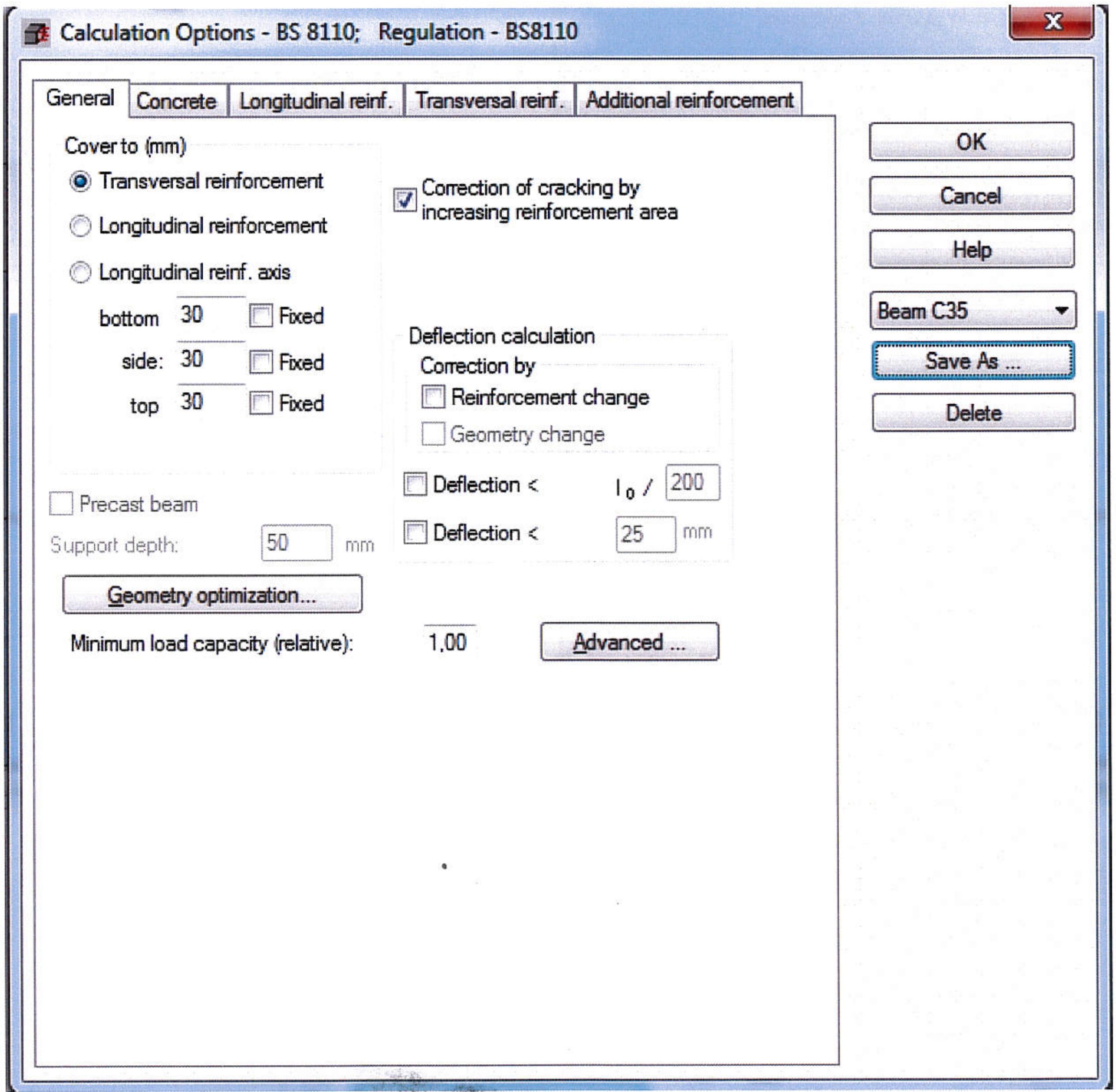
4.2.1. Concrete beam

For concrete beam the following parameters have to be defined:

Story parameters (fire resistance, admissible cracking);

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Calculation option (material parameters, cover, allowable steel bar diameters, number of calculation points, torsion, axial forces taken into account)



Reinforcement pattern (parameters for reinforcement distribution in element)

Each of those parameter sets can be saved and assigned to dimensioned beam. The best way for parameters **assigning is use of Object inspector**

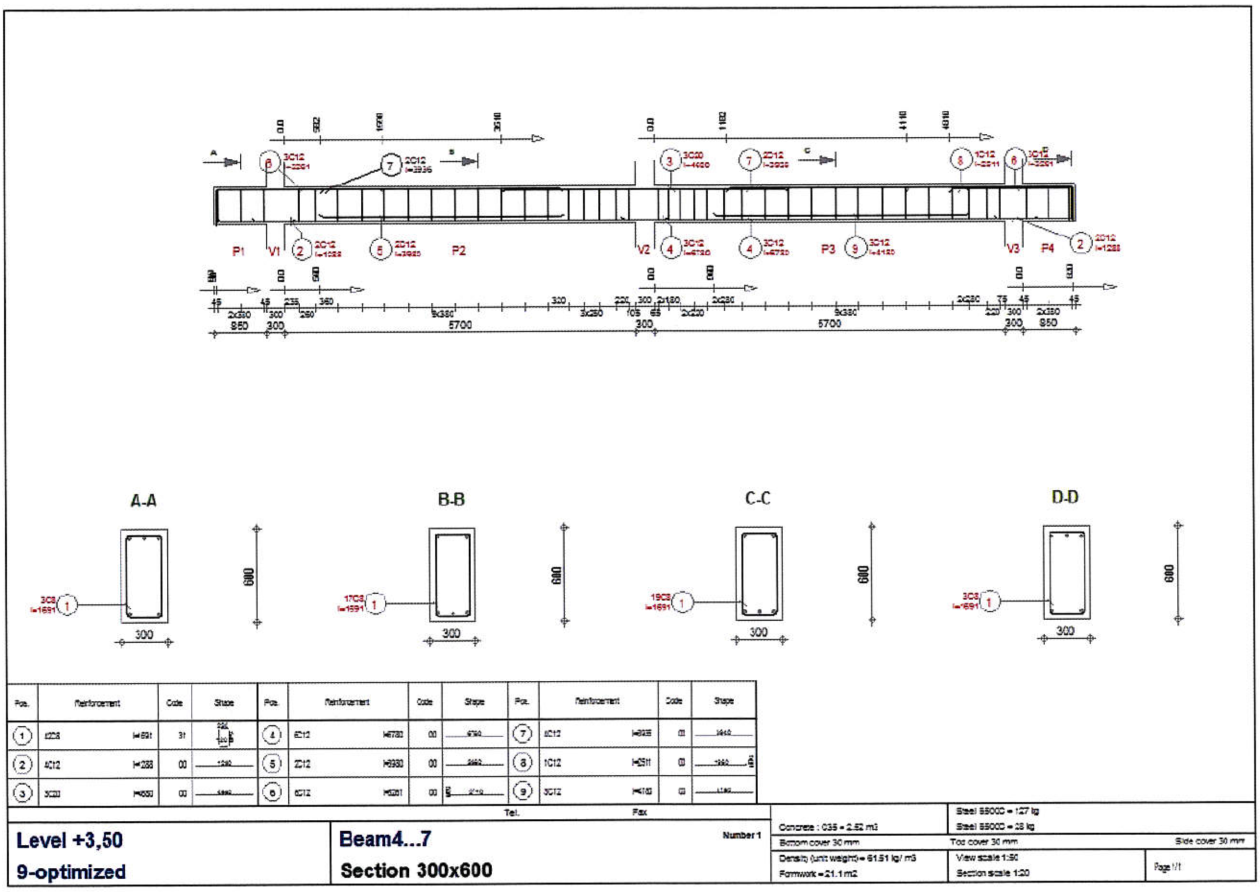
For this particular beam one can set

In **Calculation option** – concrete C35, steel B500C, longitudal reinforcement diameters 12, 16, 20, transversal reinforcement 8. It can be saved as "beam c35" and assigned to beam in **Object inspector**

In **Reinforcement pattern-** Assembly reinforcement considered in load capacity. It can be saved as "Pattern C35" and assigned to beam in **Object Inspector**

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Then reinforcement can be calculated. Results are available in text file **Results / Calculation note**
Reinforcement drawing can be obtained after choosing beam drawing template **Analysis / Drawing parameters** and then **Results / Drawings**



While beam drawing is edited one can choose generate sections position and change drawing scale.

Drawings generated for analyzed beam can be opened in **Object Inspector**

The following operations can be done for calculated beam

RC element / RC element update – when after beam is transferred to dimensioning module whole structure was updated and calculated once again. New internal forces in the beam can be then imported. In **Object inspector** one can select beam and from context menu RC element update can be launched
Manual reinforcement modification. In **Beam reinforcement** tab reinforcement can be presented in 3D then bar layout, shape, dimensions can be modified. For reinforcement edition one can use options Reinforcement / Translation and Reinforcement / Modification. Reinforcement table can be used for modification as well. Once reinforcement is updated user can verify new reinforcement layout Analysis / Verification
Stirrups spacing modification Reinforcement / Stirrup spacing
Beam section dimension modification – beam view tab
Modified beam section export to model Results / Structure update. Structure should be recalculated and then RC element update option should be used.

4.2.2. Concrete column

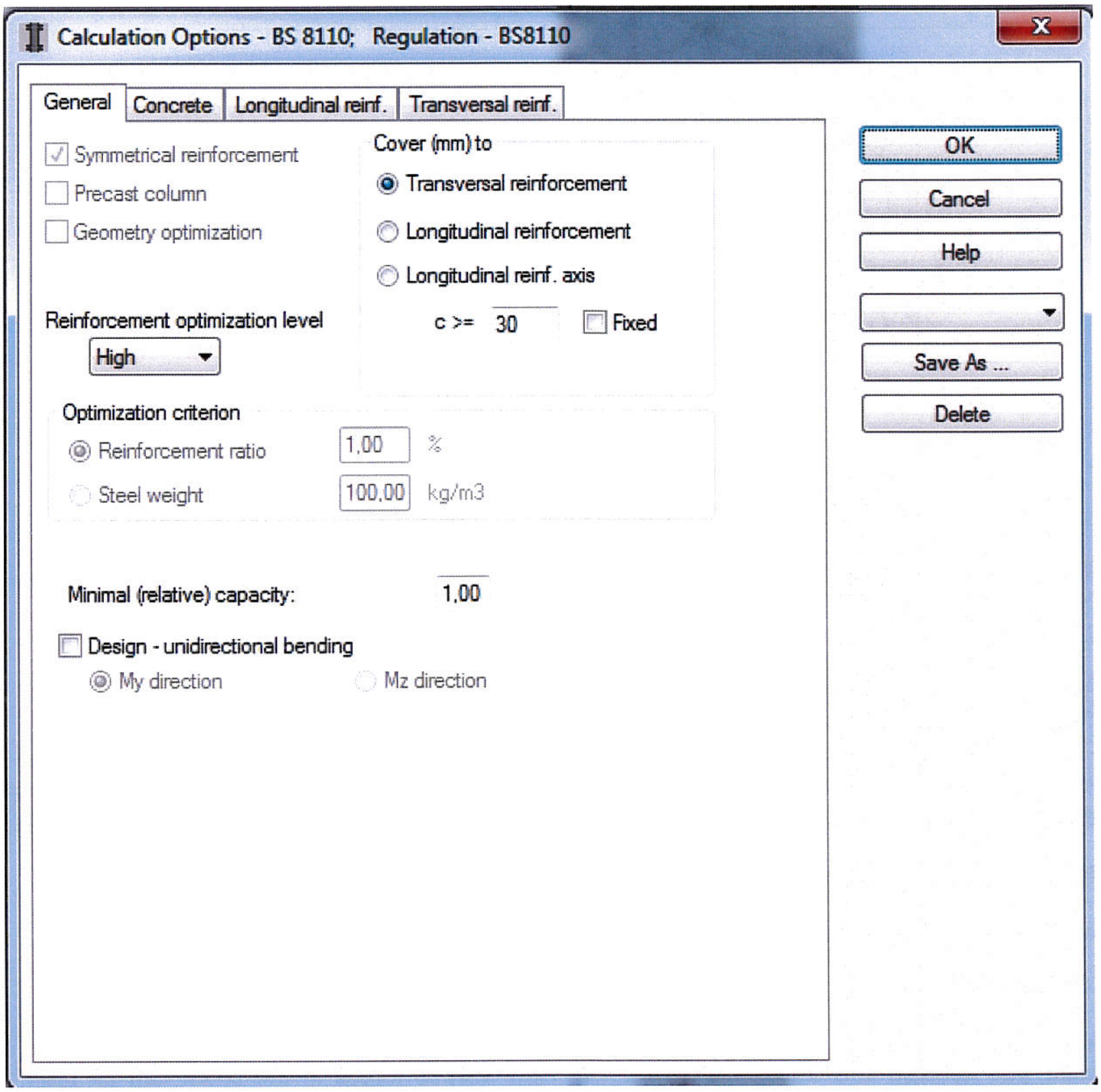
For concrete column parameters necessary for reinforcement calculation are quite similar to those defined for beam. Calculations will be made for column no 1

1 2/3

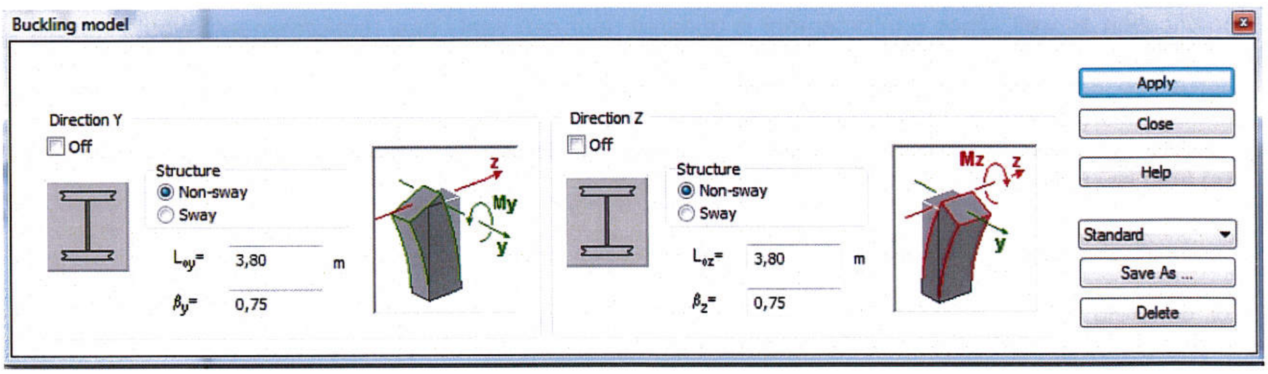
Story parameters _ (fire resistance);

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Calculation options (material parameters, cover, allowable steel bar diameters,)



Reinforcement pattern (reinforcement distribution in column)
Buckling length (buckling length definition)



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Each of those parameter sets can be saved and assigned to dimensioned column. The best way for parameters assigning is use of **Object inspector**

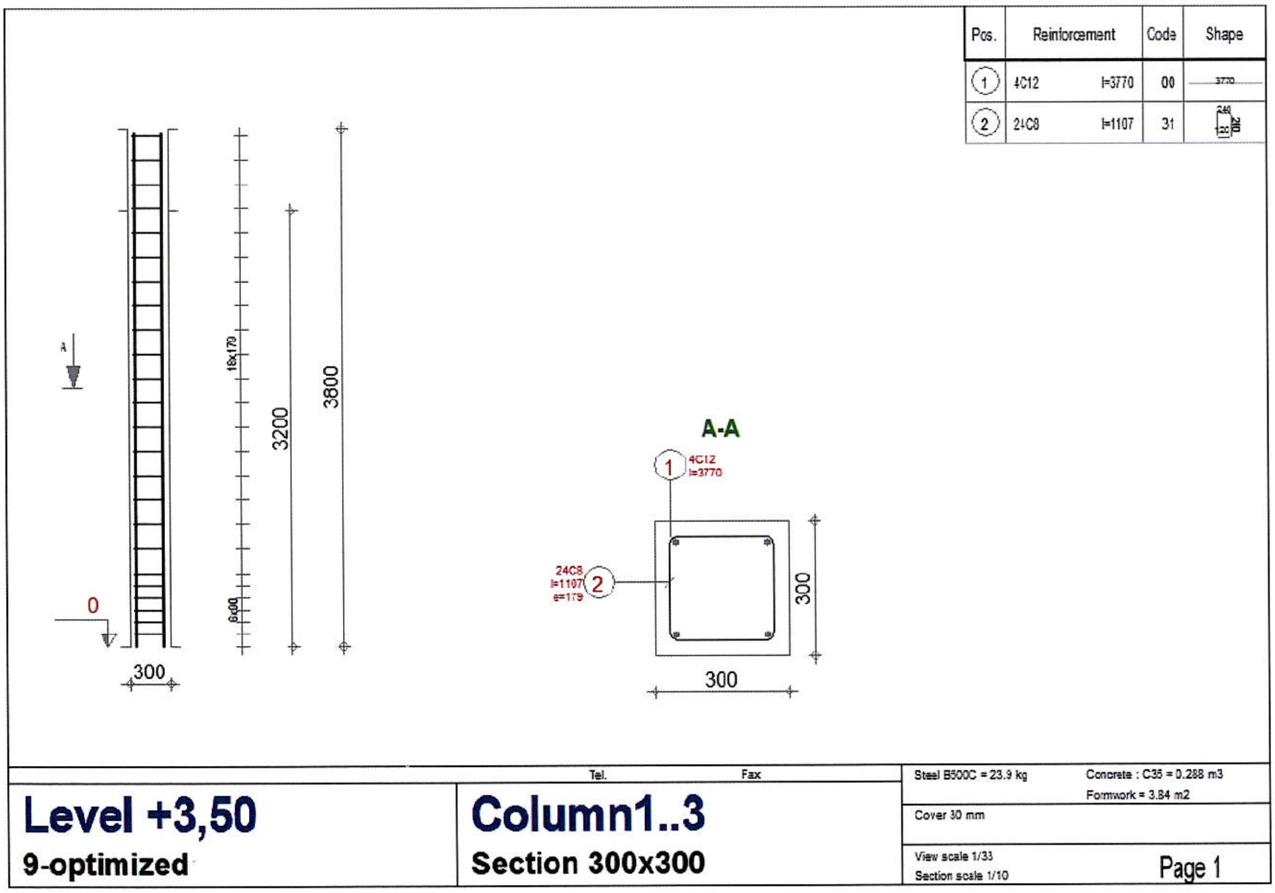
For this particular column one can set

In **Calculation option** – concrete C35, steel B500C, longitudal reinforcement diameters 12, 16, 20, transversal reinforcement 8. It can be saved as "column c35" and assigned to column in **Object inspector**

In **Reinforcement pattern**- standard set of parameters will be defined. It can be saved as "Clpattern C35" and assigned to column in **Object Inspector**

In **Buckling length** parameters for both directions buckling coefficient will be set to 0.7. It can be saved as "Buckling column" and assign to column in **Object Inspector**

Then reinforcement can be calculated. Results are available in text file **Results / Calculation note**
Reinforcement drawing can be obtained after choosing column drawing template **Analysis / Drawing parameters** and then **Results / Drawings**



Drawings generated for analyzed column can be opened in **Object Inspector**

The following operations can be done for calculated column

RC element / RC element update – when after column is transferred to dimensioning module whole structure was updated and calculated once again. New internal forces in the column can be imported. In **Object inspector** one can select column and from context menu RC element update can be launched
Manual reinforcement modification. In **Column reinforcement** tab reinforcement can be presented in 3D then bar layout, shape, dimensions can be modified. For reinforcement edition one can use options **Reinforcement / Translation** and **Reinforcement / Modification**. Reinforcement table can be used

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

for modification as well. Once reinforcement is updated user can verify new reinforcement layout
Analysis / Verification
Stirrups spacing modification Reinforcement / Stirrup spacing

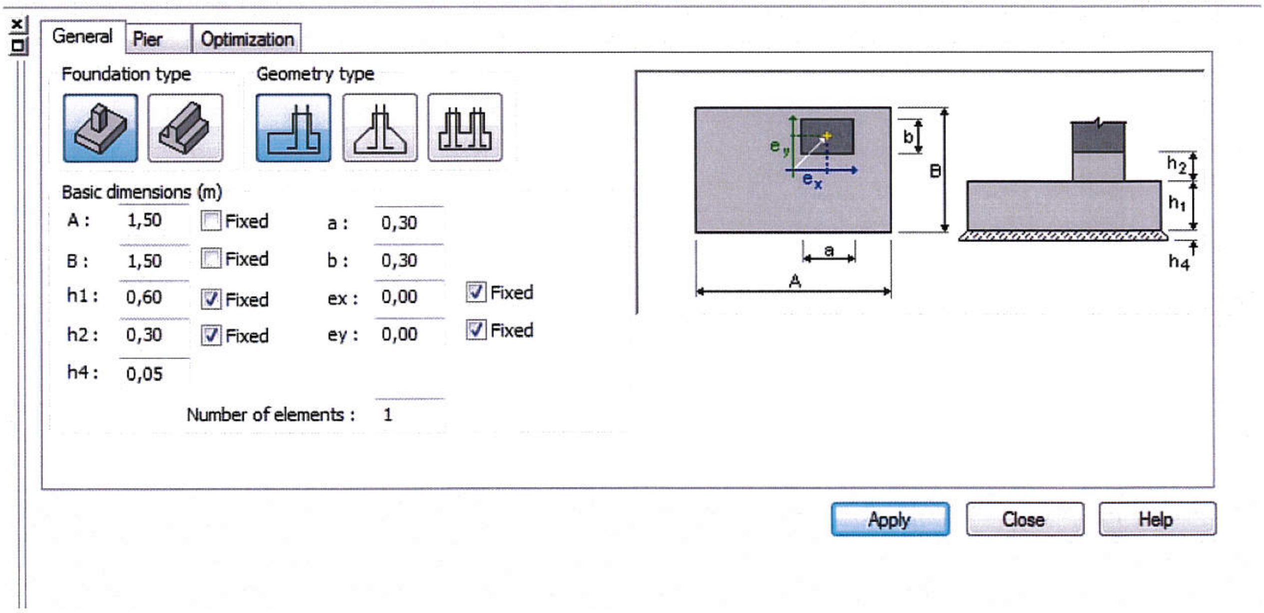
4.2.3. Spread footing

One or several supported nodes should be selected then Design / Provided reinforcement of RC elements should be chosen. Calculations will be done for supports no 1 and 3

While information about reactions is transferred program asks about loads transfer way (simple cases or manual combinations). Simple cases mode uses default code combination regulation for combination definition so user defined coefficients are not taken into account. In our model one should use manual combination option.

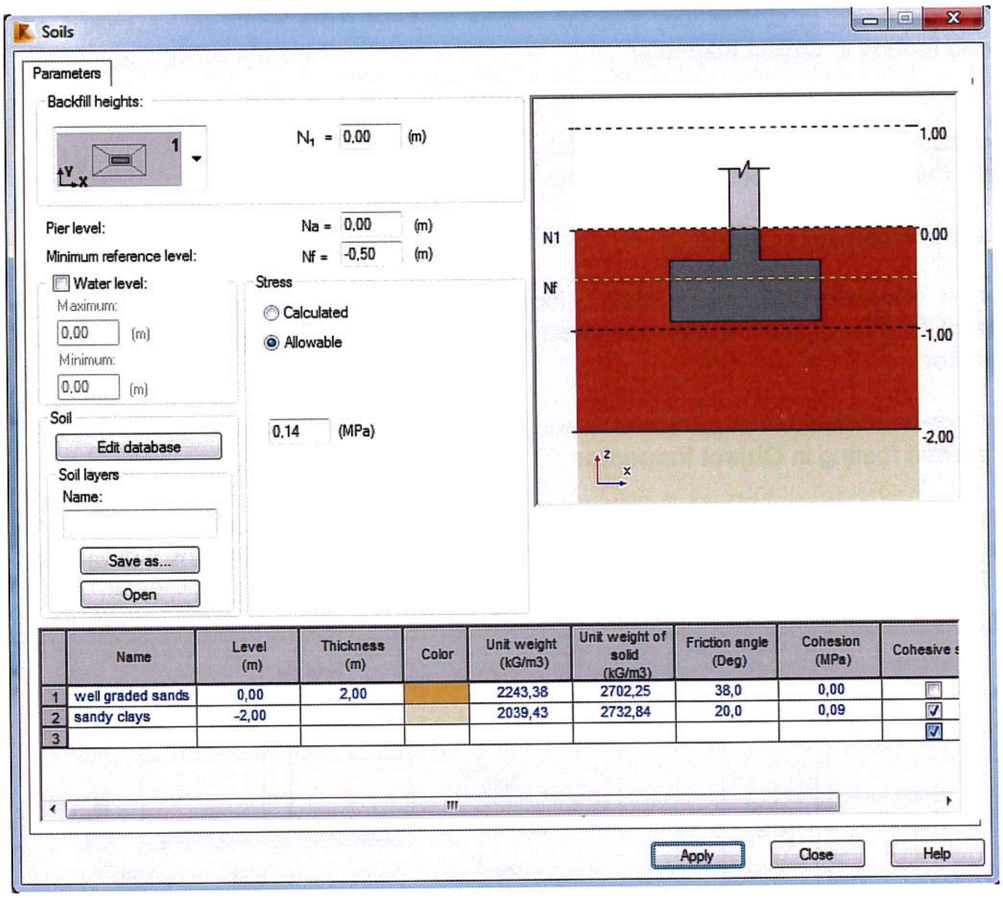
For spread footing except parameters similar for beams and columns additional information for dimensioning is necessary

Spread footing diameters for optimization (for this spread footing one can block thickness, ex and ey).

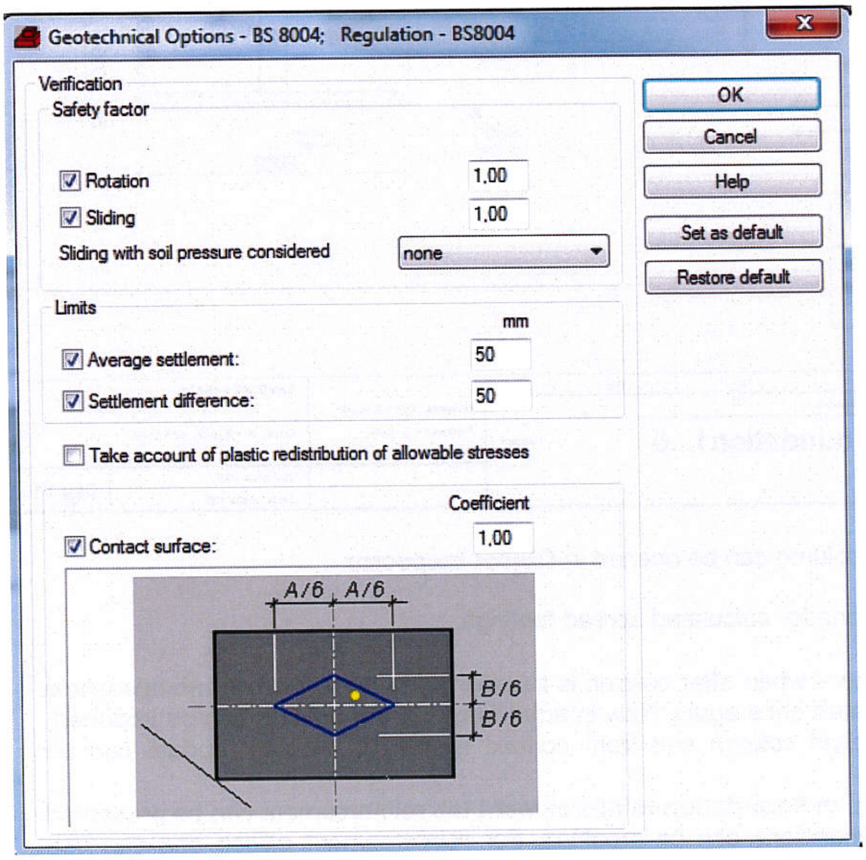


RC element / soil – soil layers together with water level. Soil can be chosen from soil database (can be updated by user). Once defined soil layers can be saved in separate file. Backfill height and its loads can be defined as well

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Analysis / Geotechnical options



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

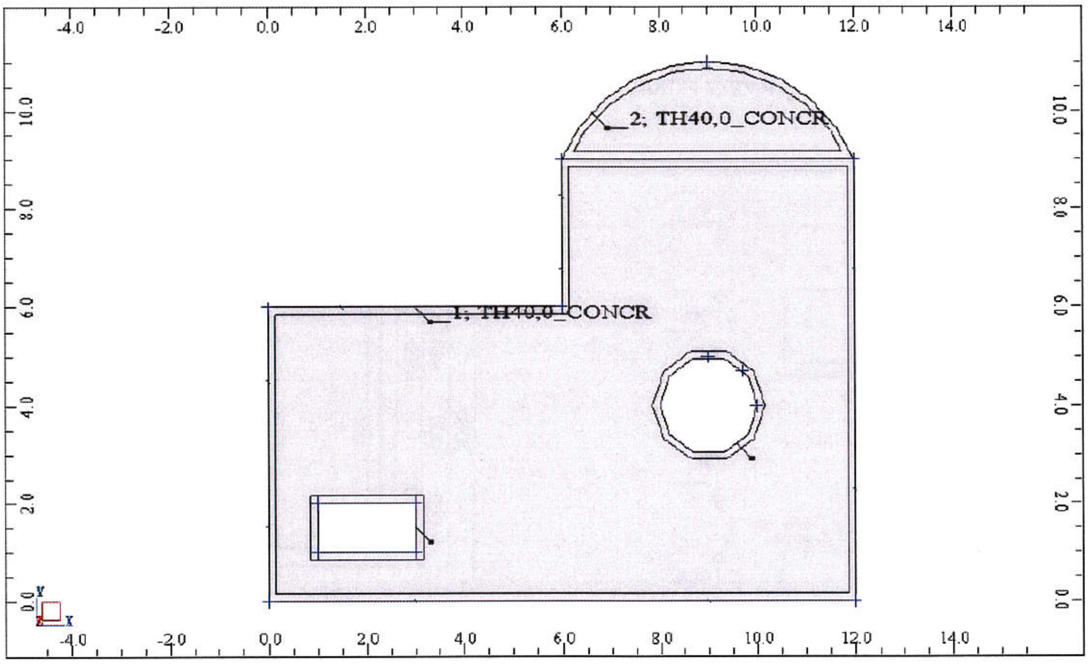
5. EXERCISE NO. 5 "DEFINITION OF A RC PLATE"

5.1. Structure geometry

5.1.1. Opening a new project

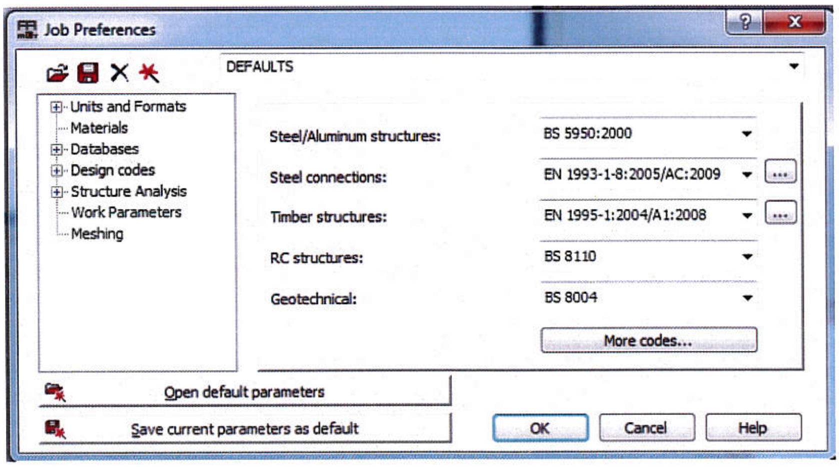
Enter ***Autodesk Robot Structural Analysis*** and select structure type *Plate Design* (icon ) or select the *File / New Project / Plate Design* command if the program is already running.

5.1.2. Proposed geometrical scheme of the plate



5.1.3. Setting the codes

Before commencing structure definition, one should check the codes to be applied and language to be used in the project. To do so, select the *Tools / Job Preferences* command from the main menu and check or set the codes as shown in the figures below.



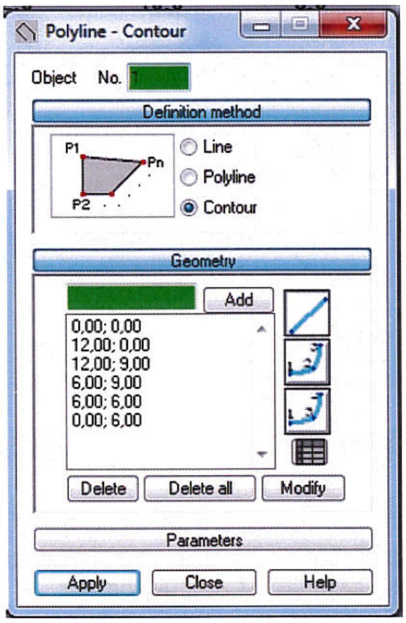
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

5.1.4. Definition of the contour of the lower part of the plate

To define a contour, enter the main menu and select the *Geometry / Objects / Polyline–contour* command or click the *Objects command* then *Polyline – contour* icon located in the right-hand toolbar. Once there appears the dialog box, one should define the settings as defined below:

- In the *Definition Method* field: *Contour,*
- In the *Geometry* field: introduce the following coordinates of points: 0,0; 12,0; 12,9; 6,9; 6,6; 0,6; accepting each point by pressing **Add**. Once the points are introduced, click the **Apply** button.

The points may also be introduced graphically, by placing the cursor in the *Geometry* edit field, and indicating the required points in the graphical viewer. Clicking the first of the defined points for the second time closes the contour.



5.1.5. Definition of the contour of the external, arc-shape part of the plate

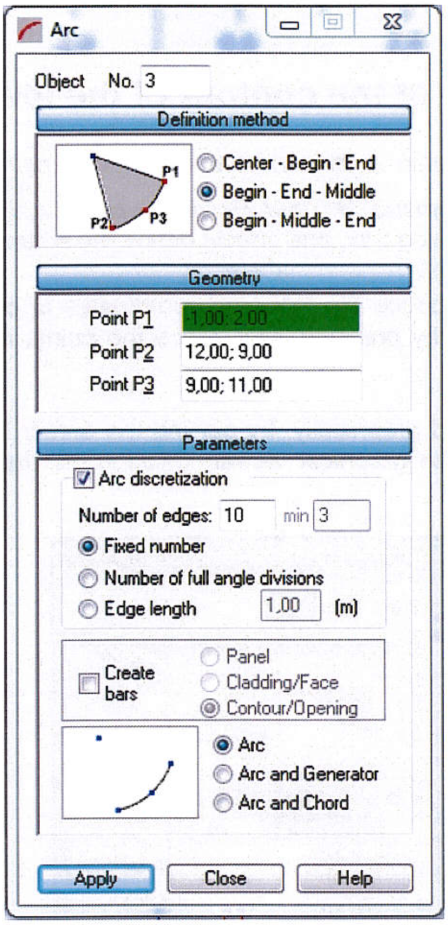
Arc definition: enter the main menu and select the *Geometry / Objects / Arc* command (or click the *Objects command* then the *Arc* icon located in the right-hand toolbar). Once there appears the dialog box, one should define the settings as shown below:

- In the *Definition Method* field: *Begin-End-Middle*
- In the *Geometry* field *Point P1:* 6,9
  Point P2: 12,9
  Point P3: 9,11
- In the *Parameters* field: *Sides* introduce (10)
  Switch on the *Arc discretization*.

Note: *Points from P1 to P3 may be introduced graphically as well, by indicating the required points in the graphical viewer.*

- Confirm by clicking **Apply**. Close the dialog box by clicking **Close**.

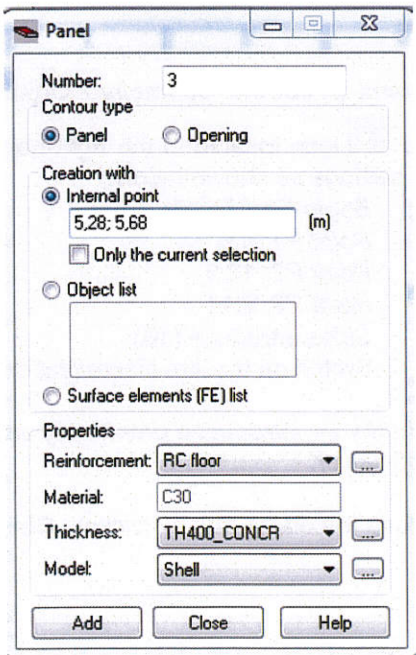
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



5.1.6. Definition of the properties of the plate

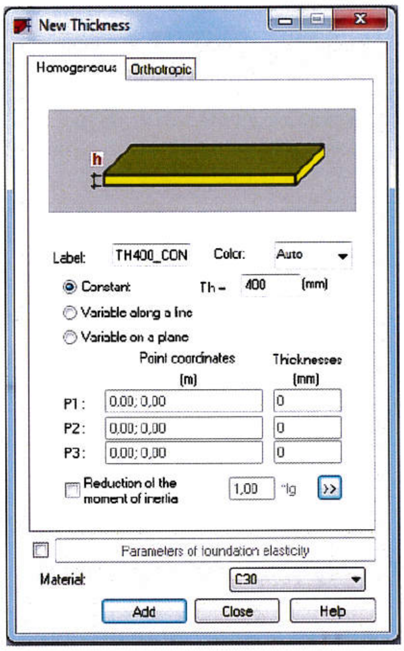
To define plate properties, one should enter the main menu and select the *Geometry / Panels* command (or click the *Panels* icon located in the right-hand toolbar). Once there appears the ***Panel*** dialog box, one should define the settings as defined below:

- *Contour Type:* check out the *Panel* option,
- *Creation With:* check out the *Internal Point* option.



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

To define new thickness, click the ... button to the right of the *Thickness* field and set the appropriate values in the dialog box that appears after clicking (see the picture below).



In the *Th* = field enter *400* value; in the *Label* field introduce name *TH400_CONCR*; in the *Material* field check out the *C30* option, then confirm the operation clicking the **Add** button. Close the dialog box by clicking the **Close** button. Move the cursor to the *Reinforcement* field situated in the *Properties* field and set the reinforcement type as *none*. Set the cursor in the *Internal Point* field. Move the mouse cursor to the graphical viewer and indicate once a point within the boundaries of the angular plate then, indicate a point within the boundaries of the second, smaller plate.
Confirm the settings by clicking **Add** and close the ***New Thickness*** dialog box by clicking **Close**.

5.1.7. Definition of openings within the contour

Rectangular opening

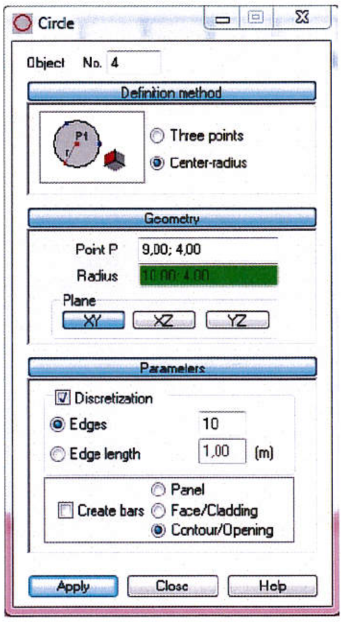
To define a rectangular opening, select from the main menu the *Geometry / Objects / Polyline–contour* command (or click the *Objects command* then *Polyline – contour* icon located in the right-hand toolbar). Once there appears the dialog box introduce the following coordinates of the opening in the *Geometry* field of the opened ***Polyline–contour*** dialog box: *1,1; 3,1; 3,2; 1,2*. Confirm the introduction of each point by clicking **Add**. Once the points are introduced, one should click **Apply**.
The points may also be introduced graphically, by placing the cursor in the *Geometry* edit field, and indicating the required points in the graphical viewer. To close the contour one should introduce the first point once again. Close the dialog box by clicking the **Close** button.

Circular opening
Enter the main menu and select the *Geometry / Objects / Circle* command (or click the *Objects command* then *Circle* icon located in the right-hand toolbar). Once there appears the dialog box, one should define the settings as shown below:

– In the *Definition Method* field:	*center-radius*,
– In the *Geometry* field:	*Point P* introduce coordinates (9,4)
	Radius introduce value (10,4)
– In the *Parameters* field:	*Sides* introduce 10,
	Switch on the *Discretization*,

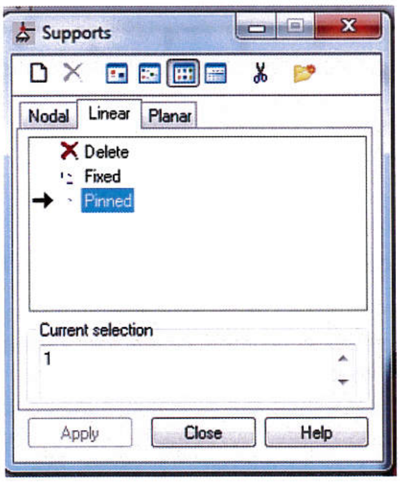
Leave the default values of the remaining parameters. Confirm by clicking **Apply**. Close the dialog box by clicking **Close**.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



5.1.8. Definition of support conditions

To define supports, one should select the *Geometry / Support* command from the main menu or click the *Supports* icon located in the right-hand toolbar. Once there appears the dialog box (see the picture below), one should select the *Pinned* support type by means of the mouse cursor and in the *Current Selection* field check out the *Line* option. Switch to the graphical viewer, point the plate edges and click once the edge, which is highlighted.



5.2. Definition of plate loads

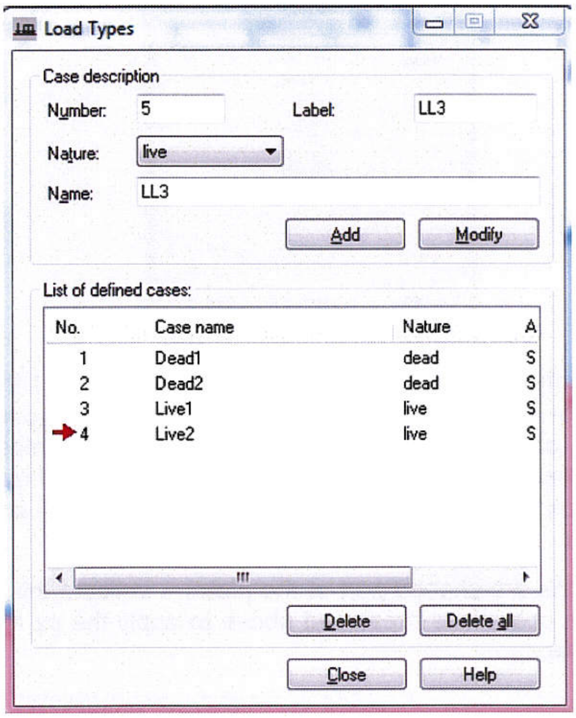
In order to define load cases, one should enter the main menu and select the *Loads / Load Types* command or click the *Load Types* icon located in the right-hand toolbar. Once there appears the dialog box (see the picture below), generate the following load cases:

- *Dead (DL1)* – in the *Nature* field, one should select the *dead* case and click **New**. This load case is automatically defined as the *self-weight* load, and set in the first position in the *Loads* table
- *Dead (DL2)* – to define it, one should click the **New** button located to the right of the *Nature* field,
- *Live1 (LL1)* – in the *Nature* field, one should select the *live* case and click the **New**,

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

– *Live2 (LL2)* – one should click **New** again,

All the defined cases are registered in the *List of Defined Cases* field,

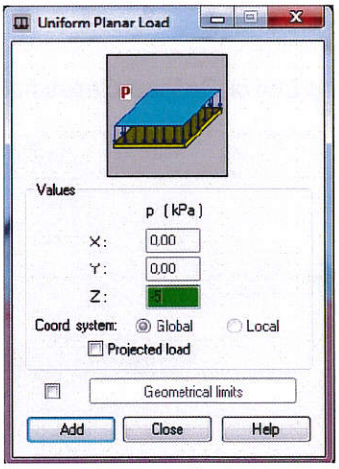


In order to determine the values of particular loads, one should indicate successively (red arrow) the following items in the *List of Defined Cases* list:

- *DL2* (dead load applied to the entire plate surface) – enter the main menu and select the *Loads / Load Definition* command (or click the *Load Definition* icon from the right-hand toolbar) and go to the *Surface* tab in the dialog box that has just opened (see the picture below).

Click the *Uniform Planar Load* icon and introduce the load value pz = - 5 (kPa) into the new dialog box (see the picture below),

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

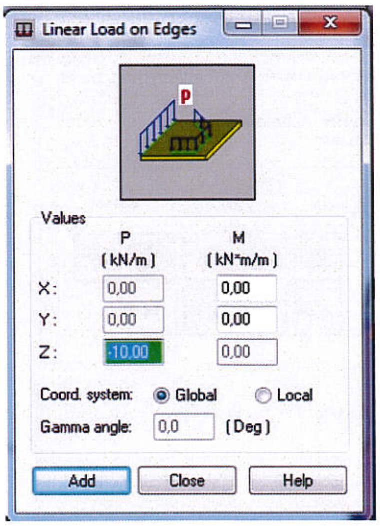


Once the operation is confirmed by pressing **Add**, which causes that the ***Uniform Planar Load*** dialog box disappears, set the cursor in the *Apply To* field (in the ***Load Definition*** dialog box). Then indicate the names of both panels (click when they get highlighted) with the mouse cursor (with the **Ctrl** button pressed). The panel numbers become registered in the *Apply To* field. Then, click the **Apply** button (thus, the load is applied to the indicated panels) and close the dialog box by pressing the **Close** button.

- *LL1* (live load applied to the arc-shaped part of the plate) – indicate this case in the ***Load Types*** dialog box and repeat the operations presented above to apply the pz = -2 (kPa) load only to the arc-shaped part of the plate.

- *LL2* (linear live load on the edge of the rectangular opening) - indicate this case in the ***Load Types*** dialog box and open the ***Load Definition*** dialog box; go to the *Surface* tab and click the *Linear Load on edges icon*. Once there appears a new dialog box (shown on the picture below), one should introduce the following load parameters: PZ = -10 kN/m; then, press the **Apply** button and click on rectangular opening contour

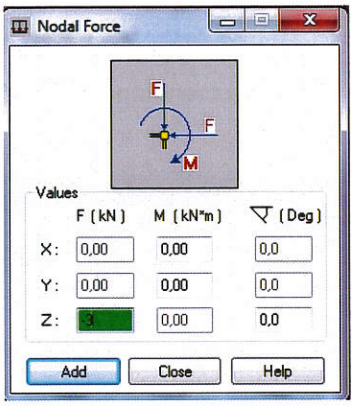
.



- *LL2* (live load, concentrated in the nodes of the circular opening. To make this operation easy one should define nodes in circular contour edges) – the case is already selected in the ***Load Types*** dialog box; open the ***Load Definition*** dialog box and go to the *Node* tab; click the *Nodal Force* icon. Once there appears a new dialog box (shown on the picture below), one should introduce the following load parameters: Fz = -3 kN and click the **Add** button. Set the cursor in the *Apply To* field of the ***Load Definition*** dialog box, go to the graphical viewer and select with window-shaped cursor the circular opening in the plate.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Confirm the selection by pressing **Apply** and close the ***Load Definition*** dialog box. Close the ***Load Types*** dialog box.



It is also possible to define loads in the relevant table. To do so, one should go to the layout of loads by indicating *Loads* option in combo-box in the top right corner. Once the layout changes, one should define appropriate load types and define their values in the table.

Save structure definition under the name **Exercise 9.1**.

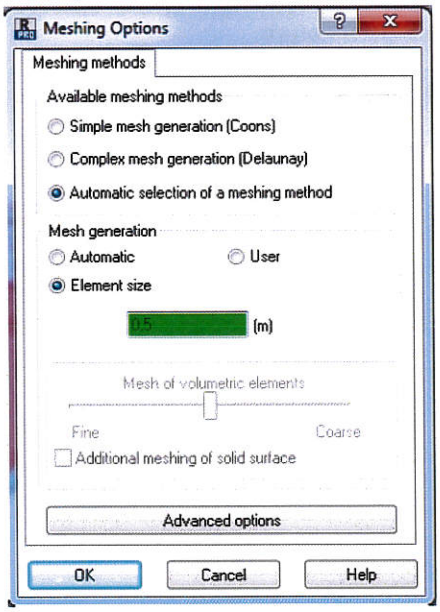
5.3. Meshing options

5.3.1. Definition of mesh parameters

In order to set globally the meshing parameters, one should enter the main menu and select the *Tools / Job Preferences / Meshing Options* command, and then click **Modification** button. Once the ***Meshing Options*** dialog box appears on screen (see the picture below), one may define the settings.

To define meshing parameters individually for each selected panels one should select panel/panels and open *Analysis / Meshing / Meshing options*. We select all panels and open *Meshing options* then

- Indicate the *Complex mesh generation (Deulaney)* option in the *Meshing Methods* field.
- Indicate *Element size 0.5 m* the *Mesh Generation* field

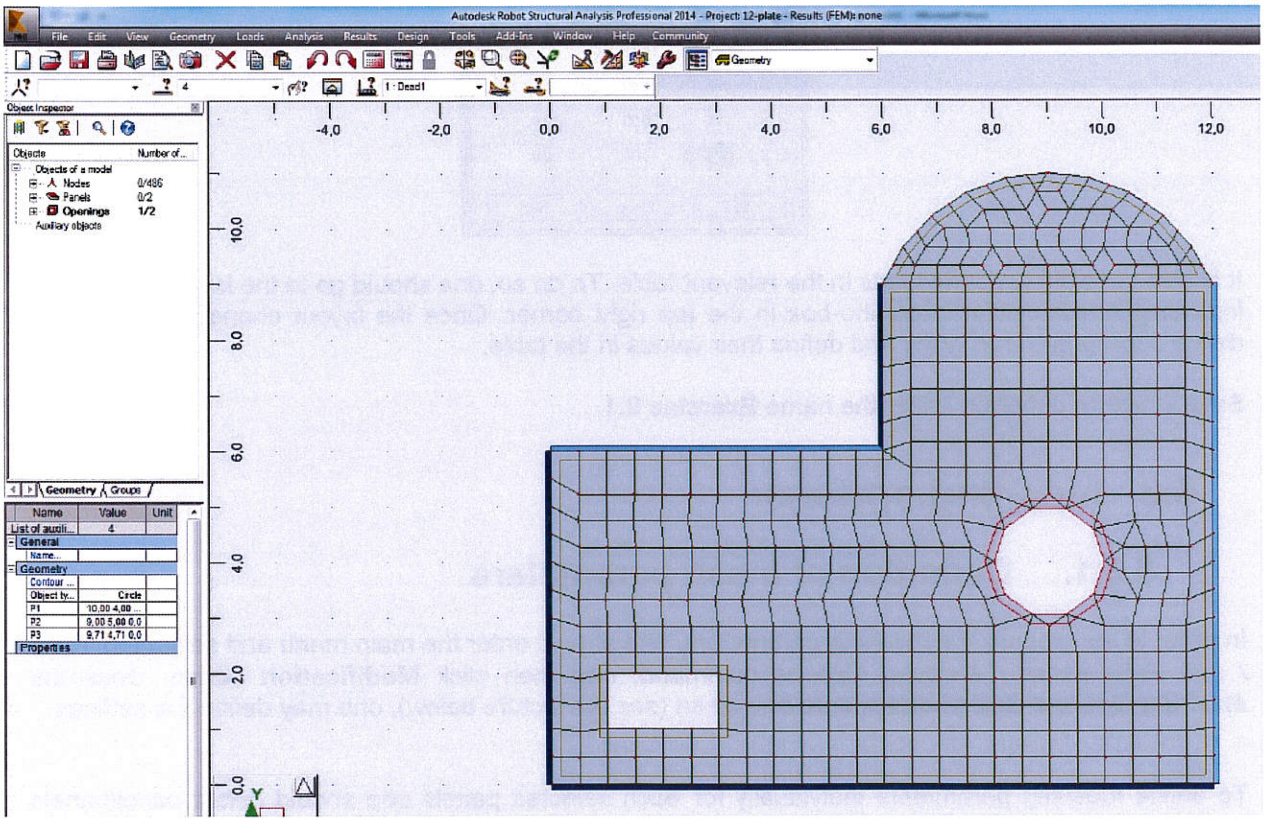


Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Leave the default values of the remaining parameters. Close the dialog box by pressing **OK**. Confirm the operation clicking the **OK** button.

5.3.2. Preview of the finite element mesh

To obtain the preview, one should enter the main menu and select the *Analysis / Meshing / Local mesh generation* command.



5.3.3. Possible modification of mesh parameters

In order to modify mesh parameters for selected panels one should select panel and from the menu *Analysis / Meshing / Meshing options.* command and in the opened ***Meshing Options*** dialog box made adequate modifications of mesh parameters.

Save the structure under the name **12-plate**

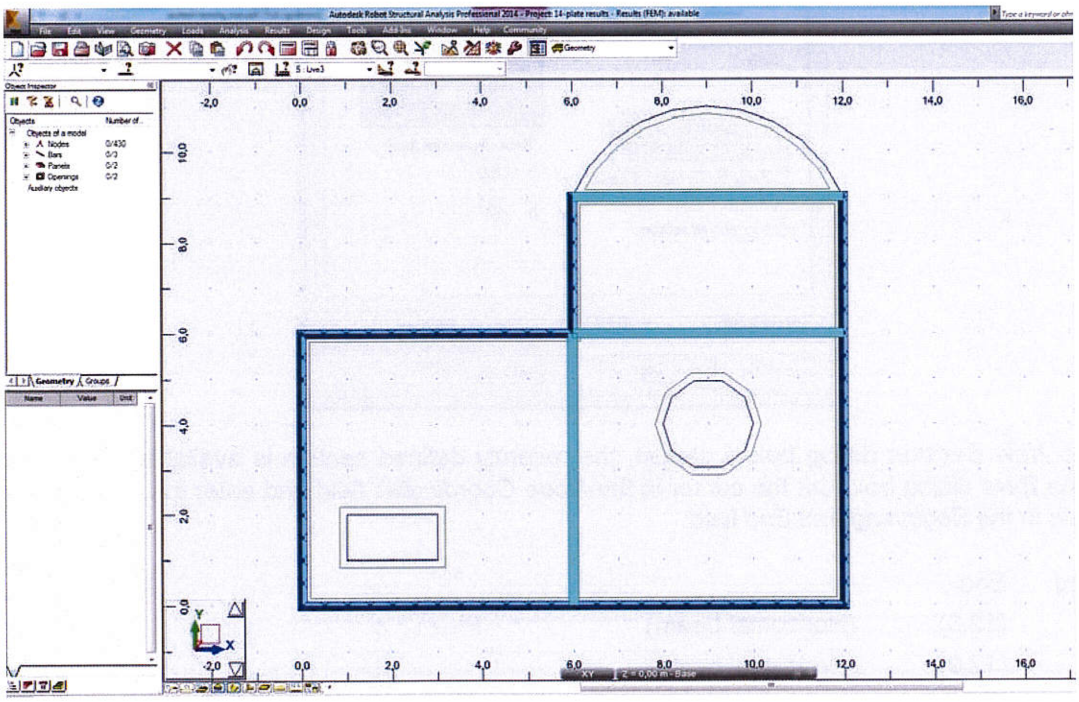
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

6. EXERCISE NO. 6 "MODIFICATION OF PLATE SCHEME; DEFINITION OF A GRILLAGE OF ADDITIONAL NODES FOR CONCENTRATED LOADS; LOAD COMBINATIONS"

6.1. Structure geometry

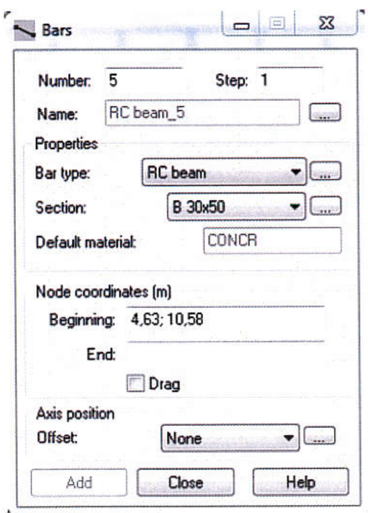
Open the **12-plate** file

6.1.1. Proposed scheme of the plate



6.1.2. Definition of beams supporting the plate

In order to define beams, one should enter the main menu and select the *Geometry / Bars* command or use the *Bars* icon from the right toolbar.

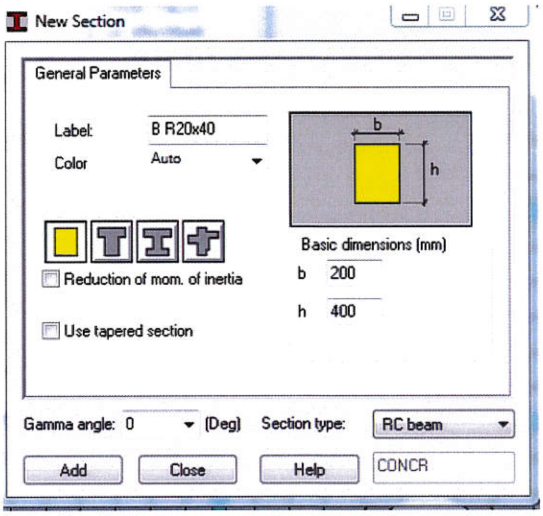


Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Select *RC Beam* option in the *Bar Type* field of the ***Bars*** dialog box. Click the ... button located to the right of the *Section* field. This results in the appearance of the ***New Section*** dialog box (shown on the picture below) and define a concrete beam with the section dimensions *200x400*. One should set the following parameters in the dialog box:

- Select *RC beam* in the *Section Type* field,
- Introduce the following values in the *Basic Dimension* field: *b = 200, h = 400,*
- In the *Label* field program will generate automatically the name for a section being defined *B R20x40*

Confirm settings by pressing **Add** button and close the dialog box by pressing **Close** button,



Once the ***New Section*** dialog box is closed, the recently defined section is available in the *Section* field of the ***Bars*** dialog box. Set the cursor in the *Node Coordinates* field and enter the following points coordinate in the *Beginning* and *End* field:

Beginning	End	
(6,6),	(12,6),	(Horizontal beam)
(6,9),	(12,9),	(Horizontal beam)
(6,0)	(6,6)	(Vertical beam)

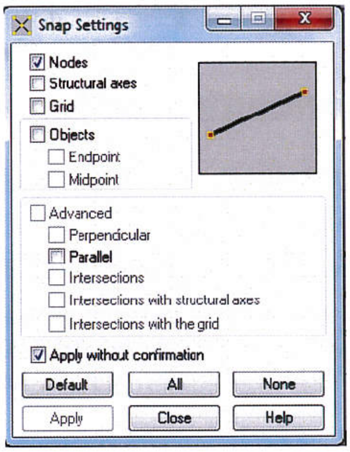
6.1.3. Definition of the beam supporting the arc-shaped part of the plate

There are two ways to define an arc-shaped beam supporting the arc-shaped part of the plate.

First method

Select the *Geometry / Bars* command from the main menu, which opens the ***Bars*** dialog box. Open the context menu (available by clicking the right-hand mouse button) and select the *Snap Settings / Snap Settings* command, which open the ***Snap Settings*** dialog box presented on the drawing below.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Switch off the following options:

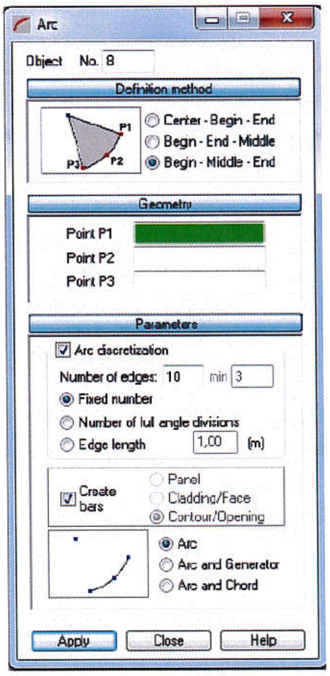
- *Structural axes*
- *Grid*
- *Advanced*
- *Apply without confirmation*

Confirm the operation clicking the **Apply** button; close the dialog box clicking the **Close** button. The settings allow one to define a beam on a plate arc by drawing it from-point-to-point with the *Drag* option switched on. One should remember that the plate fragment should be zoomed in appropriately to the needs. Once the operation is finished, one should close the ***Bars*** dialog box.

Second method

Activate the *Geometry / Objects / Arc* command from the main menu or click the *Objects command*  then *Arc*  icon. Once the relevant dialog box is opened (see the picture below), set the following parameters:

- In the *Definition Method* field select the *Begin – Middle - End* option.
- switch on *Arc discretization*
- Introduce *Number of edges 10* in the *Parameters* field and switch on the *Explode* option.
- switch on *Create bars*
- Set the cursor in *Point P1* of the *Geometry* field, go to the graphical viewer and indicate the beginning, intermediate and end points of the defined arc.



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Confirm arc generation by clicking **Apply** button and close the dialog box by clicking **Close**.

6.2. Definition of concentrated forces

After selecting all panels and running *Analysis / Meshing / Local mesh deleting* one can remove previously defined mesh

Then one can define two nodes with the following coordinates: (8; 5) and (9; 10). Open the *Geometry / Nodes* option by means of the main menu command (or click the *Nodes* icon). Introduce (8; 5) coordinates in the *Coordinates* field and confirm with the **Add** button. Introduce coordinates of the second point (9; 10) in the *Coordinates* field and confirm with the **Add** button. Two new nodes are thus created. Nodal forces will be applied in these nodes.

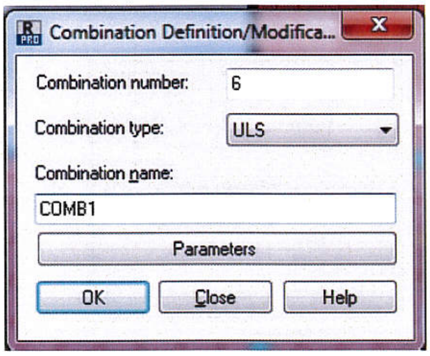
In order to define a new load case, one should select the *Loads / Load Types* command from the main menu (or click the *Load Types* icon). After opening the dialog box for load definition, define a new load case of the *Live (LL3)* nature. Highlight the recently defined load in the *List of Defined Cases* field and open the relevant dialog box by means of the *Loads / Load Definition* command from the main menu (or click the *Load Definition* icon). Select the *Nodal Force* load type in the *Node* tab of the dialog box. Define the force
Fz = -5kN and confirm by clicking **Add**. Set the cursor in the *Apply To* field of the ***Load Definition*** dialog box and introduce the numbers of the two above-defined nodes. Confirm by clicking the **Apply** button. Close the dialog box by clicking **Close** button.

6.3. Model generation

In order to carry out model generation, one should select the *Analysis / Mesh / Local mesh generation* command from the main menu. Once the option is selected, the program adjusts the FE mesh to the newly defined nodes.

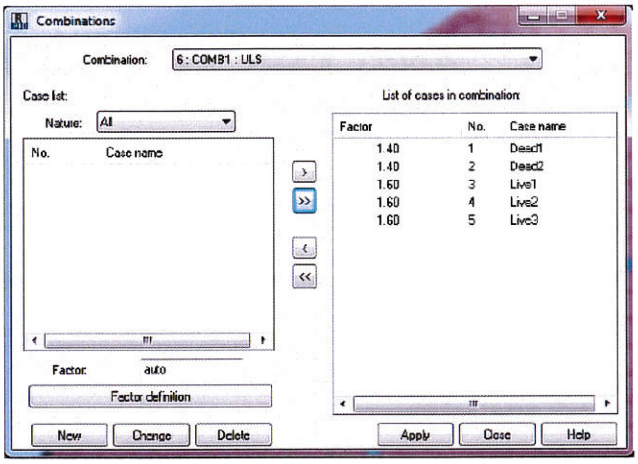
6.4. Definition of a load combination

In order to define a load combination, one should select the *Loads / Manual Combinations* command from the main menu. Once the ***Combination Definition/Modification*** dialog box is opened (see below), one should check out the *ULS* option in the *Combination Type* field.



Once the operation is confirmed by clicking **OK,** there appears the ***Combinations*** dialog box that allows one to define combination parameters.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



The *Combination* field contains the name and type of the generated combination. The *Case List* field contains all the load cases that may be included in the new combination. The *Factor* field allows one to modify the combination factor. The default factor values may be checked and modified (for a nature) by clicking the **Factor Definition** button. In order to carry out a modification of the factor, one should indicate one of the load cases (e.g. *LL1*) and introduce the new factor value in the *Factor* field, e.g. *1.49*, and then click the > button. Then, the selected case will be moved to the right-hand panel of the ***List of Cases in Combinations*** dialog box with the new factor value. The remaining load cases should be moved to the right-hand panel without factor modification by clicking the >> button. Confirm by clicking the **Apply** button.

Note: *In the exercise all factors was set as default.*

In order to define a new combination type, one should click the *New* option. Once the ***Combination Definition/Modification*** dialog box appears on screen, one should check out the **SLS** combination type and confirm by clicking **OK**.
In the ***Combinations*** dialog box, one should select all simple load cases (without combinations) and move them to the right-hand panel of the *List of Cases in Combinations* field. The above settings should be confirmed by pressing the **Apply** button. Close the dialog box by clicking **Close.**

Save the structure as **13-plate modification.**

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

7. EXERCISE NO. 7 "ANALYSIS OF RESULTS FOR A PLATE"

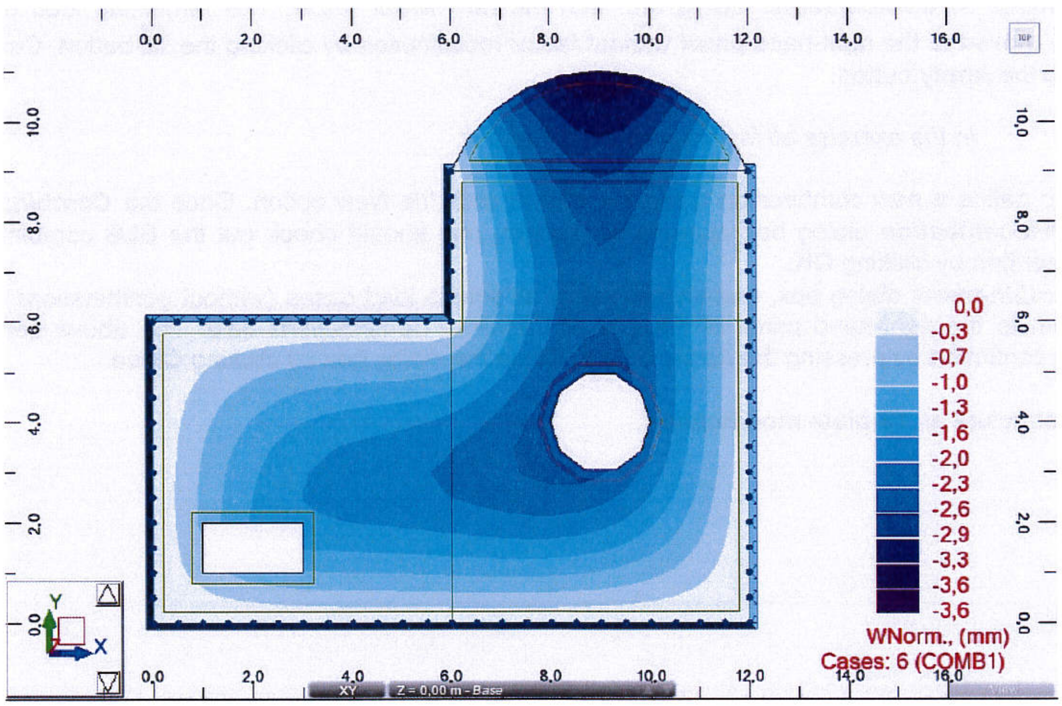
7.1. Running calculations

Open the Exercise 10 project. To start calculations select from the menu: *Analysis / Calculations* or press the *Calculations*  icon from toolbar.

7.2. Analysis of results

7.2.1. Results in the graphical form

To see the results in the graphical form select from the menu: *Results / Maps* that opens the ***Maps*** dialog box, which is used to display the chosen maps of the internal forces, stresses and displacements across the surface elements.



The ***Maps*** dialog box consists of seven tabs:

- *Detailed* tab, which is used for presentation of stresses, membrane forces, moments, shear stresses, displacements, rotations and soil resistance in the main direction. The *Direction X* option, which button is located in the upper portion of the dialog box, allows the user to define the main direction (x) of the co-ordinate system, which will be used by the user during surface FE result presentation (direction y is always perpendicular to x); or assumes the cylindrical coordinate system for the purposes of presentation (then the direction x runs along the radius and y is the circumference direction). The direction may be selected in several ways:
  - by selecting the axis of the global system (according the X, Y or Z) axis.
  - by defining a vector with the co-ordinates: X, Y and Z, which defines the main direction for definition of surface element calculation results. There is one limit in the selection of the direction; the vector cannot be perpendicular to the element (i.e. parallel to the local Z-axis). If

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

the user selects such a direction (the vector is projected onto the element as a point), then all results will be equal to zero.

- The following general convention of the presentation of results has been adopted:
  - e.g. *Sxx* – presentation of stresses (S) over the cross-section created by cutting the plate with a line perpendicular to the selected base direction (**x**); the stress direction agrees with the base direction,
  - e.g. *Myy* presentation of moments (M) over the cross-section created by cutting the plate with the normal line with respect to the base direction (y); the moment causes the appearance of stresses in the direction perpendicular to the base direction (y), situation is analogous in the case of other result types.

All results are shown in nodes or centers of surface elements of local, user-defined co-ordinate system and the selected layer. The upper part of the dialog box informs for which layer the calculation results will be presented.

- *Principal* tab allows for presentation of extreme values: stresses, membrane forces, moments, shear stresses and shear forces in the main coordinate system.

- *Complex* tab is used for presentation of the reduced values of membrane forces, moments and stresses

- *Parameters* tab. On this tab user may select the layer for which the results will be presented. In the *Layer Selection* field user may select the layer parallel to the middle surface of the surface structure for which calculation results will be presented. The layer selection has no effect on shear forces or displacements, which are defined for the middle surface and are constant along the entire height of the FE cross-section; this selection is important during the presentation of stresses. The *Layer selection* option also has a strong influence on the values presented in the table. It allows for the selection of a layer parallel to the middle surface of the surface structure for which calculation results will be presented. The selection of the layer has no effect on shear forces or displacements, which are selected for the middle surface and are constant along the whole height of the FE cross-section; this selection is important during stress presentation.

- *Scale* tab allows one for definition of parameters of the scale for displaying results. The user may change the map presentation parameters:
  - the type of color palette
  - maximum and minimum division number, in which a map of the selected value will be presented (minimum and maximum value inclusion may also be turned off) and define the map color change division number
  - scale type
  - the colors using which the value map and values for the colors will be presented will

- *Deformation* tab allows for presentation of structure deformation and animation and contains the following options:
  - *active* - if one selects this option the program will present deformation of the currently designed structure
  - *Constant Scale* – it means that the same scale will be selected for all the presented diagrams Scale
  - *Scale +* pressing this button results in decreasing the number of units per 1 cm of a diagram of the selected quantity
  - *Scale -* pressing this button results in increasing the number of units per 1 cm of a diagram of the selected quantity
  - *Normalize* - pressing this button results in presenting maps of the selected quantities in such a way that the scale will be adjusted to the maximum and the minimum value of the selected quantity. This tab provides also options used for animation of structure deformation diagrams to be presented on screen. In order to run animation, one should define two animation parameters: number of frames and number of frames per second. When the **Start** button is pressed, the program prepares animation of the selected quantity on the basis of the defined

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

parameters and starts to run the animation. When the animation is presented, there appears on screen a toolbar that allows one to stop animation, start it again, rewind it, etc.

- *Crosses* tab. The results for planar finite elements may be presented in the form of crosses. The crosses of a selected quantity will be presented if one selects the *Active* option. Results may be presented in the form of crosses for three types of quantities: stresses, forces and moments.

In the lower portion of the ***Maps*** dialog box there are options that allow for selecting form of results presentation:

- *Isolines* – if this option is selected, results obtained for surface FE will be presented in the form of isolines.
- *Maps* if this option is selected, results obtained for surface FE will be presented in the form of maps.
- *Values* if this option is selected, results obtained for surface FE will be presented in the form of values
- *With Description* – if this option is turn on, maps will be presented with a description of the values of individual isolines and maps.
- *With Normalization* – if this option is active, the maps of a selected quantity will be automatically presented in such a way that the scale will be adjusted to the maximum and the minimum value of the selected quantity.
- *Open New Window* – if this option is turn on, a new window in which maps or isolines of the values selected will appear on the screen.
- *Smoothing* – results for surface FE are obtained in the Gauss points located within each FE (results estimated in the common nodes adjoined elements may differentiate a bit in each element and isolines might be discontinuous). To obtain 'smooth' map of selected quantity, one should choose the *With Smoothing* option.

To see the results in the graphical form select from the load case list: *1: DL1*. From the **text** menu select: *Results / Maps* that opens the ***Maps*** dialog box. In the ***Detailed*** dialog box select the *Displacement u,w* option and press the **Apply** button. To make drawing more clear select from the main menu the *View / Display* command, which opens the ***Display*** dialog box. On the *Sections* tab switch off the *Section – shape* option, similarly on the *Finite elements* tab switch off the *Numbers and panels description* option.

It is also possible to see results in the map form on the panel cuts. To do so select from the main menu *Results/ Panel Cuts* option, which results in displaying sectional force diagrams in the chosen plate or shell section.
In the ***Panel Cuts*** dialog box, which appears on the screen there are available nine tabs. Some of them are analogous to that one available in the ***Maps*** dialog box and some of them are new one:

- *Definition* tab - the options provided on this tab allow one to define a new cut
- *Cuts* tab - presents all the cuts defined for a structure. For each cut three pieces of information are shown thus: presence of the cut (if the option is active, the cut will be presented on the structure together with the selected diagrams of the indicated quantities), color and finally the name of the cut
- *Diagrams* tab- allows one to select the manner of structure diagram presentation
- *Reinforcement* tab - allows one to select for presentation some quantities connected with the reinforcement
- *SLS* tab - allows one to select for presentation following parameters: cracking width and deflection
  Note: the tab is available only if there has been selected a RC code recognizing calculations according to SLS in the program.

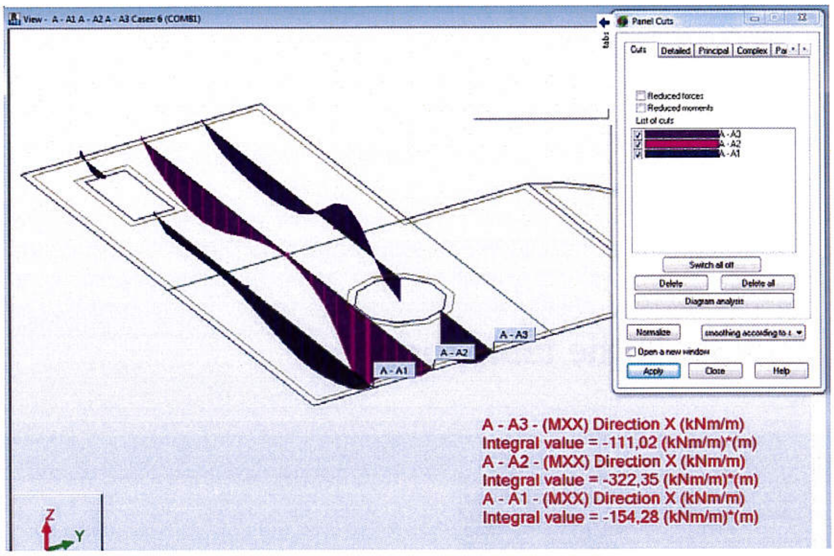
In order to see the results on the cuts, select from load list *1: DL1* load case then from the main menu chose: *Results / Panel Cuts* option that opens the ***Panel Cuts*** dialog box. In the ***Detailed*** dialog box select the *Rotations -R yy* option. Then go to the *Definition* tab and selecting the *2 points* option enter

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

the coordinates of the cut e.g. (0,00; 0,00) and (12,00; 6,00). Set the layer selection as middle in the *Parameters* tab and finally set the following options available on the *Diagrams* tab:

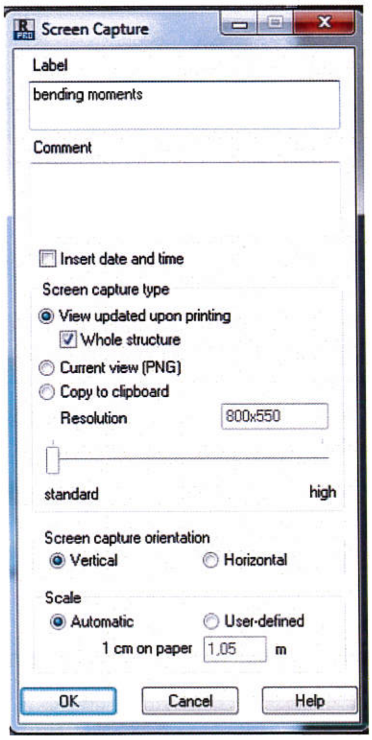
- *filled* in the *Filling* field
- *normal* option in the *Diagram position* field

Confirm the operation pressing the **Apply** button.
The map of chosen quantity on the selected panel cut will be added to the list of the available cuts and presented on the screen (see the picture below).



7.2.2. Screen captures

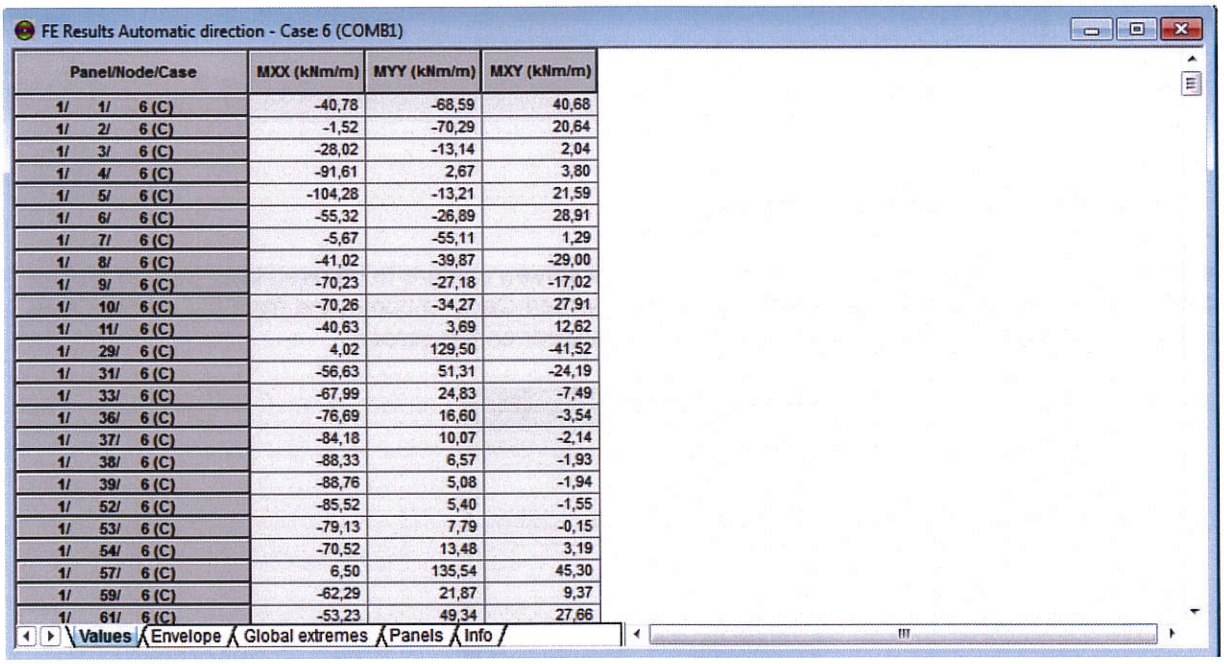
The *Screen Capture* option allows for saving the current screen capture to be used later (for the project documentation). It is available after selecting the *File/Screen Capture* command from the menu. Once the option is selected, the dialog box shown below will appear on the screen.



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

To capture viewer contents type in a screen capture name (or accept a standard name provided within the program) and press the **OK** button. The screen capture will be saved to the left panel of the *Screen Captures* tab in the ***Printout Composition*** dialog box. To see screen capture, choose from the menu: ***File / Printout Composition – Wizard*** dialog box and go to the *Screen Capture* tab divided into two panels. The left panel presents the names of the screens captured by the user while the right panel contains the complete printout composed by the user from the object available on the left panel. To add defined screen captures to the right panel highlight the appropriate screen capture and click the **Add** button what results in adding to the printout only the screen capture that is selected (highlighted) in the left panel. To see the preview of the selected screen capture click the *Preview of selected components* icon.

7.2.3. Results in the table form



Panel/Node/Case	MXX (kNm/m)	MYY (kNm/m)	MXY (kNm/m)
1/ 1/ 6 (C)	-40,78	-68,59	40,68
1/ 2/ 6 (C)	-1,52	-70,29	20,64
1/ 3/ 6 (C)	-28,02	-13,14	2,04
1/ 4/ 6 (C)	-91,61	2,67	3,80
1/ 5/ 6 (C)	-104,28	-13,21	21,59
1/ 6/ 6 (C)	-55,32	-26,89	28,91
1/ 7/ 6 (C)	-5,67	-55,11	1,29
1/ 8/ 6 (C)	-41,02	-39,87	-29,00
1/ 9/ 6 (C)	-70,23	-27,18	-17,02
1/ 10/ 6 (C)	-70,26	-34,27	27,91
1/ 11/ 6 (C)	-40,63	3,69	12,62
1/ 29/ 6 (C)	4,02	129,50	-41,52
1/ 31/ 6 (C)	-56,63	51,31	-24,19
1/ 33/ 6 (C)	-67,99	24,83	-7,49
1/ 36/ 6 (C)	-76,69	16,60	-3,54
1/ 37/ 6 (C)	-84,18	10,07	-2,14
1/ 38/ 6 (C)	-88,33	6,57	-1,93
1/ 39/ 6 (C)	-88,76	5,08	-1,94
1/ 52/ 6 (C)	-85,52	5,40	-1,55
1/ 53/ 6 (C)	-79,13	7,79	-0,15
1/ 54/ 6 (C)	-70,52	13,48	3,19
1/ 57/ 6 (C)	6,50	135,54	45,30
1/ 59/ 6 (C)	-62,29	21,87	9,37
1/ 61/ 6 (C)	-53,23	49,34	27,66

To obtain the results in the table form one should select from the menu *Results / Plate and shell Results* that opens the ***FE Results*** dialog box, which is divided into four tabs: *Values*, *Envelope*, *Global Extremes* and *Info*.
To change table configuration click the right mouse button and select *Table Columns* option that opens the ***Result for Finite Elements*** dialog box, which consist a few tabs:

- *Detailed* tab allows presentation of different quantities in the local coordinate system
- *Principal* tab. Options on this tab are used for presentation maintained quantities in the main coordinate system.
- *Complex* tab is used for presentation reduced Stresses and Moments.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

- *Parameters* tab. On this tab one can choose where the structure calculation results will be presented.

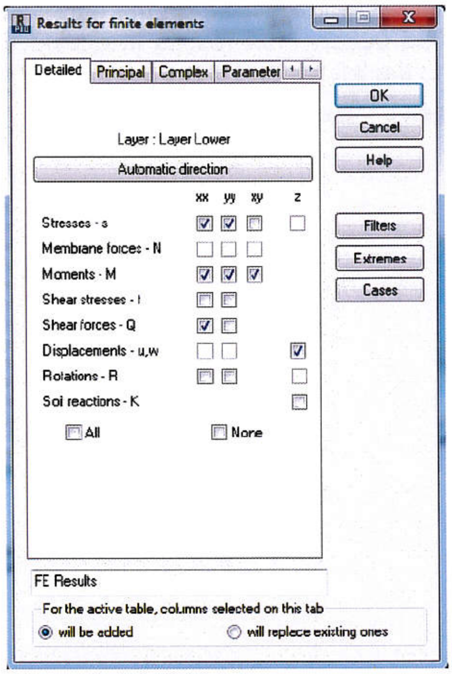
- *Data* tab allows presentation planar FE data: node numbers, material, section (thickness type), panel number and planar FE type.

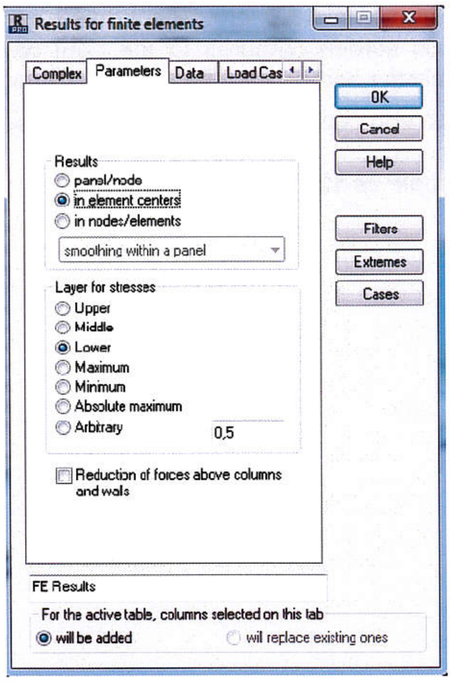
- *Load cases* tab. The Selection of presented information field allows one to select the following quantities to be presented in the table: case name, case nature, analysis type (linear, non-linear, buckling, etc.) and the definition of combinations (full combination definition with the coefficients for particular component cases).

- *Filters* tab. Options in this tab are used for value selection based on which sorting or filtering of data/results presented in the table will be done.

- *Extremes* tab

In the *Detailed* field activate (☑) options: *Stresses – s* in direction *xx*, *Shear Forces – Q* in direction *xx* and *Displacement u,w* in direction **z**. Press the **OK** button. The table with selected quantities will appear on the screen.



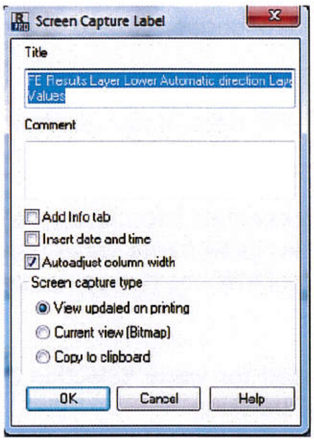


In the *Parameters* field activate:

- in the field *Results – in Element Centers* option
- in the field *Layer Selection – Lower* option

From load list select load case: *Simple Cases* To save the current screen capture select the command *File/Screen Capture* from the menu. Once the option is selected, the small dialog box shown below will appear on the screen.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



To capture viewer contents type in a screen capture name (or accept a standard name provided within the program) and press the **OK** button. The screen capture will be saved to the left panel of the *Screen Captures* tab in the ***Printout Composition*** dialog box. To see screen capture, choose from the menu: ***File / Printout Composition – Wizard*** dialog box and go to the *Screen Capture* tab divided into two panels. The left panel presents the names of the screens captured by the user while the right panel contains the complete printout composed by the user from the object available on the left panel. To add defined screen captures to the right panel click the **Add** button what results in adding to the printout only the screen capture that is selected (highlighted) in the left panel. To see the preview of the selected screen capture click the *Preview of selected components* icon.

Save example as **13-plate results.**

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

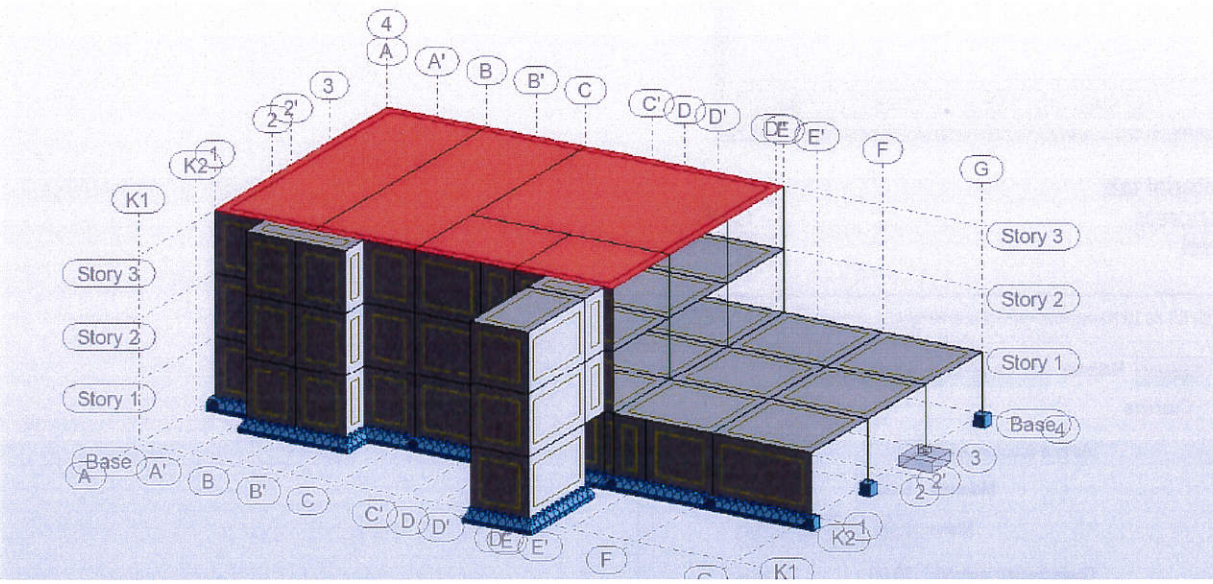
EXERCISE NO. 8 "RC PLATE REINFORCEMENT DESIGN"

8.1. Concrete slabs and walls dimensioning – design parameters

8.1.1. Concrete slab design parameters

Open file 15-slab reinforcement3d

Story 3 slab will be taken into account



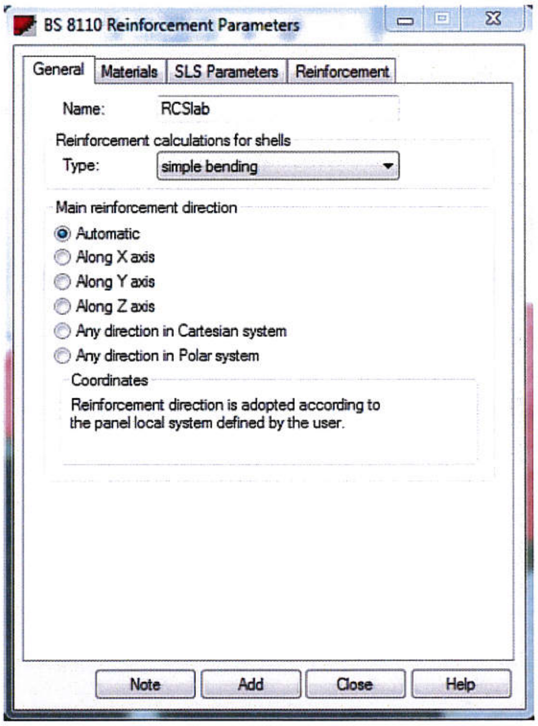
While dimensioning parameters definition one has to determine main reinforcement direction. In that case local panel coordinate system can be used. If main reinforcement direction will be set to **Automatic** it will be parallel to local panel X axis. In results AX area will be presented for reinforcement parallel to panel local x axis AY for reinforcement parallel to parallel local Y axis.

Slab design parameters can be set by **Design / Required reinforcement for Slab/Walls Option / Code Parameters**.

Proposed parameters for analyzed slab.

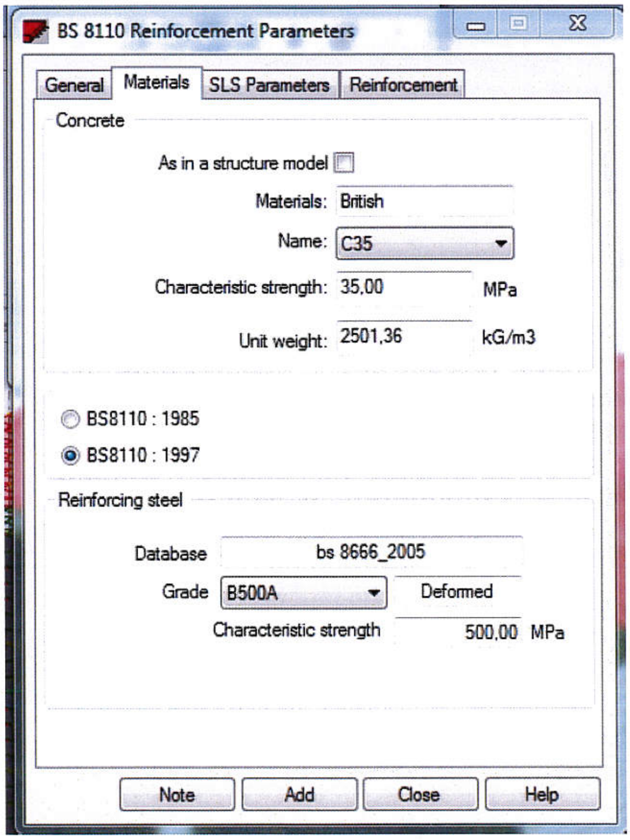
General tab:
Name - RC Slab
Type - simple bending
Main reinforcement direction - Automatic

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Material tab

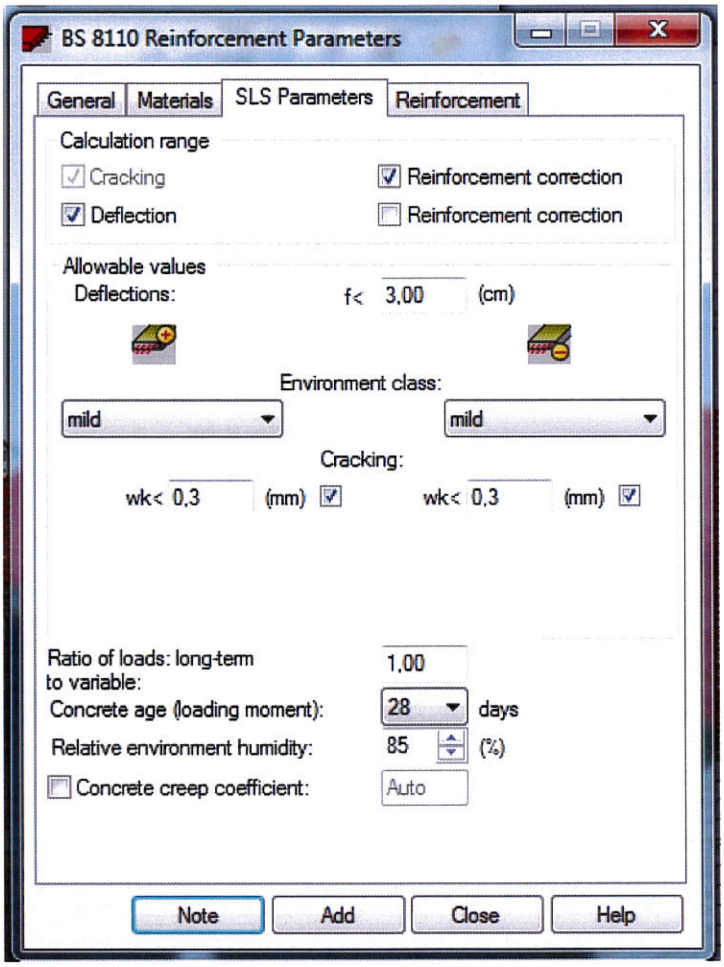
Concrete	- C35
Steel	- B500A



SLS Parameters tab

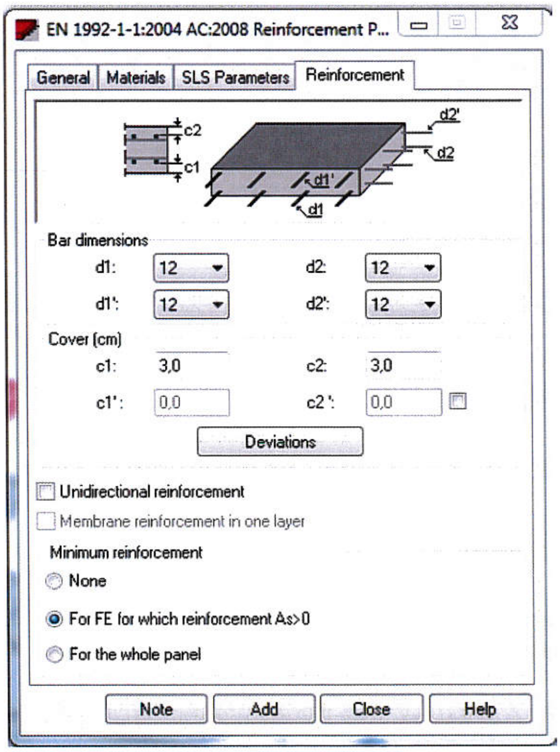
Calculation range	- Reinforcement correction for cracking, Deflection
Environment class	- mild
Cracking	- 0.3 mm

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Reinforcement tab

Bar dimensions	**- 12**
Cover	**- 3 cm**

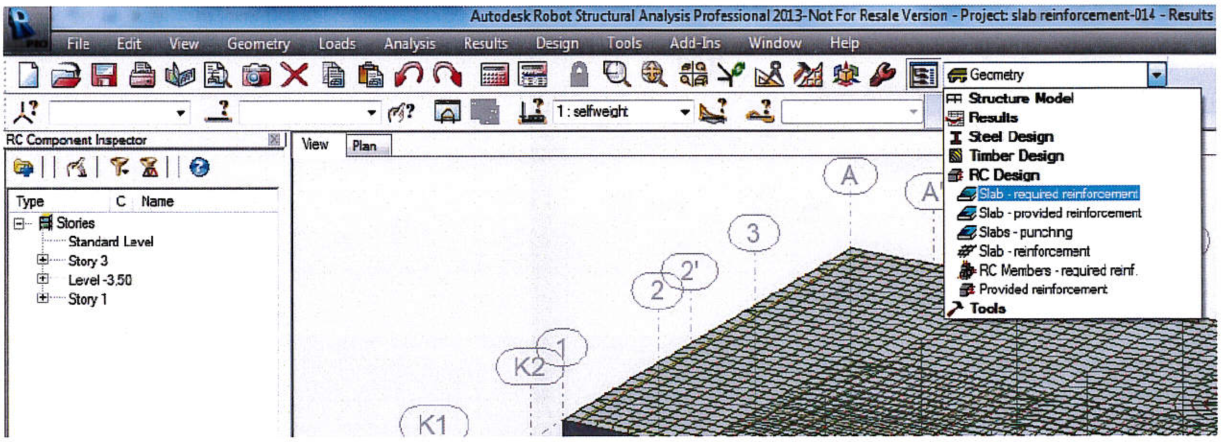


Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

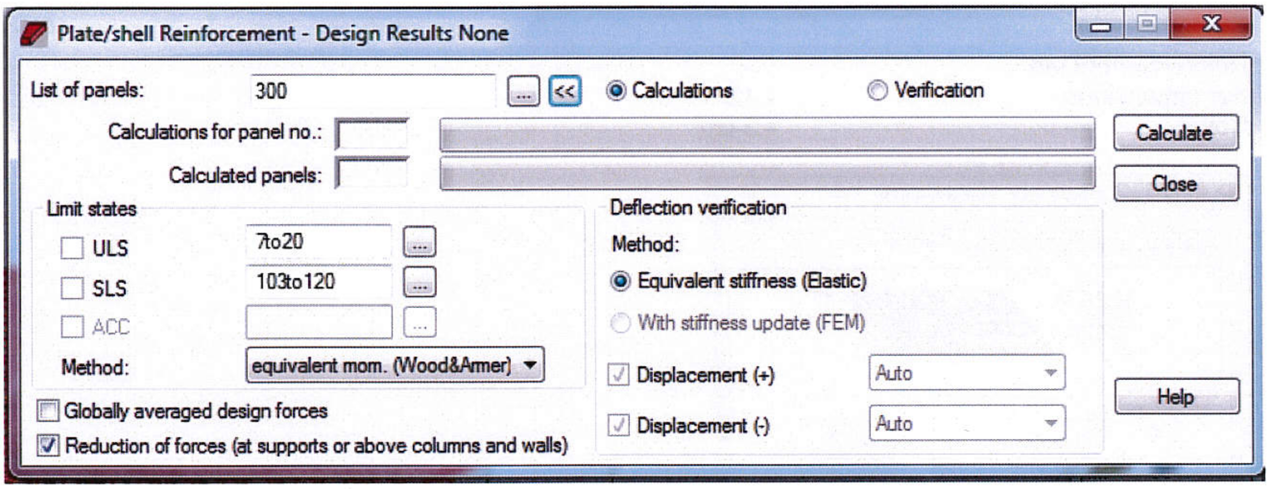
Once parameters are set one should click Add to save and assign new defined parameters to structure Story 3 slab.

8.1.2. Slab and wall reinforcement calculation

Once can use user's layout Slab – required reinforcement to launch calculation



After story 3 lab and staircase wall selection one can use „<<" in **Plate and shell reinforcement** window to calculate reinforcement for selected object only. Then Calculate can be launched. For objects calculated with compression/tension option on, **Method- analytical** should be used. Option **Reduction of forces at supports and above columns and walls** should be checked on to avoid reinforcement concentration above supports



8.1.3. Required reinforcement analysis

Provided reinforcement for plates and walls can be presented as reinforcement maps. Reinforcement area can be obtained in reinforcement table as well.

Reinforcement maps can be controlled from Reinforcement window

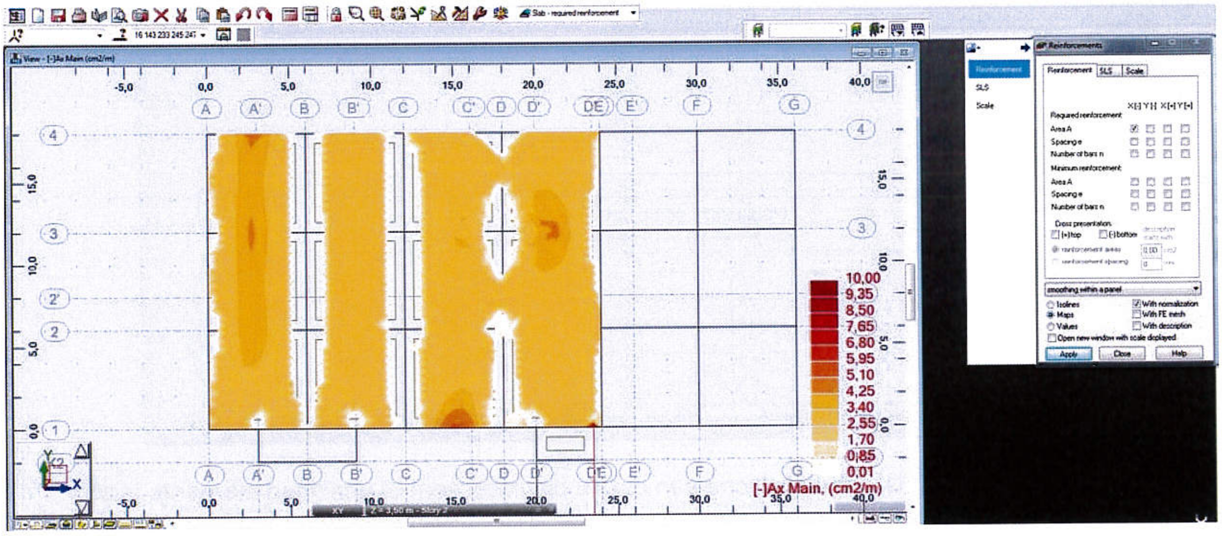
For Story 3 slab and Staircase wall results should be interpreted in the following way:
AX- - bottom reinforcement area in slab local X axis (Global X axis) for wall horizontal reinforcement area
AX+ - top reinforcement area in slab local X axis (Global X axis) for wall horizontal reinforcement area

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

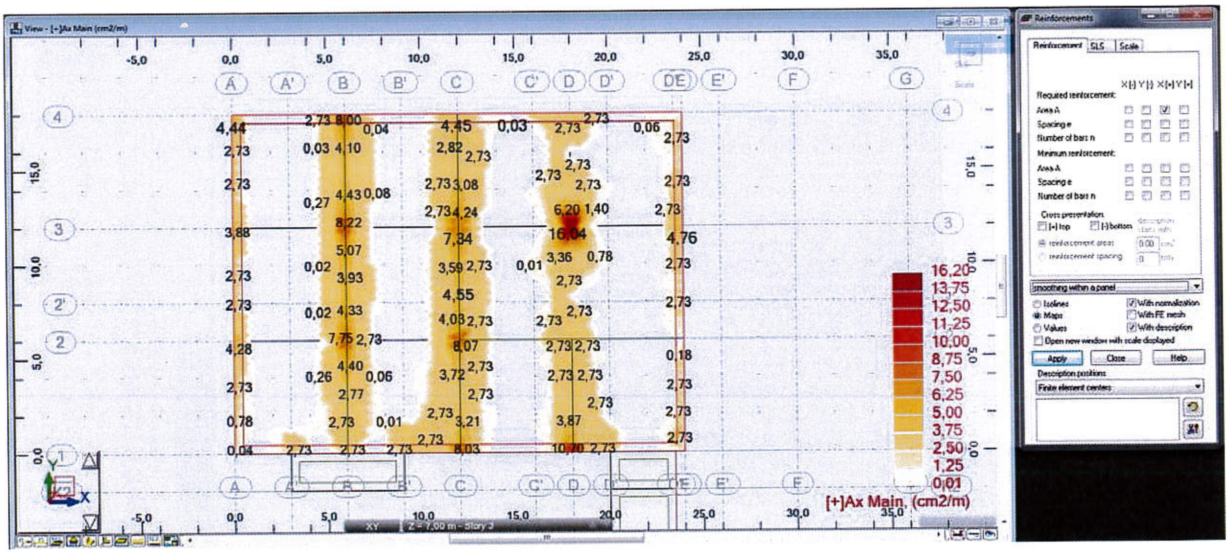
AY- area - bottom reinforcement area in slab local Y axis (Global Y axis) for wall vertical reinforcement area
AY+ - top reinforcement area in slab local Y axis (Global Y axis) for wall vertical reinforcement area

Reinforcement map for story 3 slab

Bottom reinforcement in global x direction



Results can be presented as values
Top reinforcement in global x direction



Results can be presented in reinforcement table (**Design / Required reinforcement for Slab/Walls Option** / Table Plate and shell reinforcement).

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

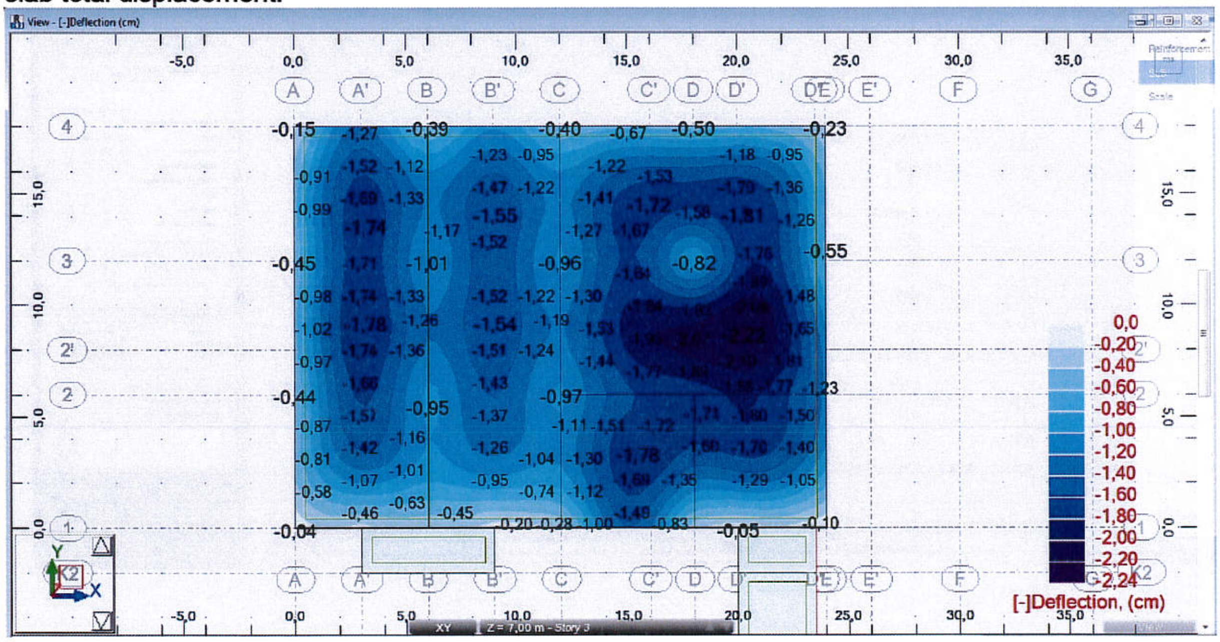
Reinforcement Areas

Panel/Node	[-]Ax Main (cm2/m)	[-]Ay Perpendicular	[+]Ax Main (cm2/m)	[+]Ay Perpendicular
300/ 9021	0,0	0,0	2,73	2,73
300/ 9022	0,0	0,0	2,73	2,73
300/ 9023	2,73	0,0	0,0	2,73
300/ 9024	3,28	2,73	0,0	2,73
300/ 9025	4,10	2,73	0,0	0,0
300/ 9026	4,35	2,73	0,0	0,0
300/ 9027	4,08	2,73	0,0	0,0
300/ 9028	3,30	2,73	0,0	2,73
300/ 9029	2,73	0,0	0,0	2,73
300/ 9030	2,73	0,0	2,73	2,73
300/ 9031	0,0	0,0	2,83	2,73
300/ 9032	0,0	0,0	4,44	2,73
300/ 9033	0,0	0,0	2,73	2,73
300/ 9034	2,73	0,0	0,0	2,73
300/ 9035	2,73	2,73	0,0	2,73
300/ 9036	2,73	2,73	0,0	2,73
300/ 9037	2,84	2,73	0,0	2,73
300/ 9038	2,73	2,73	0,0	2,73
300/ 9039	2,73	0,0	0,0	2,73
300/ 9040	2,73	0,0	0,0	2,73
300/ 9041	0,0	0,0	2,73	2,73
300/ 9042	0,0	0,0	2,73	2,73
300/ 9043	2,73	0,0	0,0	2,73
300/ 9044	2,73	0,0	0,0	2,73
300/ 9045	2,92	0,0	0,0	2,73
300/ 9046	3,43	0,0	0,0	2,73
300/ 9047	4,35	0,0	0,0	2,73
300/ 9048	4,39	0,0	0,0	2,73
300/ 9049	4,04	0,0	0,0	2,73

Values | Global extremes | Info | Calculation parameters

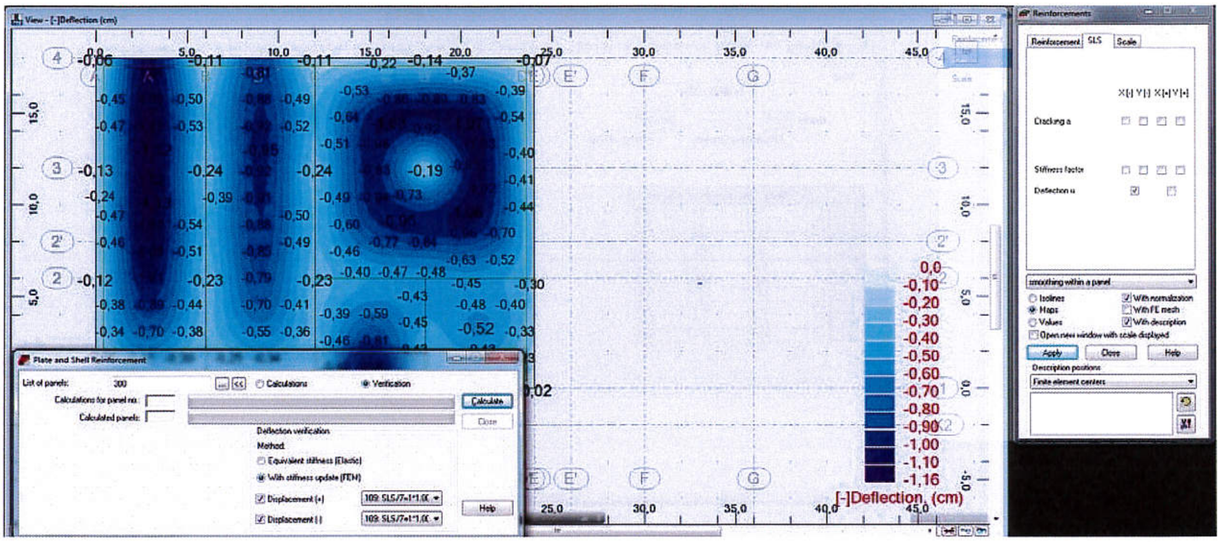
Once required reinforcement is calculated one can obtain displacement of designed elements (slab)

While reinforcement calculation displacement (with stiffness reduction resulted in element cracking) is estimated with use of simplified method (Equivalent stiffness- elastic). It is enough to estimate cracked slab total displacement.



More accurate results can be obtained with use of deflection **Verification with stiffness update.** Verification with stiffness update can be done after selection of critical (for deflection) SLS combination case. Usually one can choose combination with maximal static displacement for designed element.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



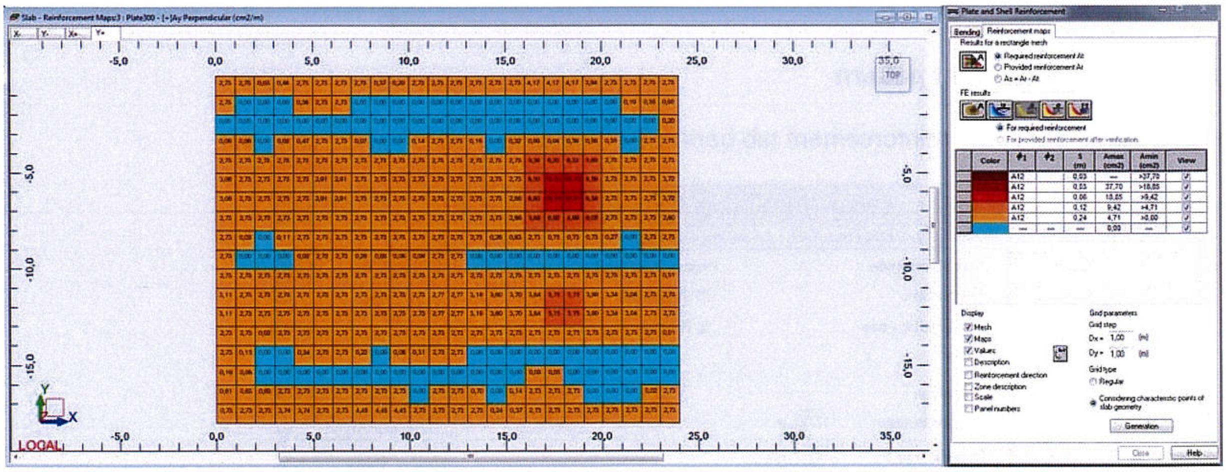
Reinforcement maps and reinforcement tables can be saved with use of Screen Capture option and then edited in Printout composition module.

File slab reinforcement-014.rtd

8.1.4. Provided reinforcement design

Provided slab reinforcement can be designed after required reinforcement was calculated. Panel (story 3 slab) **Design / Provided reinforcement of RC elements.**

The following plate view will be obtained



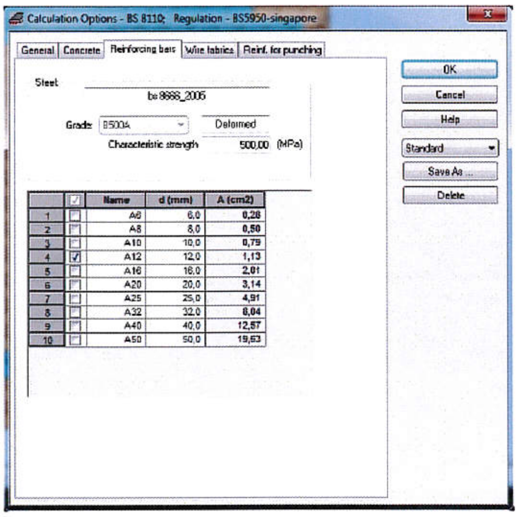
One can analyse reinforcement area displacement and cracking obtained in required reinforcement module. User can determine provided mesh diameter for provided reinforcement distribution.

Then like for bar elements one can define design parameters

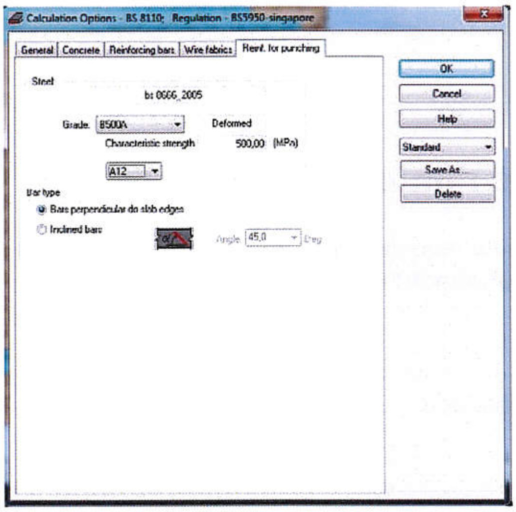
Analysis / Calculation option

Reinforcing bars tab - A12 can be chosen

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

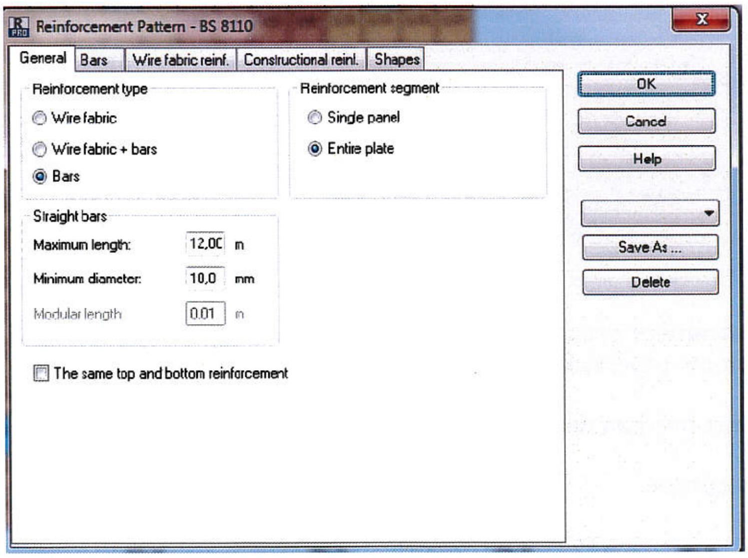


Reinforcement for punching tab - 12 can be chosen



Analysis / Reinforcement pattern

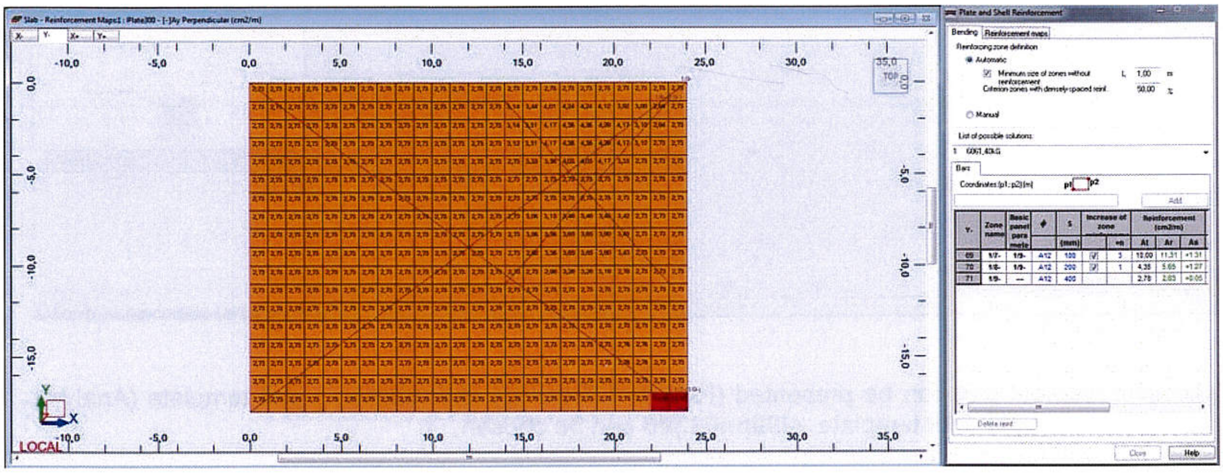
General tab - reinforcement tab bars



Then calculations can be launched **Analysis / Calculation**

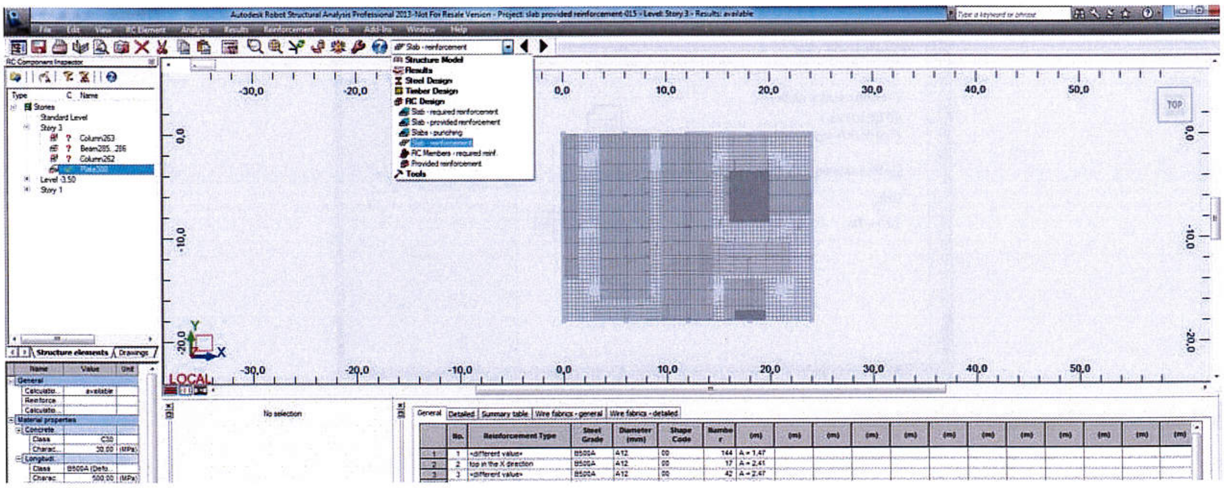
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Robot will present provided reinforcement distribution zones and bar spacing



Bottom reinforcement in Y direction

Slab provided reinforcement can be presented as well



Punching can be verified

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

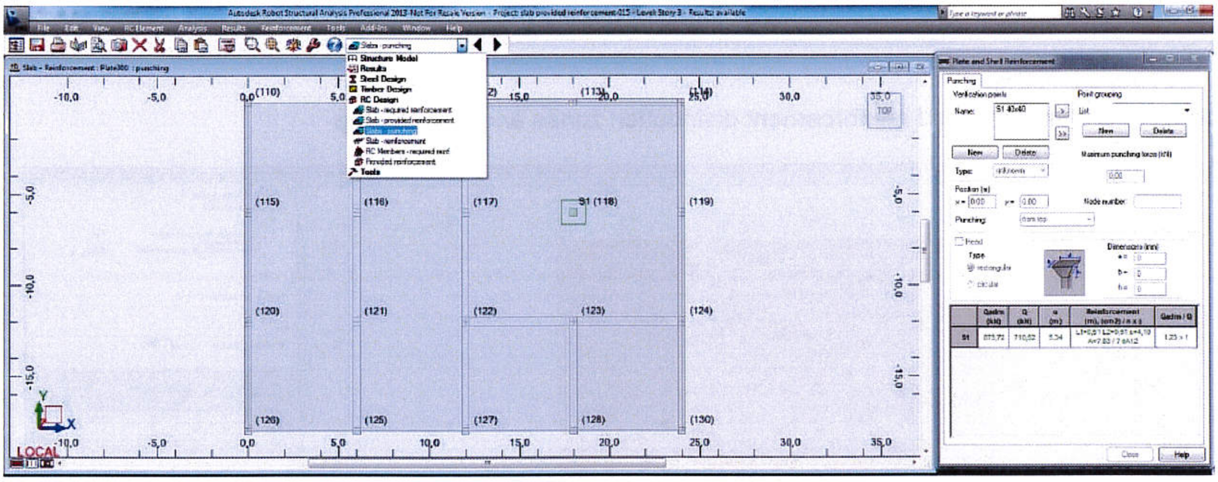
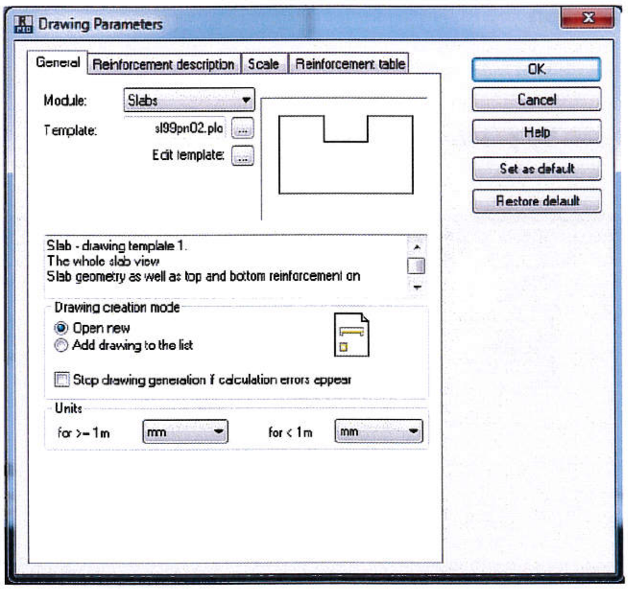
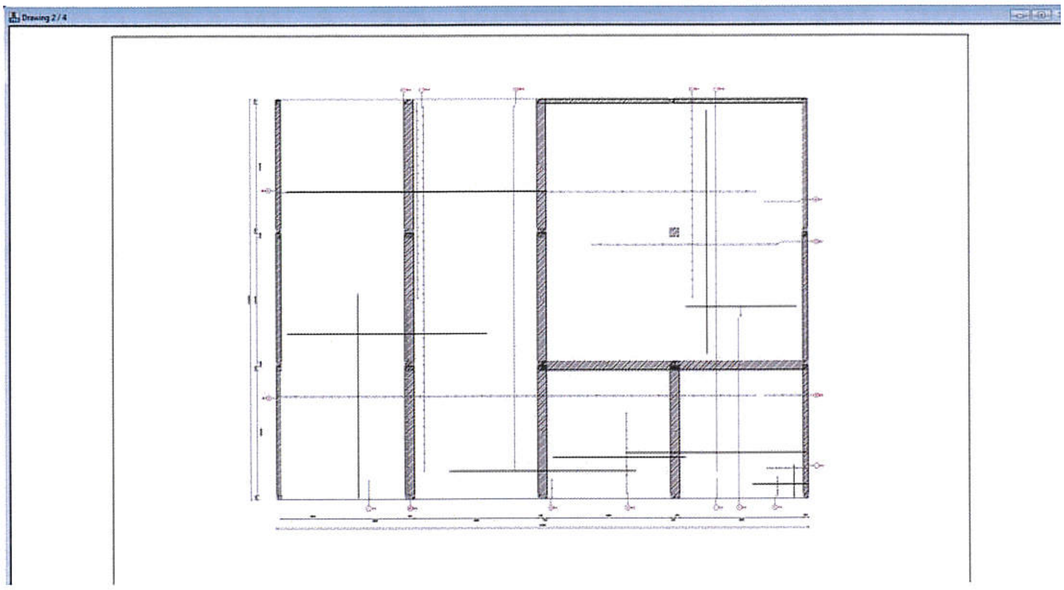


Plate reinforcement plot can be presented (Results / Drawing) after choosing plot template (Analysis /Drawing parameters). Plot template **sl99pn02.plo** can be chosen.

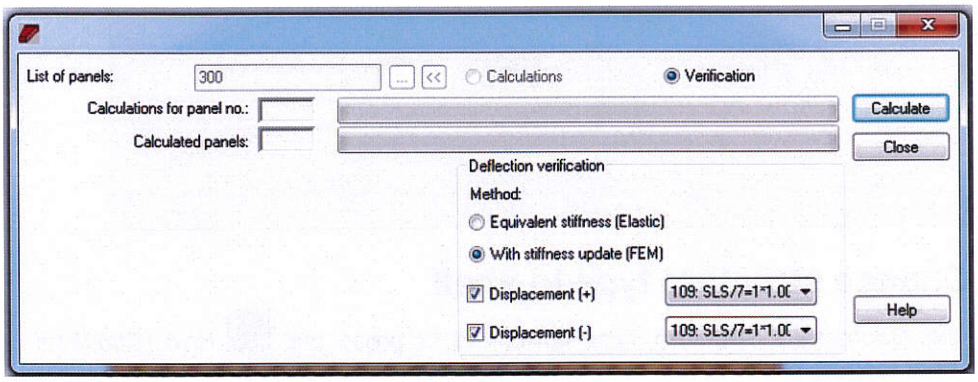


Final drawing (Results / drawings)

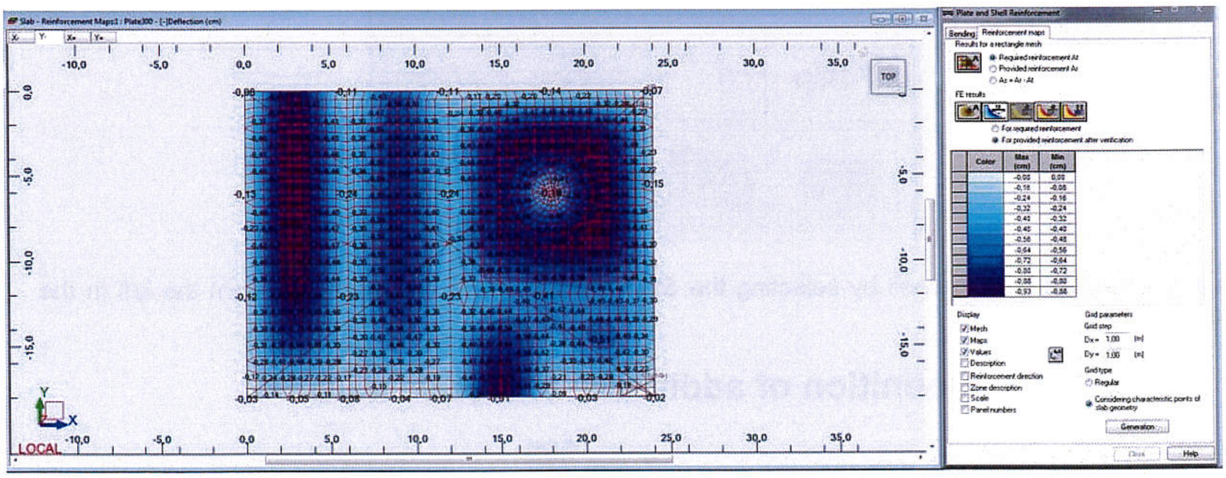
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



For provided reinforcement deflection verification can be done as well. Like for required reinforcement one can choose Verification with stiffness update for SLS combination with maximal static deflection (Analysis / Verification)



After calculation deflection map can be presented (should be smaller than obtained for required reinforcement)

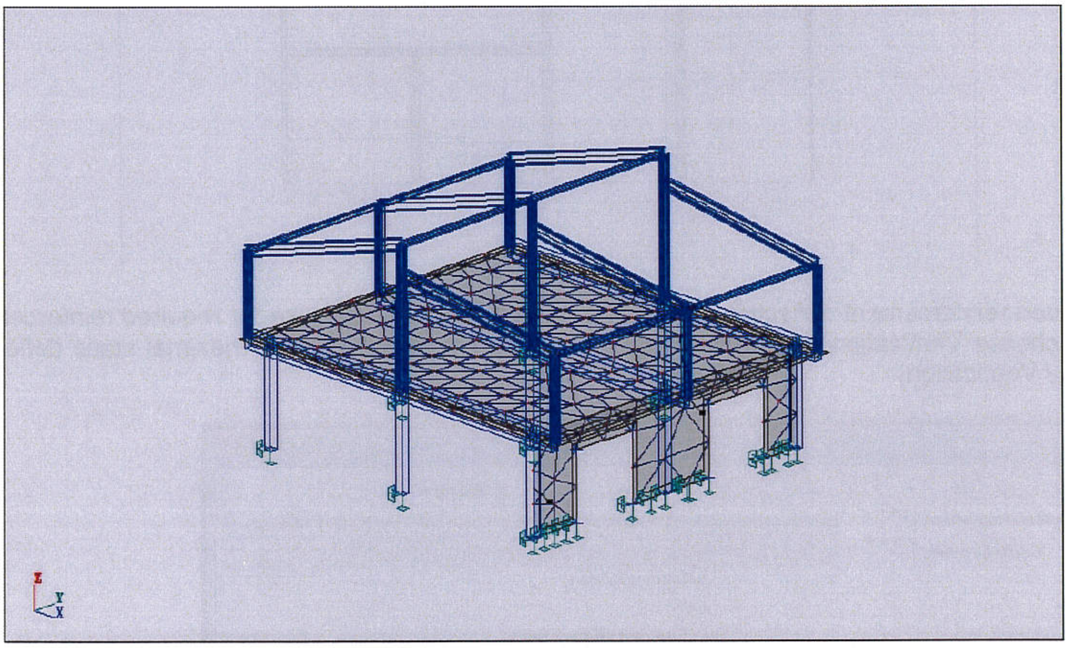


Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

EXERCISE NO. 9 "DEFINITION OF A PLATE-COLUMN STRUCTURE"

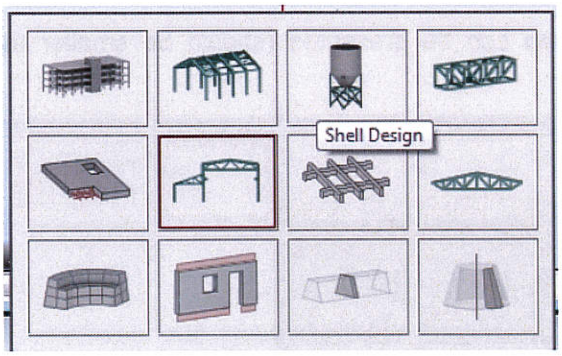
9.1. Structure geometry

Open the Exercise 1-loads project. Proposed scheme of the structure is displayed on the drawing below.



9.1.1. Change structure type to shell

From the menu select the *Geometry / Structure Type* command or press the icon (*Structure Definition*) located in the right toolbar, which opens the window presented on the drawing below.

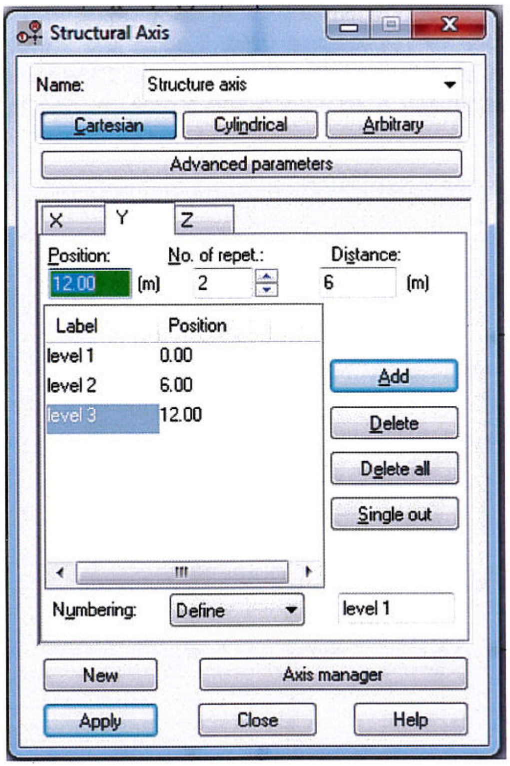


Change structure type to shell by selecting the *Shell Design* option (the third one from the left in the second row).

9.1.2. Definition of additional structure axis

In order to define additional axis click the *Axis Definition* icon, located in the right-hand toolbar or use the *Geometry / Axis Definition* command from the main menu, which open the ***Structural Axis*** dialog box.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



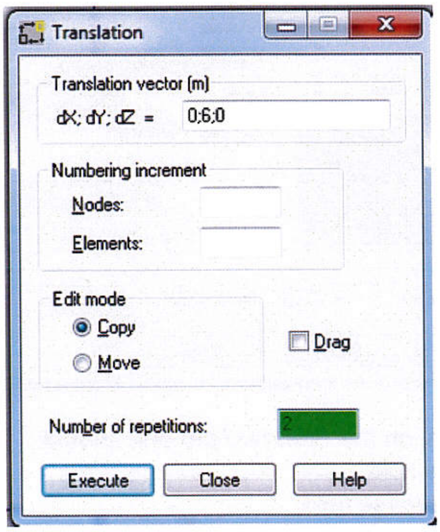
In the *Y* tab introduce the following values:

- 0 into the *Position* field
- 2 into the *No. of Repet.* field
- 6 into the *Distance* field

In the *Numbering* combo box select *Define* option end set the axis name for *Level 1*. Set it into the *Set of Created Axis* field by pressing **Insert** button. Confirm by clicking the **Apply** and **Close** buttons.

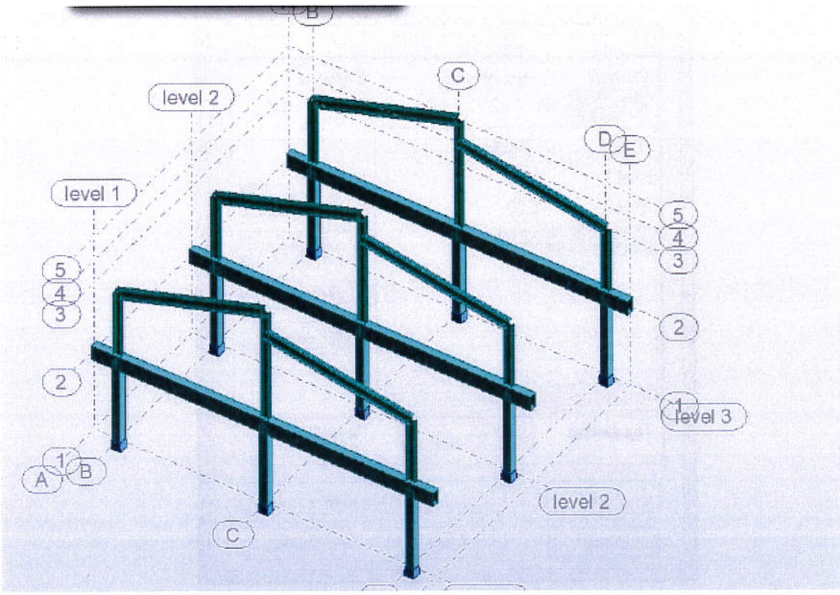
9.1.3. Copying the frame in the Y direction

Indicate the whole structure pressing the **Ctrl + A** button (selected bars will be highlighted into the red color). From the menu select the *Edit / Edit / Translate* command that opens the ***Translation*** dialog box presented on the drawing below.



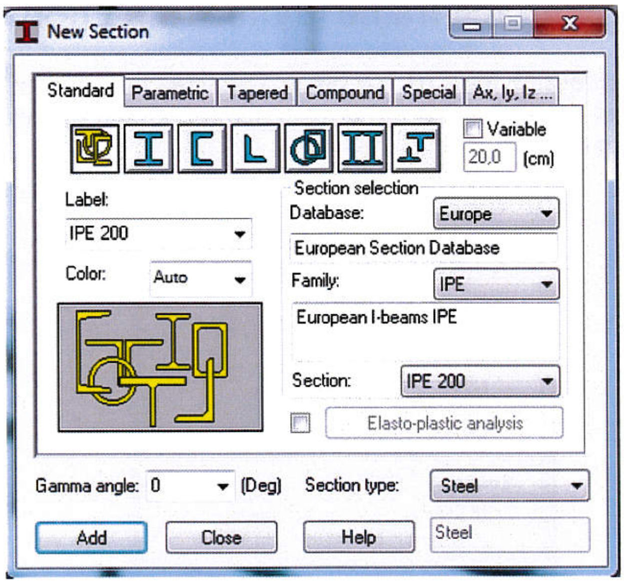
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

In the *dX; dY; dZ* edit field define the translation vector by typing in the coordinates of the translation vector: *0; 6; 0* (one may do it graphically by clicking on the beginning and end of the translation vector). Enter the value 2 in the *Number of repetitions* field. Leave the default values for the remaining parameters. Finish the operation by pressing the **Execute** and **Close** buttons. Select from the menu *View / Projection / 3d xyz*, which displays a 3d view of the structure (see the picture below).



9.1.4. Definition of additional steel beams between frames

Select the **BARS** layout from the available ***Autodesk Robot Structural Analysis*** layouts by clicking the list box in the top right corner (the screen will be divided into the *View* field, the ***Bars*** dialog box and the *Bars* table. If the UB 152x89x16 section is not present in the available section list, one should press the button ... located to the right side of the *Section* field. The ***New Section*** dialog box will be opened.



To define the *UB 152x89x16* section, on the *Standard* tab one should:

- set *Steel* in the *Section Type* field,
- set *Simple Catpro* in the *Database* field,
- set *UB* in the *Family* field,
- set *UB 152x89x16* in the *Section* field,

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

– end the operation by pressing **Add** and **Close** buttons

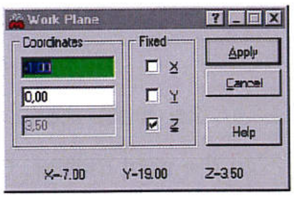
The newly defined section appears in the ***Bars*** dialog box.
To define bar element one should set *Bar Type* for *Beam* in the *Properties* field and select *UB 152x89x16* profile in the *Section* field. Then move the cursor to the graphic viewer, click with the left mouse key on the point depicting the beginning and end point of the bar element. These points may be defined as the coordinates of the intersection points of following structure axis:

Beginning of the beam:		End of the beam:	
Axis intersection	Coordinates	Axis intersection	Coordinates
- E, Level 1, 3	13.00; 0.00; 6.50	- E, Level 2, 3	13.00; 6.00; 6.50
- E, Level 2, 3	13.00; 6.00; 6.50	- E, Level 3, 3	13.00; 12.00; 6.50
- C, Level 1, 5	6.00; 0.00; 8.50	- C, Level 2, 5	6.00; 6.00; 8.50
- C, Level 2, 5	6.00; 6.00; 8.50	- C, Level 3, 5	6.00; 12.00; 8.50
- A, Level 1, 3	-1.00; 0.00; 6.50	- A, Level 2, 3	-1.00; 6.00; 6.50
- A, Level 2, 3	-1.00; 6.00; 6.50	- A, Level 3, 3	-1.00; 12.00; 6.50

9.2. Plate definition

9.2.1. Setting the work plane

In order to go to the plane where concrete beams are located select from the menu *View / Work in 3D / Global Work Plane* command, which opens the ***Work Plane*** dialog box.

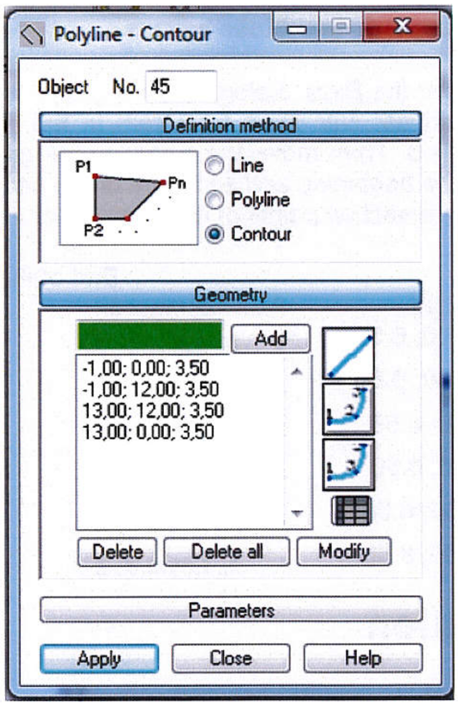


Move the cursor to the graphic viewer and select the left end point of the concrete beam. The coordinates in the ***Work Plane*** dialog box will change automatically to the selected one e.g. (-1.00,0.00,3.50). In the *Fixed* field switch on the *Z* option and then press the **Apply** button. Select the *View / Projection / Xy* command from the menu. Once this option is selected the structure is set on the XY plane at the recently defined Z – coordinate value (e.g. Z = 3.5) and only structure components from this plane will be displayed.

9.2.2. Definition of the plate contour

In order to define plate contour pick out from the menu *Geometry / Objects / Polyline-contour* command that opens the ***Polyline-Contour*** dialog box. In the *Definition Method* edit field select the *Contour* option. Place the cursor in the *Geometry* field indicate the required points of the contour in the graphic viewer. Closing the contour is effectuated by clicking the first of the defined points for the second time.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

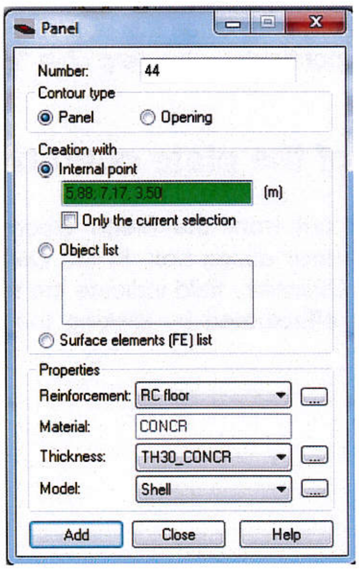


One may also defined the contour by introducing appropriate coordinates manually in the *Geometry* field.

9.2.3. Definition of the plate panel with required parameters

To define plate properties one should enter the main menu and select the *Geometry / Panels* command or click the icon (*Panels*) from the right toolbar. Once there appears the ***Panel*** dialog box, one should define the settings as defined below:

- *Contour Type:* check out the *Panel* option
- *Creation With:* check out the *Internal Point* option



Set the reinforcement type for *RC_floor* in the *Reinforcement* combo box. Select the thickness *TH300_CONCR* from the *Thickness* combo box in the ***Panel*** dialog box and then, set the cursor in the *Internal Point* field. Move the cursor to the graphical viewer and indicate once a point within the boundaries of the rectangular plate.

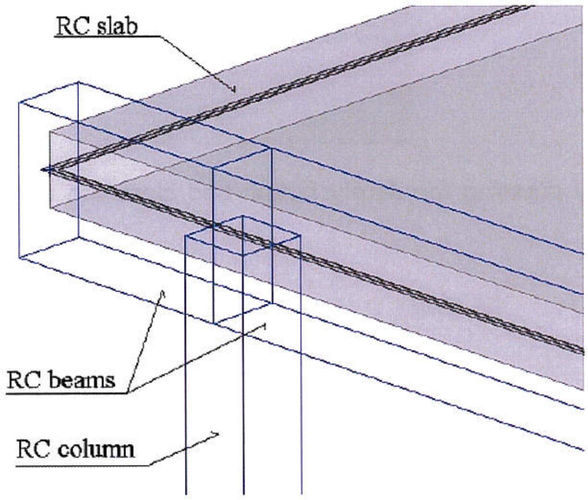
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

9.3. Definition of offsets in the beams

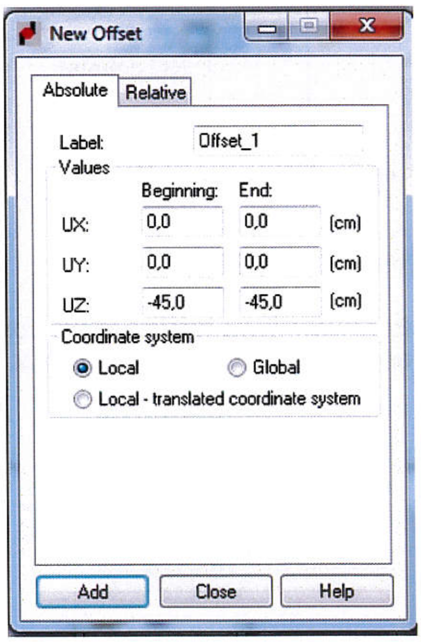
Select from the menu the *View / Display* command, which opens the ***Display*** dialog box, switch on following options:

- on the *Finite Elements* tab – *Thickness* option
- on the *Sections tab* – *Section - shape* option

Finish the operation pressing the **Apply** button and close the dialog box pressing the **OK** button. On the structure drawing, which appears on the screen one can see that axis of the RC beams and RC slab are on the same level. ***Autodesk Robot Structural Analysis*** allows one for defining of offsets within the structure.



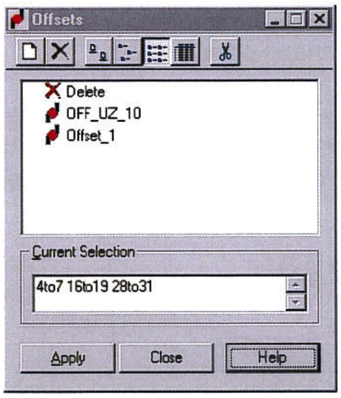
In order to define offsets select from the menu *Geometry / Additional Attributes / Offsets*, this opens the Offsets dialog box. From the upper toolbar select the icon (*New*). A dialog box for defining new offset will be open.



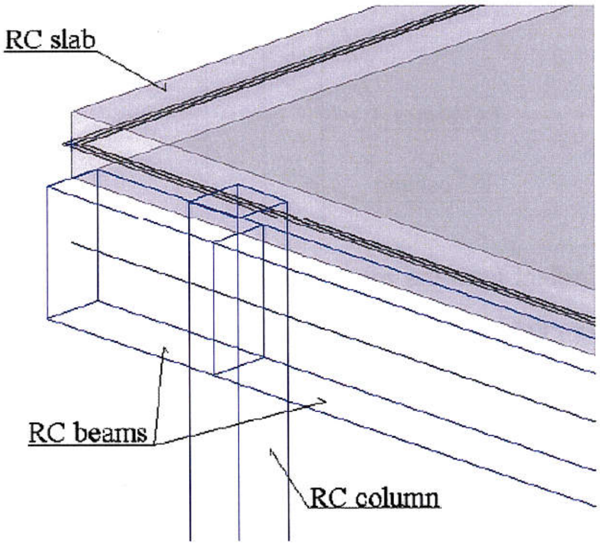
In the *Label* field enter the name *Offset_1* for new offset. Then in the *UZ* field introduce the offset value (in the exercise the offset value is equal 45 cm as it is half of RC slab height plus half of RC beams

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

height) at the beginning and at the end of the bar in the Z-axis direction. In the *Coordinate System* field activate the *Global* option. Once this option is selected, offset values will be given in the bar's global coordinate system. Finish the operation by pressing the **Add** button then close the dialog box by pushing the **Close** button. The recently defined offset is available in the active list of offsets. Place the cursor in the *Current Selection* field and type the numbers of bars (e.g. all concrete beams), to which offsets will be attributed.



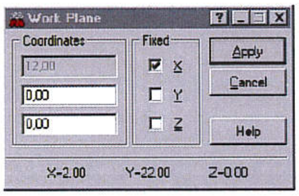
Finish the operation by pressing the **Apply** button and close the dialog box by selecting the **Close** button.



9.4. Definition of the front wall

9.4.1. Setting the work plane

In order to go to the plane where the front wall will be located select from the menu *View / Work in 3D / Global Work Plane* command, which opens the ***Work Plane*** dialog box.

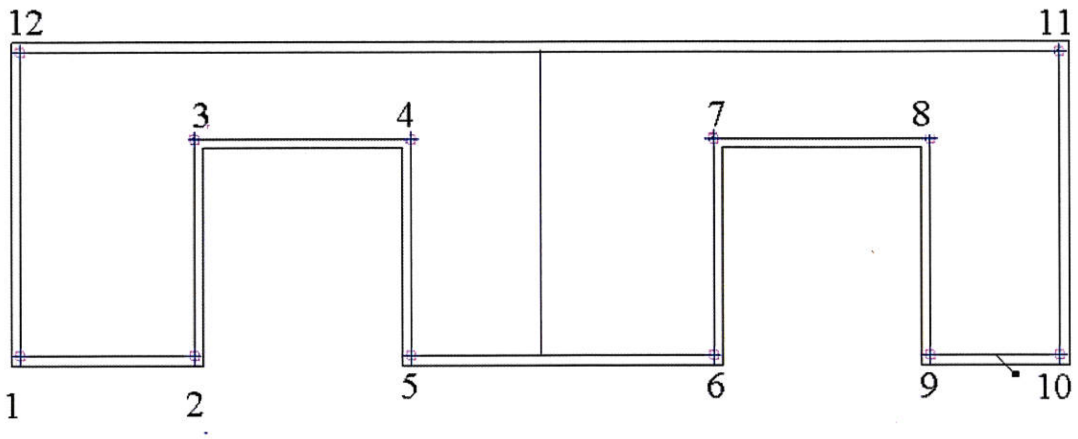


Move the cursor to the graphic viewer and select the end point of the right column. The coordinates in the ***Work Plane*** dialog box will change automatically to the selected one e.g. (12.00,0.00,0.00). In the *Fixed* field switch on the *X* option and then press the **Apply** button. Select the *View / Projection / Yz*

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

command from the menu. Once this option is selected the structure is set on the YZ plane at the recently defined X – coordinate value (e.g. Z = 12) and only structure components from this plane will be displayed.

9.4.2. Definition of the contour of the front wall with openings



Select from the menu *Geometry / Objects / Polyline-contour* command that opens the ***Polyline-Contour*** dialog box. In the *Definition Method* edit field select the *Contour* option. Place the cursor in the *Geometry* field and introduce coordinates of the points defining the appropriate contour:

Point no.	Coordinates:
1	(12.00;0.00;0.00)
2	(12.00;2.00;0.00)
3	(12.00;2.00;2.50)
4	(12.00;4.50;2.50)
5	(12.00;4.50;0.00)
6	(12.00;8.00;0.00)
7	(12.00;8.00;2.50)
8	(12.00;10.50;2.50)
9	(12.00;10.50;0.00)
10	(12.00;12.00;0.00)
11	(12.00;12.00;3.50)
12	(12.00;0.00;3.50)

Press the **Close** button which closes the ***Polyline – Contour*** dialog box.

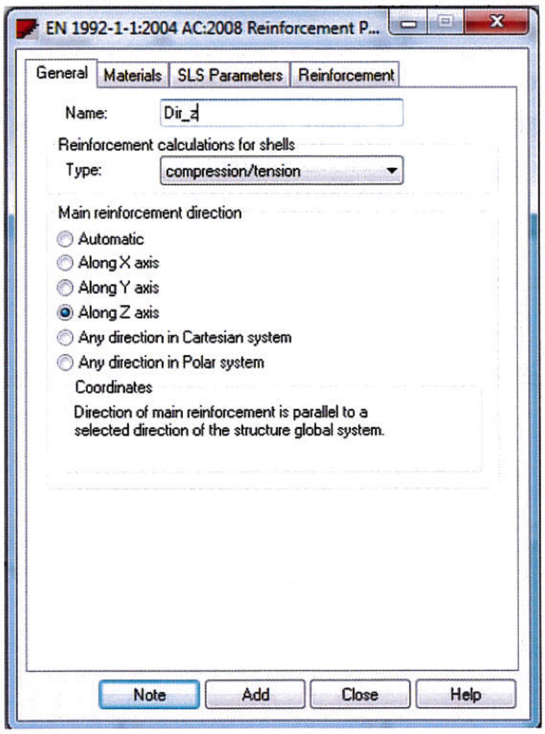
9.4.3. Definition of the panel to be applied to the front wall

To define panel properties one should select from the menu the *Geometry / Panels* command or click the icon (*Panels*) from the right toolbar, which opens the ***Panel*** dialog box. One should define the settings as defined below:

- *Contour Type:* check out the *Panel* option
- *Creation With:* check out the *Internal Point* option
- *Reinforcement:* check out the *Dir_Z* option

If the *Dir_Z* reinforcement type is not available in the reinforcement type list, press the button located to the right of the *Reinforcement* list containing all reinforcement types defined to date. The ***Reinforcement Parameters*** dialog box will be opened. On the *General* tab define settings as presented on the picture below.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Others settings leave as default.

Define new thickness by clicking the button located to the right of the *Thickness* field that opens the *New* ***Thickness*** **dialog** box. In the *Th* field type *25*, this is a new value for panel thickness. Then in the *Label* field enter new name *TH25_CONCR*. Confirm the settings by clicking the **Add** button and close the ***New Thickness*** dialog box by clicking the **Close** button.
Set the reinforcement type for *Dir_Z* in the *Reinforcement* combo box.
Select the thickness *TH25_CONCR* from the *Thickness* combo box in the ***Panel*** dialog box and then, set the cursor in the *Internal Point* field. Move the cursor to the graphical viewer and indicate once the point within the boundaries of the recently defined contour.

9.4.4. Definition of supports on wall edges

In order to define supports select from the menu the *Geometry / Supports* command or press the icon (*Supports*) from the right toolbar. The ***Supports*** dialog box will be opened. From the list of active supports check out the *fixed* support type (recently selected support type will be highlighted). Then in the *Current Selection* field activate the *Line* option. Set the cursor in the graphic viewer and click at the appropriate edge when it is highlighted.

9.5. Loads

9.5.1. Modification of self-weight load

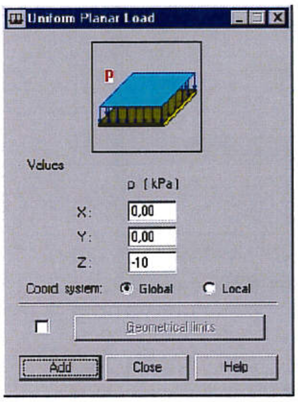
Select from the menu the *Loads / Load Table* option, which results in opening the *Loads – Case* table. Once there appears the table, select *DL1* load case, set the cursor in the third record in the loads table (List column). There will appear a list of all elements of the structure and it means that the self – weight load will be applied to whole the structure.

9.5.2. Definition of additional loads to be applied to the plate

In order to define of a new case select from the menu *Loads / Load Types*. In the ***Load Types*** dialog box select the *live* nature in the *Nature* field and press the **New** button. It results in appearance a new

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

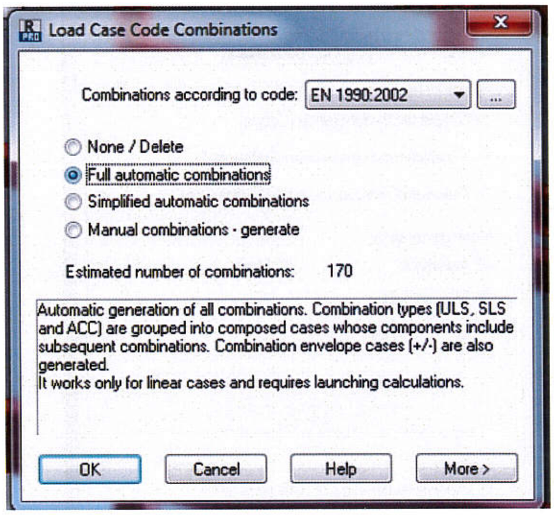
load case (*LL3*) in the list of defined cases. Select the recently define load case from the available load list. Press the icon (*Load Definition*), which opens the ***Load Definition*** dialog box. On the *Surface* tab click the *Uniform Planar Load* icon and set the parameter as presented on the picture below.



Confirm the settings pressing the **Add** button. In the *Apply To* field enter the number defining the plate to which the load will be applied. Finish the operation pressing the **Apply** button and then close the ***Load Definition*** dialog box pressing the **Close** button.

9.5.3. Definition of combinations

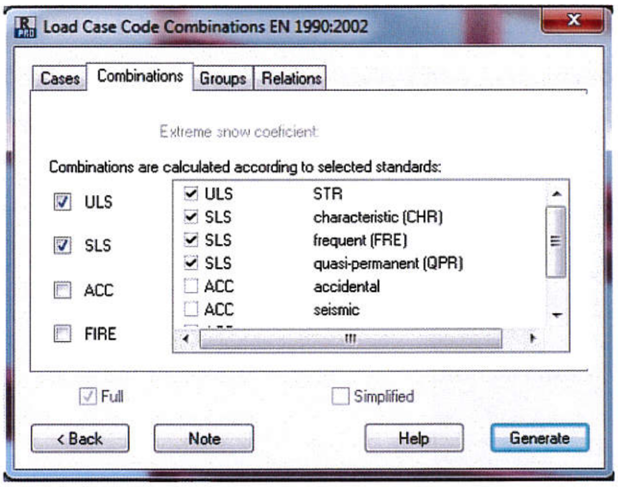
Select from the menu option *Loads / Automatic Combinations* that opens the **Load Case Code *Combination*** dialog box. Leave all parameters default and press the **OK** button. The ***Combinations*** dialog box will be open.



This option allows for definition and modification of code combinations.
To define load cases for combination:

- select Full automatic combinations
- select required combination types

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

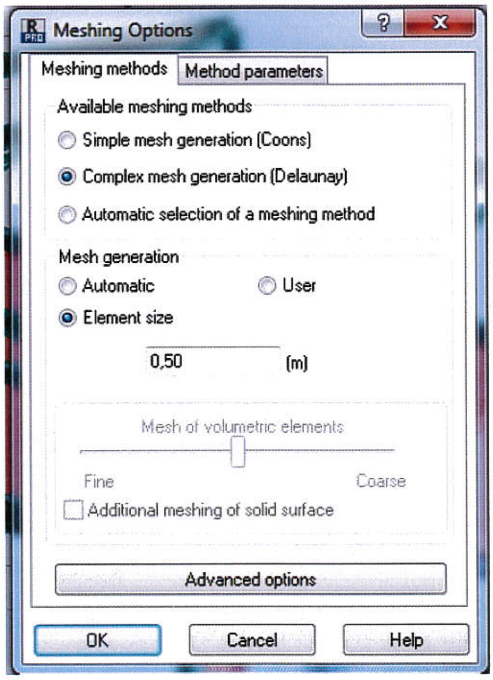


– click Generate.

Note: *for all natures, default factors will be applied.*

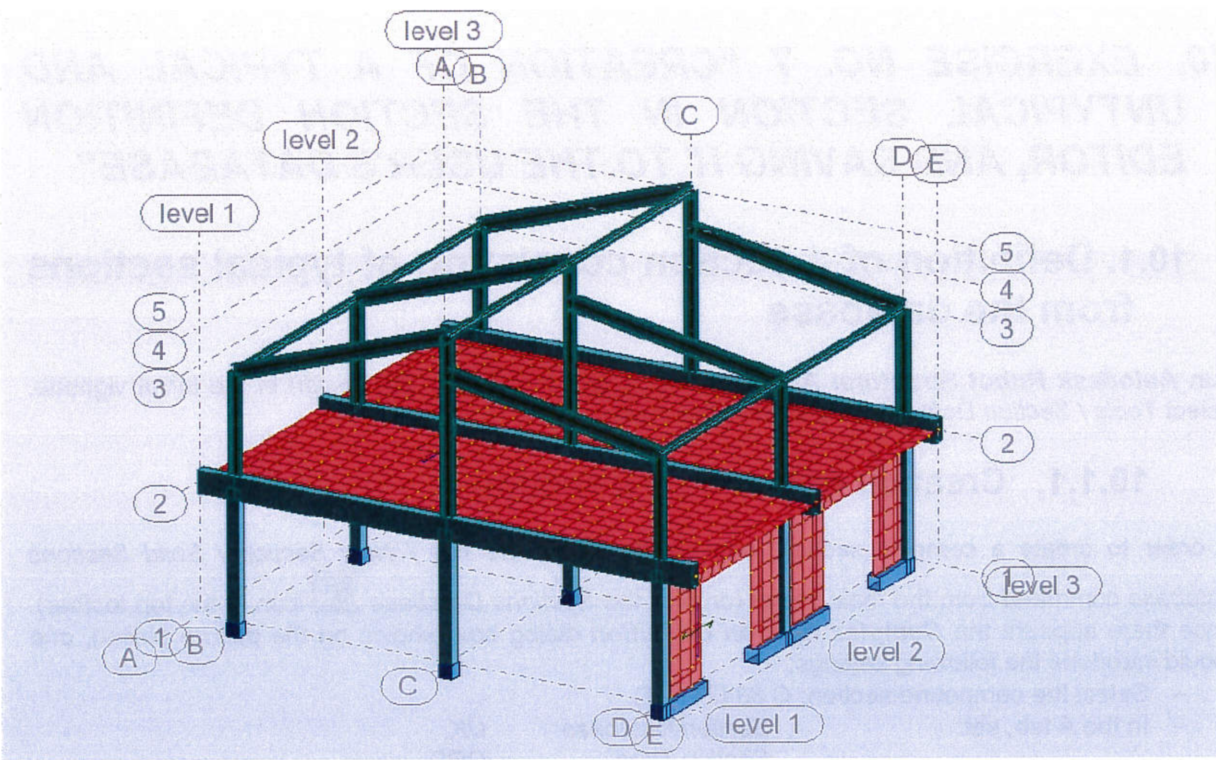
9.6. Definition of meshing options

In order to change meshing options select panels and then from the menu **Analysis /Meshing/Meshing options** open the ***Meshing Options*** dialog box. In the *Mesh Generation* field - with the *Delaunay* method switched on in the *Available Meshing Methods* field - activate ◉ *Element size option* (the FE mesh will be generated automatically using the current parameters) and enter 0.5 in the *Element size field*. Other parameters leave as default. Finish the operation pressing the **OK** button.



Select from the menu the **Analysis /Meshing/Local Mesh generation** command. If this option is selected, ***Autodesk Robot Structural Analysis*** generates the calculation model of a structure (finite elements).

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



To change meshing parameters for one panel (e.g. vertical) chose vertical panel and then select from the menu: *Analysis / Meshing / Meshing Options*, which opens the ***Meshing Options*** dialog box. In the *Available Meshing Methods* select the *Delaunay* method, switch to the *Mesh Generation* field and select the *Automatic* option. In the *Division 1* field enter 15 and press the **OK** button, which closes the ***Meshing Options*** dialog box. Select from the upper menu command: *Analysis / Calculation Model / Local Mesh Generation*.

9.7. Calculating the structure

Start the calculations by pressing the *Calculations* icon from the upper toolbar or by selecting the command *Analysis / Calculations* from the menu.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

10. EXERCISE NO. 7 "CREATION OF A TYPICAL AND UNTYPICAL SECTION IN THE SECTION DEFINITION EDITOR, AND SAVING IT TO THE USER'S DATABASE"

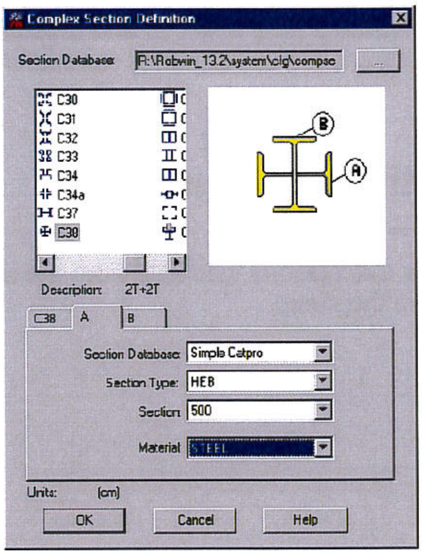
10.1. Definition of a section consisting of typical sections from the database

Run ***Autodesk Robot Structural Analysis*** then select the *Frame 2D Design* in the initial vignette. Select *Tools / Section Definition* option from the main menu.

10.1.1. Creation of a complex section

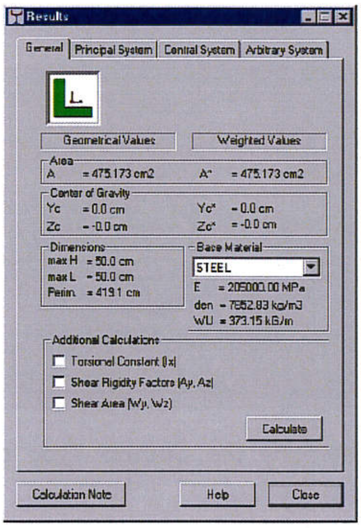
In order to create a complex section, one should select the *File / New Section / Solid Sections Database* command from the main menu (or click the *Sections Database* icon in the top toolbar). Once there appears the ***Complex Section Definition*** dialog box (shown on the picture below), one should introduce the following settings:

- Select the compound section: *C 38* (2T+2T).
- In the *A* tab, set: *Section Database:* *UK,*
  Section Type: *CRS,*
  Section: *381x114x37.2*
  Material: *STEEL,*
- Set the same parameters in the *B* tab.



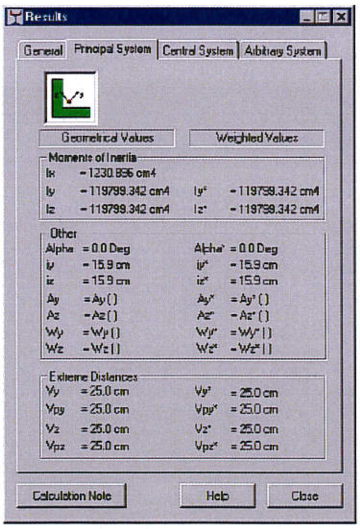
Confirm by pressing **OK**. A section consisting of two I-sections will appear in the graphical viewer. In order to generate the results, choose both contours (**CTRL+A**) and then select *Results / Geometric Properties / Results* command from the main menu (or click the *Results* icon located in the right-hand toolbar). Once the calculations are finished, there appears the ***Results*** dialog box (see the picture below), presenting information about the generated section in the following four tabs: *General, Principal System, Central System,* and *Arbitrary System.*

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



10.1.2. Creation of a single section

In order to create a new section contour which will constitute the envelope of all the contours, choose both contours (**CTRL+A**) and then select the *Contour / Standardization of Overlapping Contours* command from the main menu (or click the *Standardization* icon in the right-hand toolbar). In purpose to calculate the parameters for torsion, one should check out the *Torsional Constant (Ix)* option located in the *Additional Calculations* field and then press the **Calculate** button situated in the bottom part of the mentioned field in the *General* tab of the ***Results*** dialog box. Once the calculations are finished program automatically switches to the *Principal System* tab on the ***Results*** dialog box.



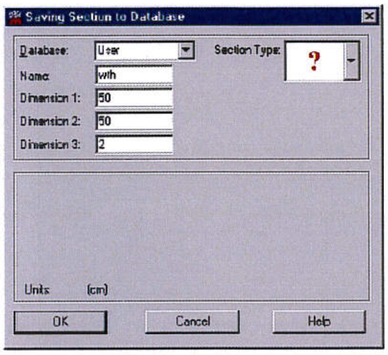
Click the **Calculation Note** button in the bottom part of the ***Results*** dialog box (or select the *Results / Geometric Properties / Calculation Note* command from the main menu) to generate a calculation note. To save the section in the user's database: one should select the *File / Save to Databases* command from the main menu (or click the icon, located in the top toolbar). There appears the ***Saving Section to Database*** dialog box (shown below), where one should describe the defined section. The proposed description runs as follows:

- Set the database: *User*,
- In the *Name* field: ***with***,
- In the *Dimension 1* field: 50,
- In the *Dimension 2* field: 50,
- In the *Dimension 3* field: 2,

Note: *digits are not allowed in the name definition.*

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Having been accepted by means of the **OK** button, the section is saved to the database.



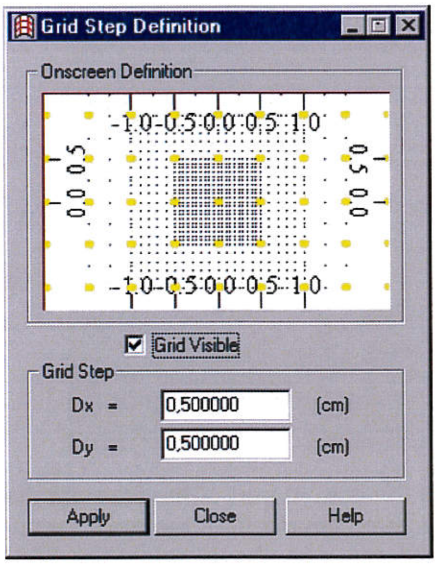
Note: *It is necessary to determine the Section Type parameter if there is a need for designing the section.*

10.2. Definition of a user's sections (solid)

In order to create a solid section (pipe with diameter = 10 cm and thickness = 0,5 cm) one should select the *File / New Section / Solid* command from the main menu or click the *New solid section* icon in the top toolbar.

10.2.1. Definition of the grid step

Select the *View / Grid Step* command from the main menu, which opens the ***Grid Step Definition*** dialog box. In the *Dx, Dy* fields, introduce *0.5* value. Confirm with the **Apply** button, close the dialog box by pressing **Close**.

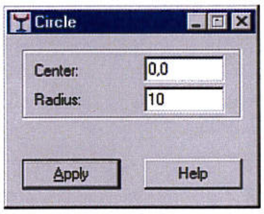


10.2.2. Definition of the external circle

Select the *Contour / Circle* command from the main menu (or the *Circle* icon from the right-hand toolbar). Once there appears the ***Circle*** dialog box, set the following parameters:

- In the *Center* field: 0; 0,
- In the *Point* field: 10,
- Confirm by pressing **Apply**.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

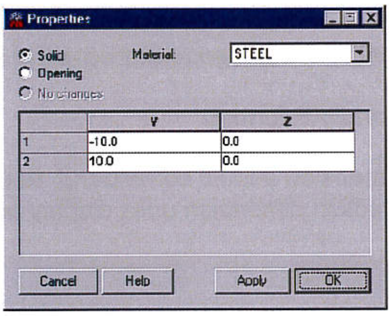


10.2.3. Definition of the internal circle

Introduce the value 9.5 into the *Radius* field in the ***Circle*** dialog box, confirm by pressing the **Apply** button, and close the dialog box by pressing .

10.2.4. Definition of the material to be applied to the contour

Select the external contour by clicking it (contour changes its color). Then, enter into the main menu and select the *Contour / Properties* command. The opened dialog box (see the picture below) allows one to change the material to be applied to the contour.



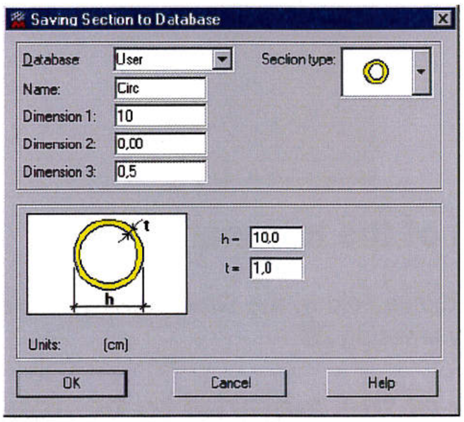
The ***Properties*** dialog box is also accessible from the context menu (activated by the right mouse button).

10.2.5. Results analysis

Select the *Results / Geometric Properties / Results* command from the main menu or click the *Results* icon in the right-hand toolbar. Once the calculations are finished, there appears the ***Results*** dialog box, presenting information on the generated section in the following four tabs: *General, Principal System, Central System* and *Arbitrary System*. In order to calculate the parameters for torsion, one should check out the *Torsion Constant (Ix)* option located in the *Additional Calculations* field and then press the **Calculate** button situated in the *Additional Calculations* field in the *General* tab of the ***Results*** dialog box. Once the calculations are finished, there will not appear the calculated value of the torsional constant in the ***Moments of Inertia*** dialog box on the *Principal System* tab, for the section thickness is too small. Click the **Calculation Note** button in the bottom part of the ***Results*** dialog box (or select the *Results / Geometric Properties / Calculation Note* command from the text menu) to generate a calculation note. To save the section in the user's database: one should select the *File / Save to Databases* command from the main menu (or click the icon, located in the top toolbar). There appears the ***Section Selection*** dialog box (shown below), where one should describe the defined section. The proposed description runs as follows:

– Set the database:	*User,*
– In the *Name* field:	*Circ*
– In the *Dimension 1* field:	*10*
– In the *Dimension 2* field:	*0.5*
– In the *Dimension 3* field:	(leave it empty)
– In the *Section Type* field:	select the circle symbol after opening the box and introduce the relevant values (below).

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



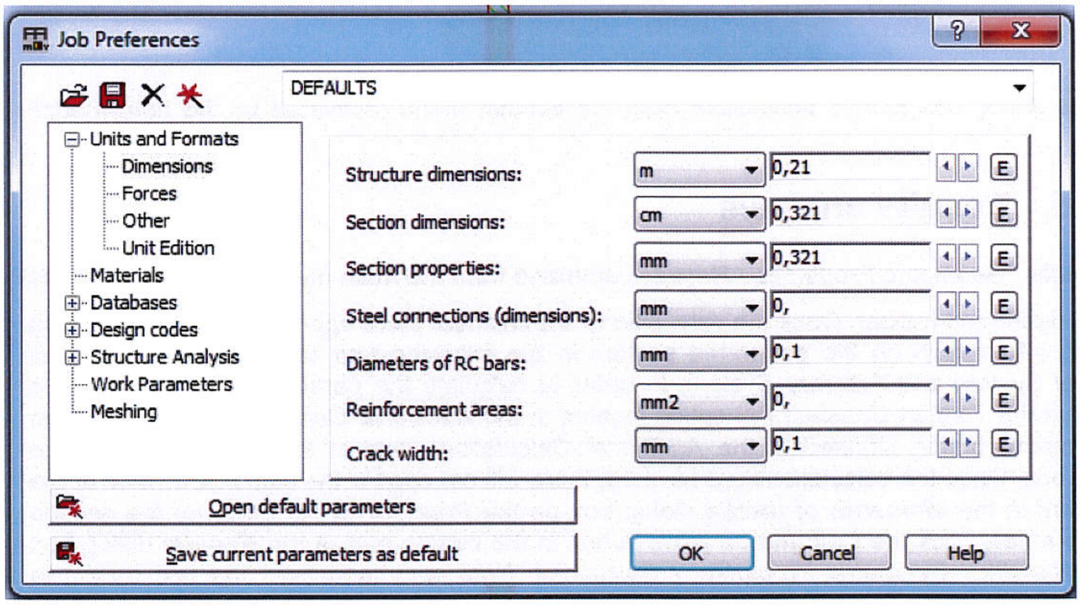
Confirm the operation clicking the **OK** button, the section is saved to the database.

10.2.6. Section shape import form DXF file

Section definition module allows importing section geometry saved in DXF format.

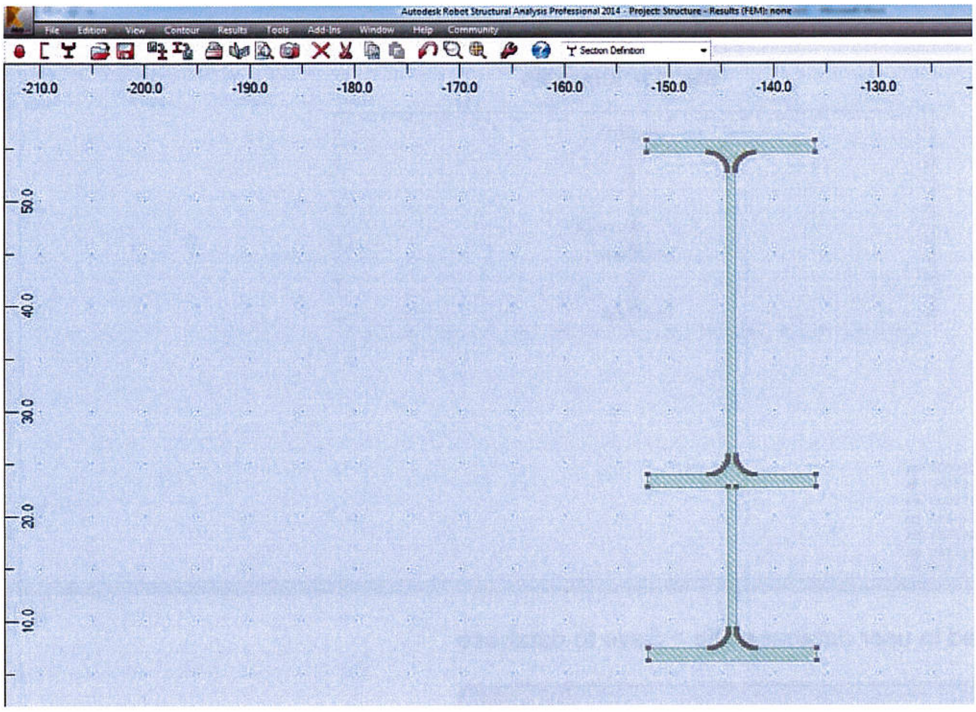
DXF file can be opened by File > Import DXF format.

Units in Job Preferences > Section dimension should correspond to units used in DXF file (in our case cm). It is recommended to increase Section dimension units display accuracy as well

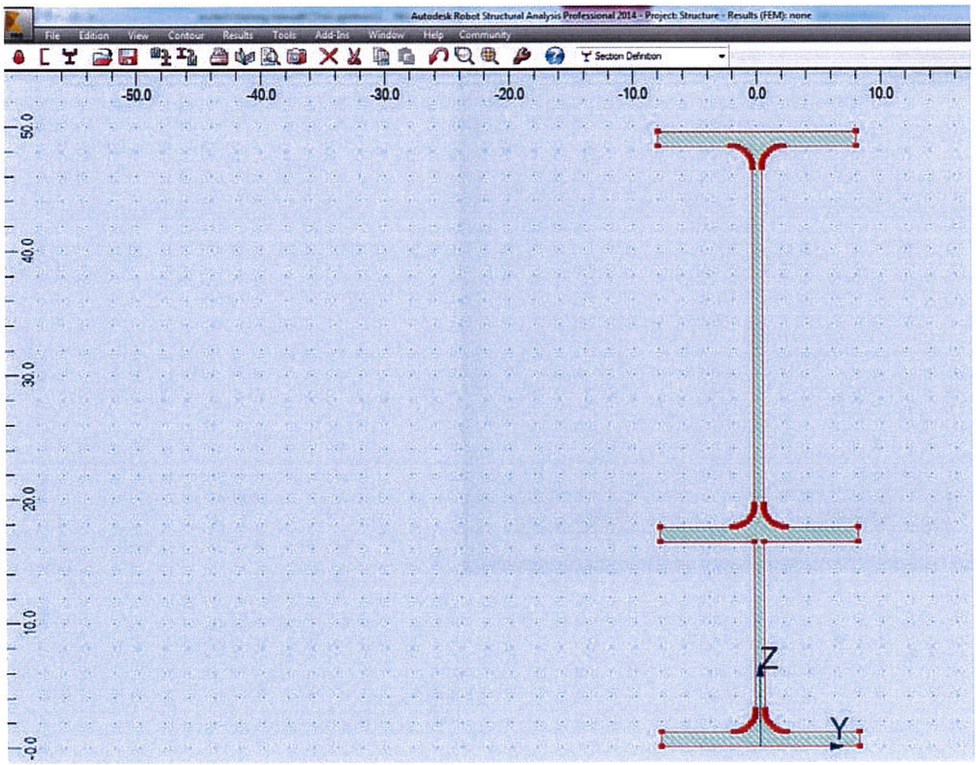


Using File > Import DXF format. We can import modified IPE 330 section

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

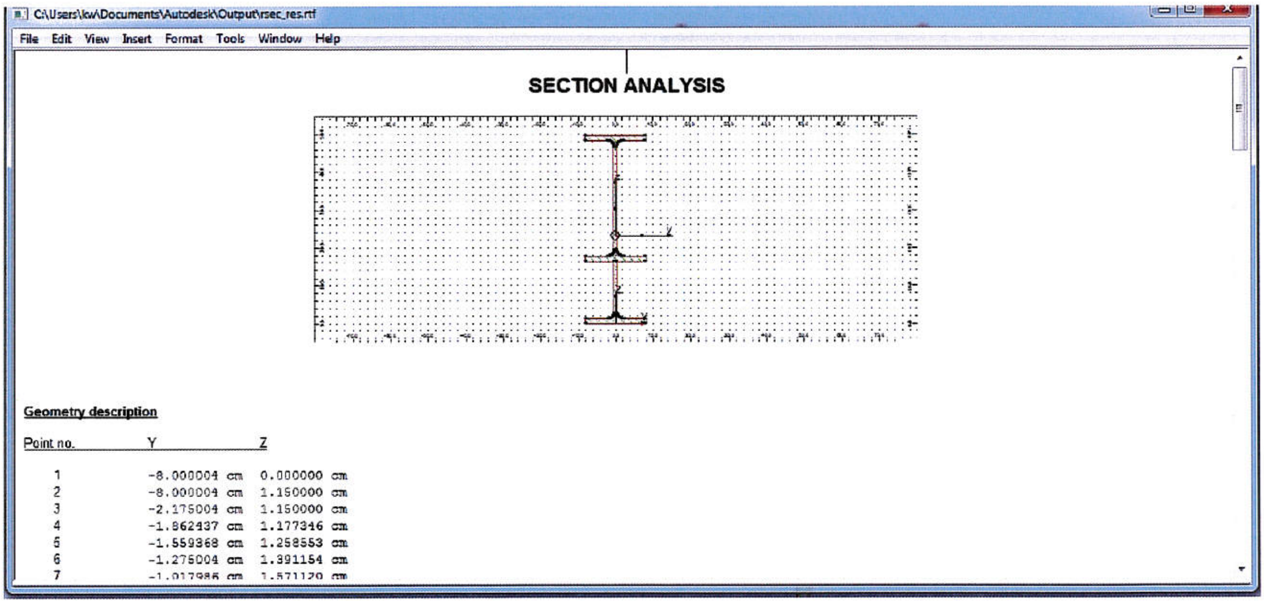


Section geometry should be placed in the coordinate system center. It can be done by selecting contour and using Edition > Translate option

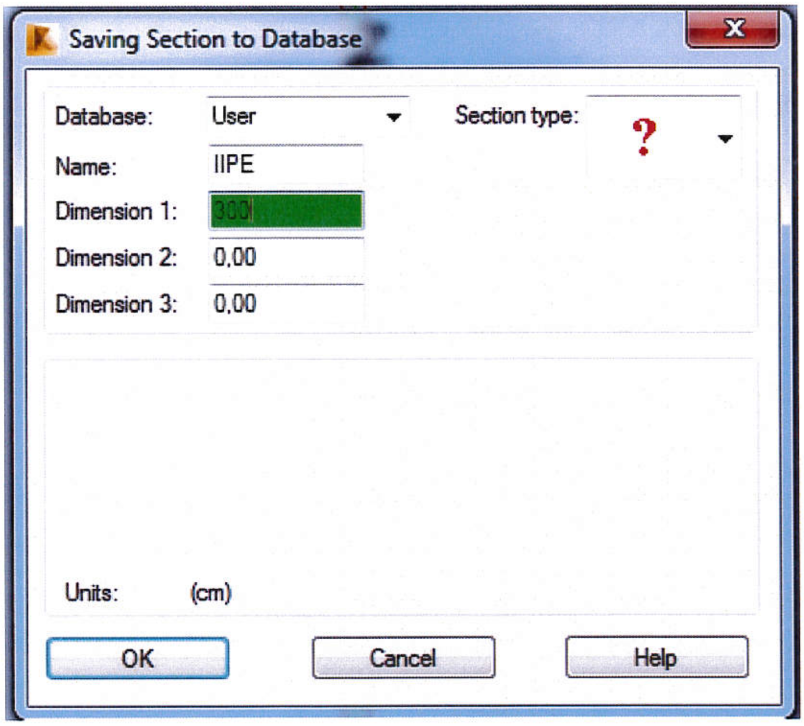


Then section properties can be calculated

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



and section can be saved in user database File > Save to database



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

10.2.7. Saving to the database

Select the *File / Save to Databases* command from the main menu (or press the icon in the top toolbar). There appears the ***Section Selection*** dialog box (see below), where one should describe the defined section. Proposed description:

- Set the database: *User*,
- In the *Name* field: *Z*,
- In the *Dimension 1* field: introduce the characteristic dimension
- In the *Dimension 2* field: introduce the characteristic dimension
- In the *Dimension 3* field: introduce the characteristic dimension

Having been accepted by means of the **OK** button, the section is saved to the database.

Note: *When a thin-walled section is used, it is treated during calculations as a solid section with the sectional properties disregarded.*

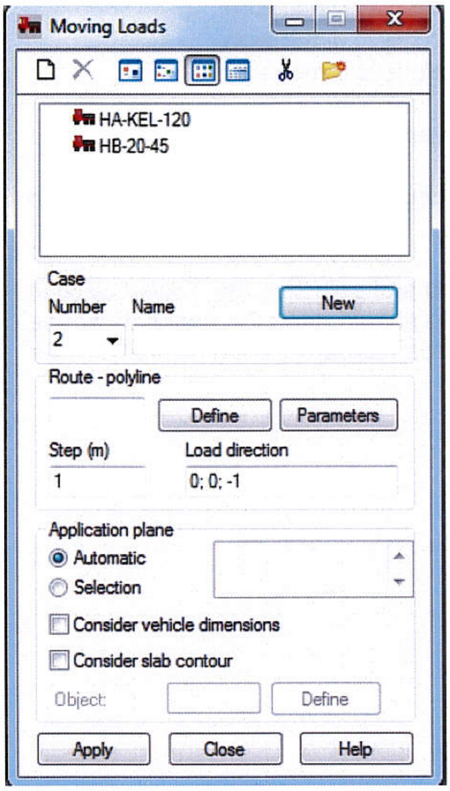
One may look through the saved sections by entering the main menu and selecting the *Tools / Section Database* command. Once there appears the section dialog box, one should open the user's database *Ruserpro (File / Open existing / Ruserpro)*. The box located in the top toolbar (first on the left) allows one to select the saved sections.

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

11. EXERCISE NO. 11 "MOVING LOAD DEFINITION"

11.1. Open file 18-moving load

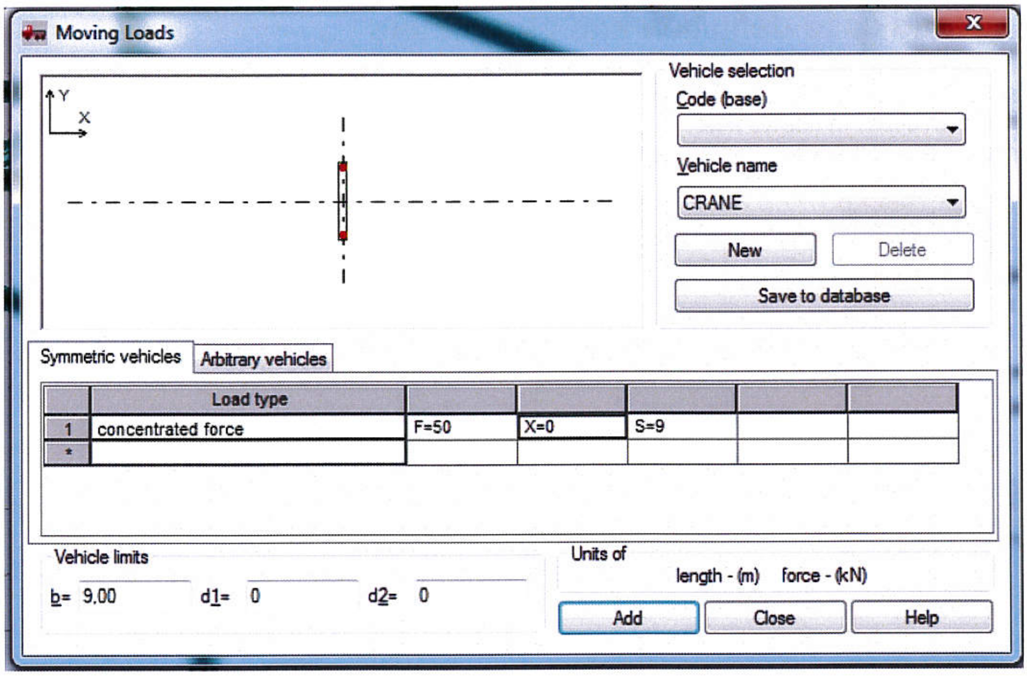
In ***Autodesk Robot Structural Analysis*** we can use special load type for moving load definition.
It can be done by use of Load > Special load > Moving load option. User can define set of arbitrary forces which will be treated as coming from vehicle. Then vehicle route can be defined together with load step



11.2. Vehicle definition

After opening of Load > Special load > Moving load option select New vehicle icon then New
Define new vehicle name CRANE
In the symmetric vehicle tab define one should select Load type > Concentrated force the define Q=50 and S=9

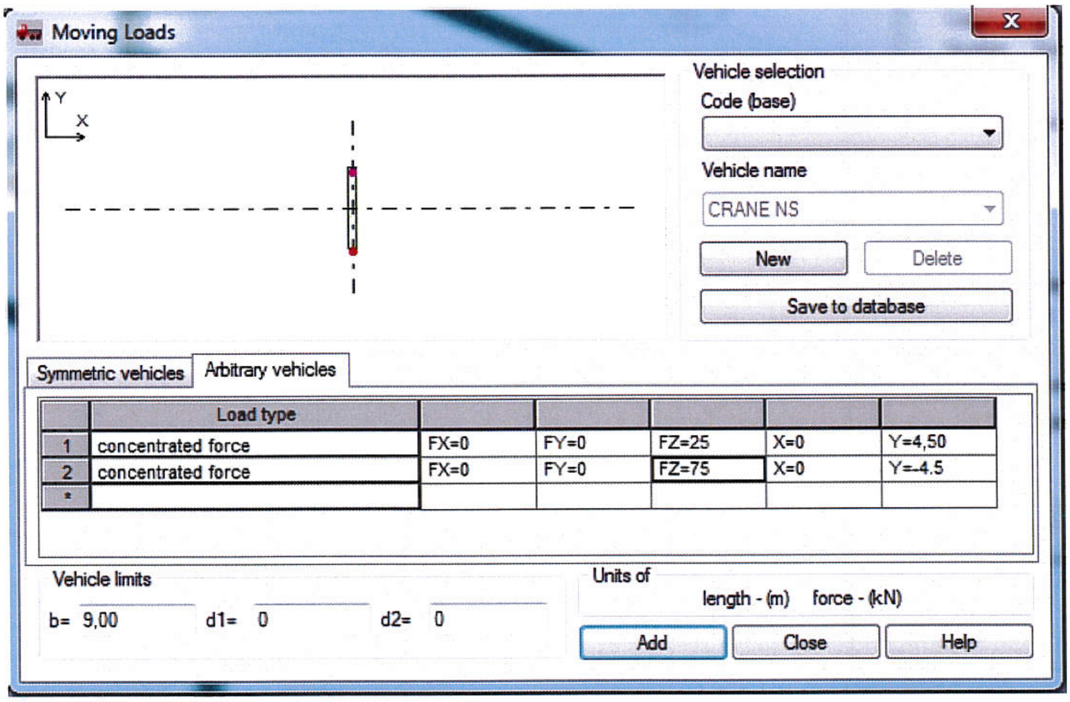
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



Then Add option should be used to save new vehicle in database

In very smilar way another type of vehicle Non Symmetrical crane can be defined

After opening of Load > Special load > Moving load option select New vehicle icon  then New.
Define new vehicle name CRANE NS
In the arbitrary vehicle tab define one should select Load type > Concentrated force then define
FZ=25 and Y=4,5
FZ=75 and Y=-4,5

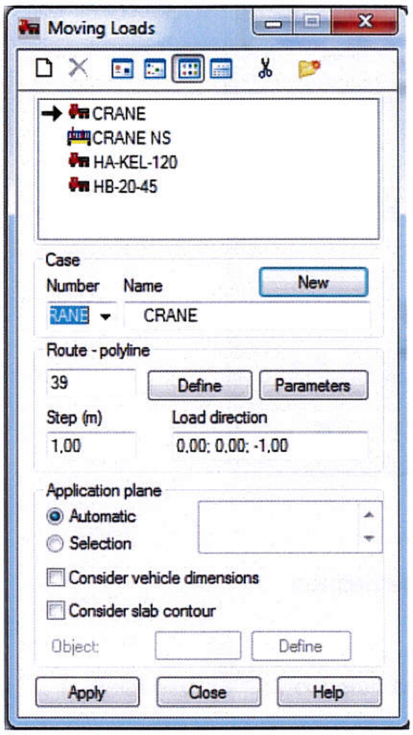


Then Add option should be used to save new vehicle in database

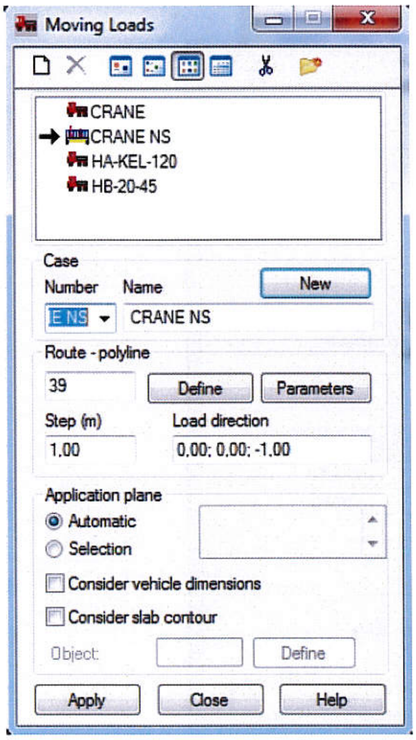
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

11.3. Moving load case definition and parameters

In Moving Loads dialog box Select vehicle CRANE (active vehicle should be indicated by arrow).
Type in load case name CRANE in Name field.
Graphically define route between nodes 1 and 2 using Define option then Apply



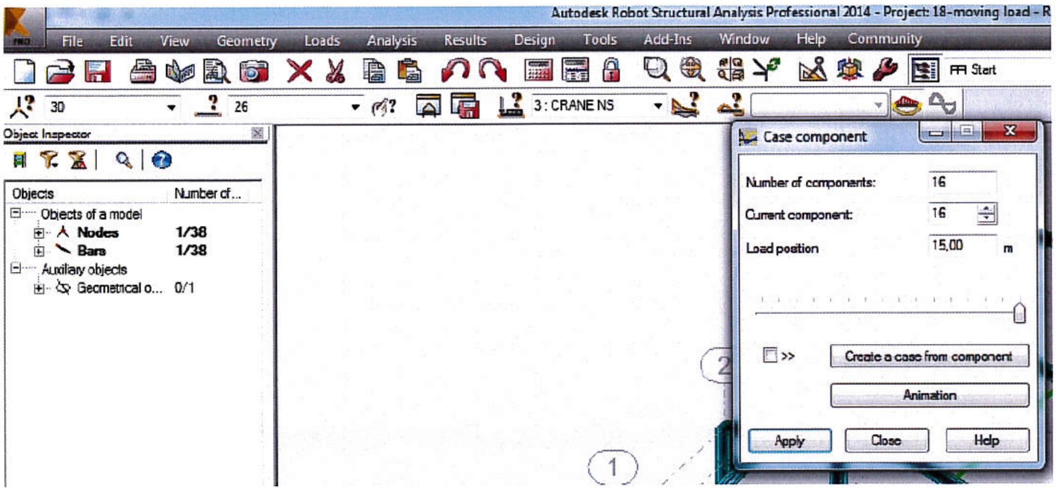
In the same way moving load case for non symetrical crane can be defined



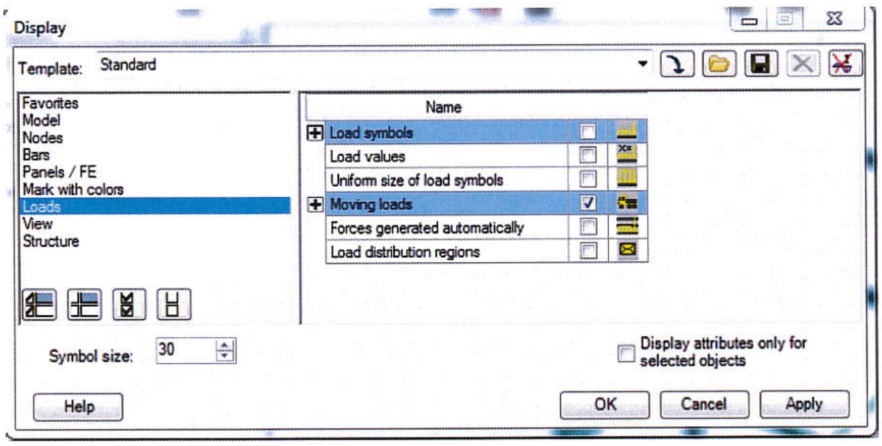
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

11.4. Moving load case definition and parameters

After structure calculation each position of moving load can be analyzed after choosing load case and its component 

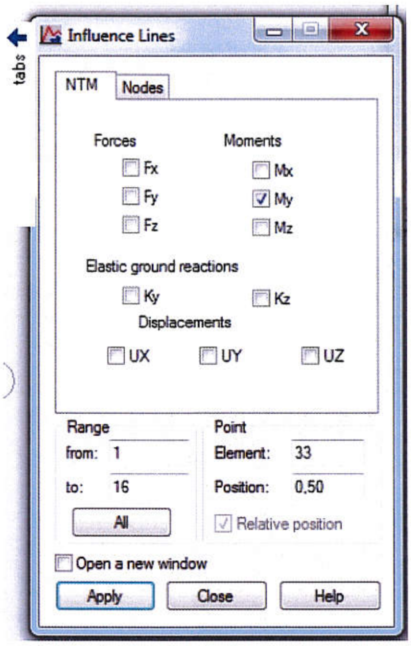


Vehicle position can be seen after switching on Moving loads option in View > Display> Loads dialog box

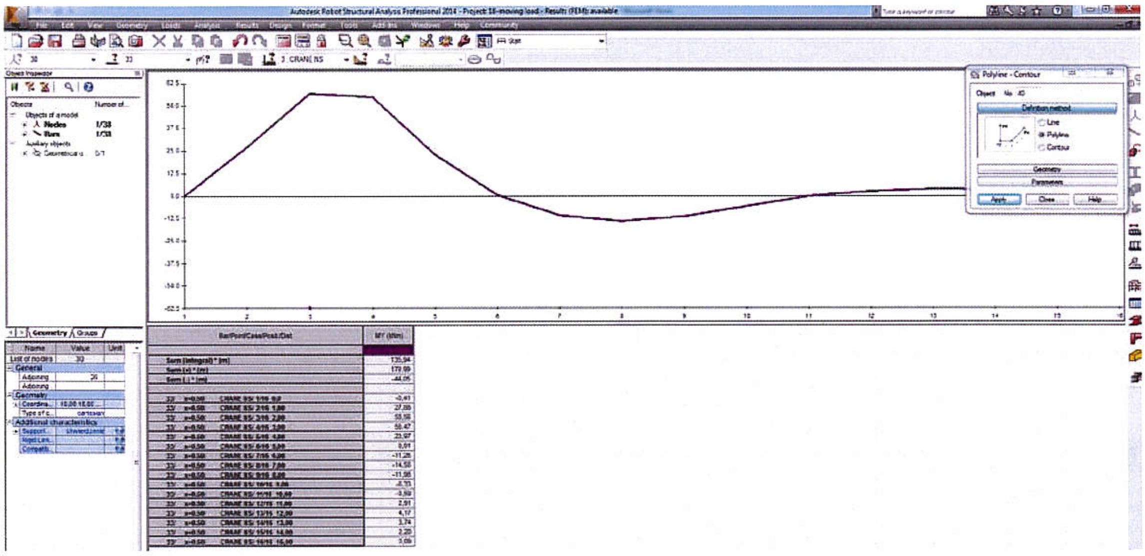


Influence line for selected elements can be displayed by Results >Advanced > Influence line option

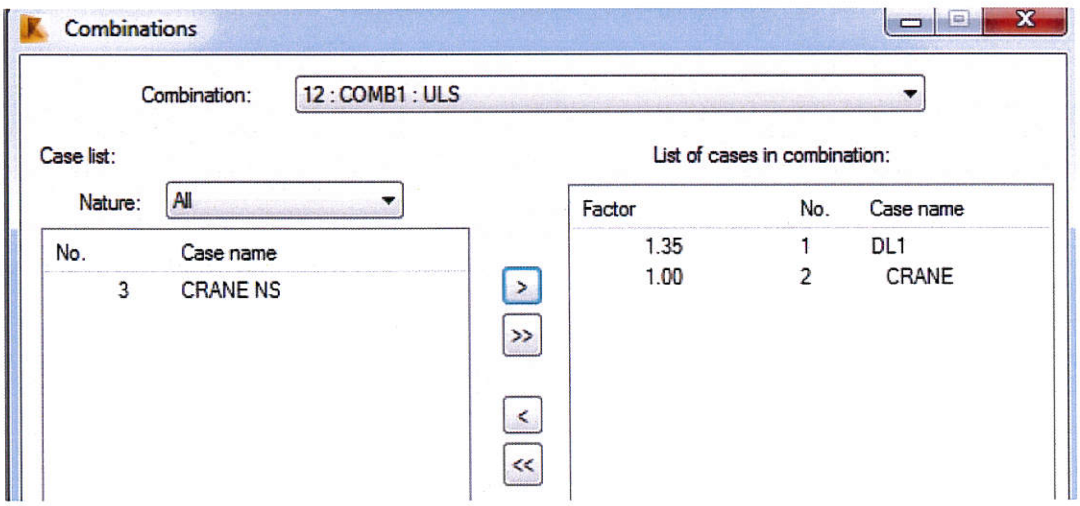
Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved



After choosing element number position and internal forces type we can Apply and obtain diagram



Moving load cases can be used in both manual and automatic combinations



Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved

Then results envelope for all moving load position can be analyzed

Copyright © 2014 Robobat Polska Sp. z o.o. All rights reserved