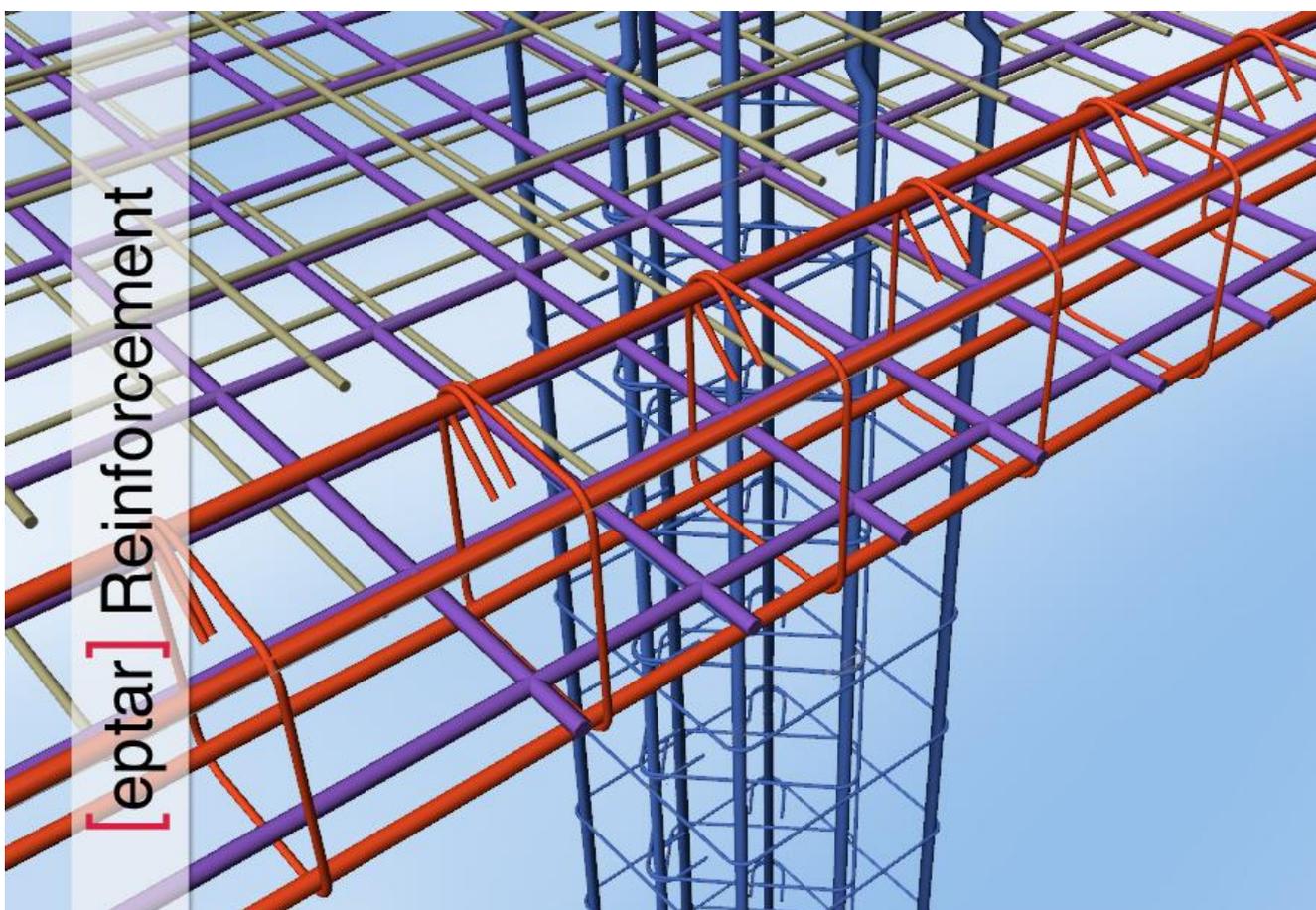


[EPTAR] REINFORCEMENT 1.33 USER GUIDE



Program- and library upgrade:

[EPTAR] Kft.
1145 Budapest
Szugló u. 61-63.
tel.: +36-1 225-7355
fax: +36-1 225-7356
www.[eptar].hu
info@[eptar].hu



[[eptar]] Reinforcement – User Guide

The [eptar] Reinforcement 1.33 – User guide is under copyright and all rights are reserved.

Copyright © 2014 Éptár Ltd.

To copy, translate or extract any parts of the document is only allowed, if written permission is given from the owner.

Version 1.3

Trademarks:

ArchiCAD® is Graphisoft® registered trademark, GDL is Graphisoft® trademark.

Contents

Chapter 1	4
First steps.....	4
1.1 <i>Installation of the solution</i>	4
1.2 <i>Activation of the solution</i>	6
1.3 <i>Loading of the solution</i>	6
Chapter 2	7
Usage of the [eptar] Reinforcement 1.33 application.....	7
2.1 <i>Basic usage of the program</i>	7
2.2 <i>Layout options for reinforcement plan</i>	7
2.3 <i>Placing of the elements and their settings</i>	10
2.4 <i>Object placement with ArchiCAD Accessory</i>	22
2.5 <i>Create Complex structures</i>	24
2.6 <i>Edit Objects</i>	35
2.7 <i>Delete Elements</i>	43
2.8 <i>Use ArchiCAD Label element</i>	43
2.9 <i>Create Section drawings</i>	44
Chapter 3	51
3.1. <i>Cutting list placement and ID check</i>	51
3.2. <i>Use Interactive Schedule</i>	54
3.3. <i>Create custom Cutting list</i>	56
Chapter 4	57
4.1. <i>Technical support</i>	57

New in Reinforcement 1.33:

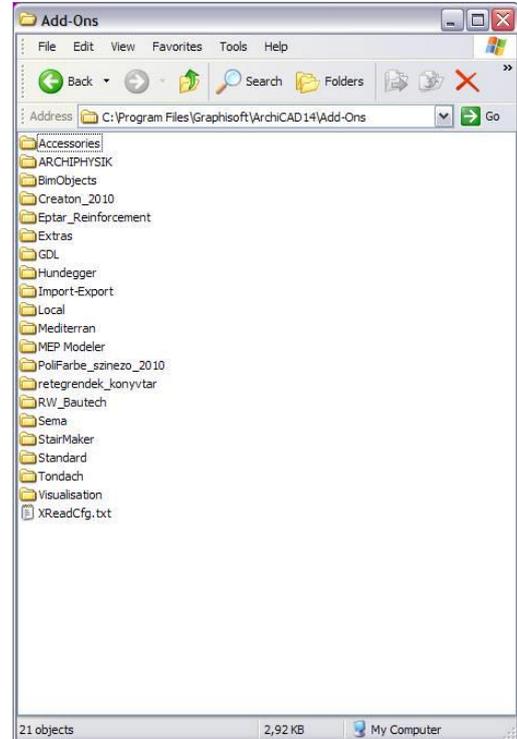
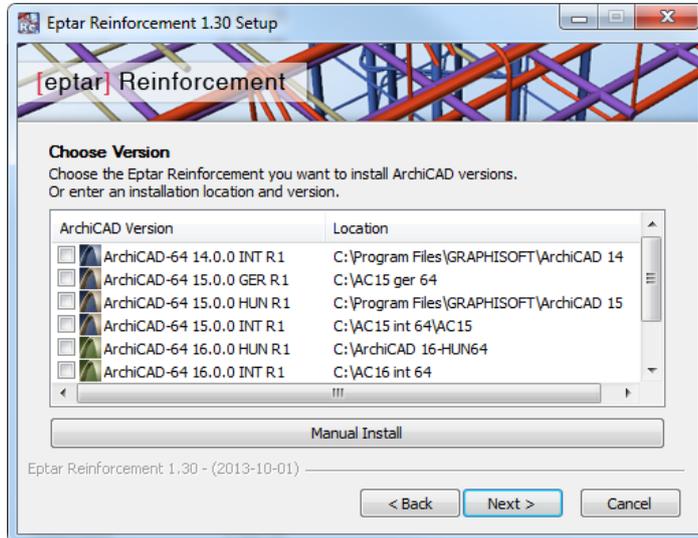
The Reinforcement add-on now supports imperial units – read more in 2.3 Unit Settings

Chapter 1.

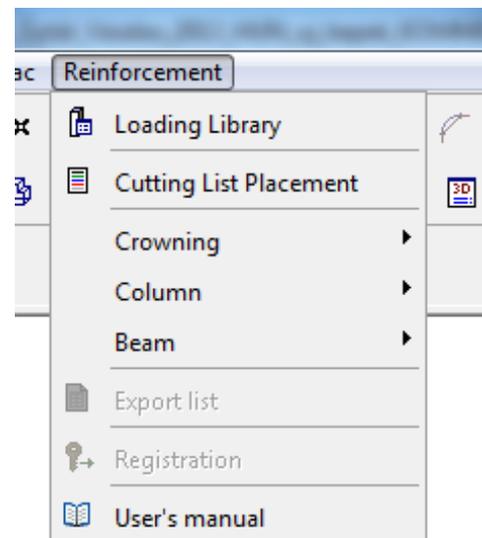
First steps

1.1 Installation of the solution

The [eptár] Reinforcement 1.33 can only be used with ArchiCAD 15 version, or higher. To install the program, start the installation file and follow the instructions given.

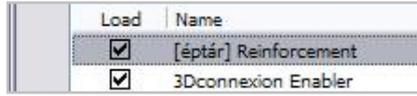


After the installation, at the first launch of ArchiCAD, „**Reinforcement**” option will be available in the menu bar.

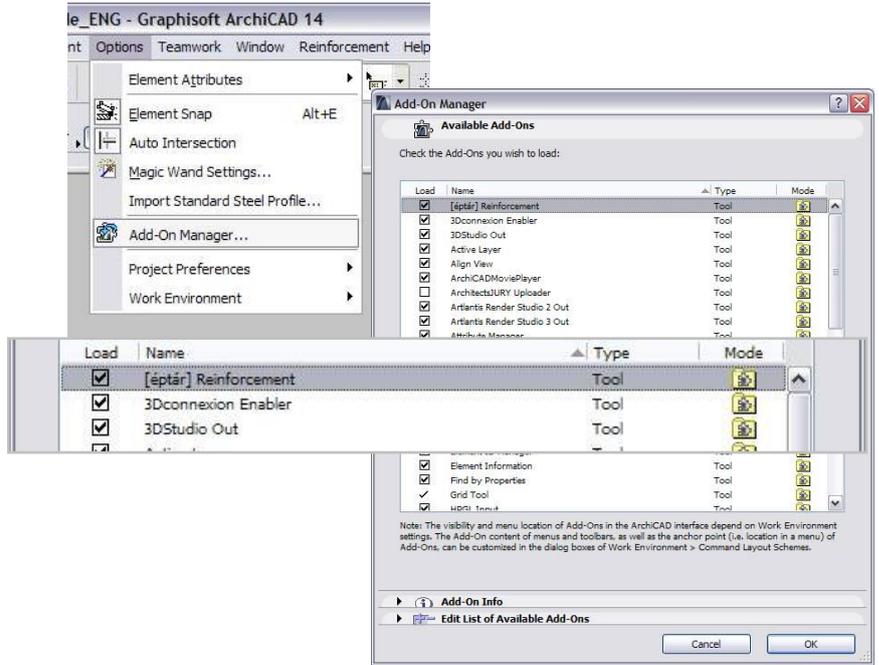


However, if the reinforcement menu does not appear, please check „Options/ Add-On Manager...” menu, if the „Reinforcement” solution is active.

In the case that, in front of the Reinforcement solution, the square is not ticked off, click it, to make the tick appear.



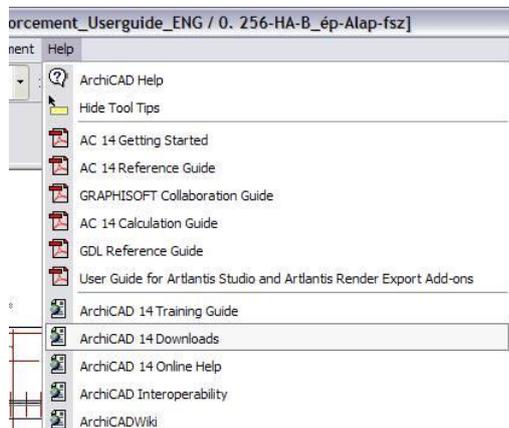
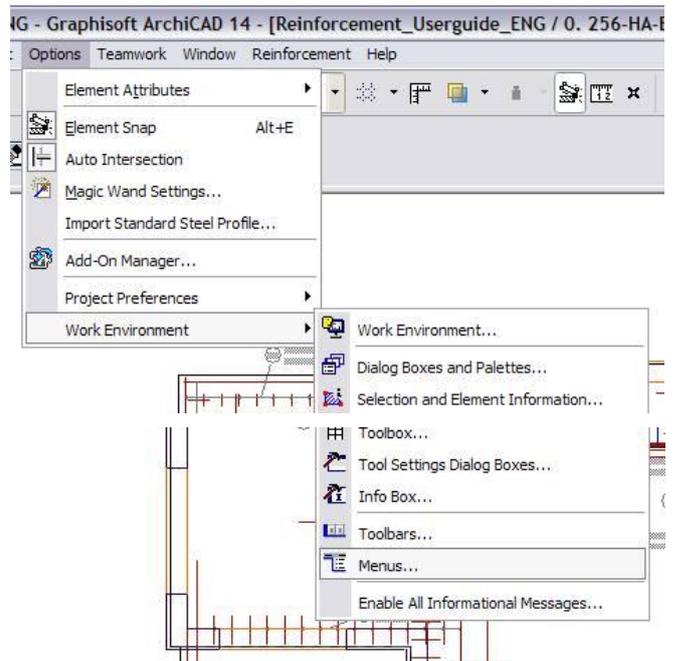
As long as the switch is checked out, please read why ArchiCAD can not detect the solution, and please contact [éptár] ltd. with the problem.



In so far as, the switch is checked out, and the program still can not see the „Reinforcement” menu, please check „Options / Work Environment / Work Environment” dialogue, if Reinforcement menu is not blocked.

Make the menu visible and click „OK”.

As long as all the possibilities have been checked and the program still can not display the „Reinforcement” menu, please contact the producer (éptár ltd.).

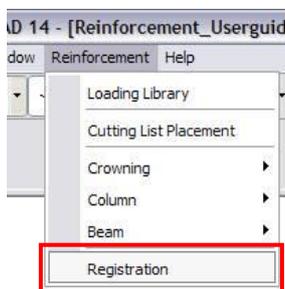


For the usage of the program, it is not necessary, but downloading the „expansions” application group for ArchiCAD can make the work much easier, by containing „Slab Accessory” and „Wall Accessory” elements. These solutions can be downloaded from www.graphisoft.com. To download, select „Help/ ArchiCAD... downloads” menu, in the program of ArchiCAD.

1.2 Activation of the solution

[eptar] Reinforcement 1.33 solution in DEMO mode, without registration, works only with limited functionality. To register the solution/product, you must buy the application in the [eptar] web shop ([www.\[eptar\].hu/cadsupport](http://www.[eptar].hu/cadsupport)), or you must own a Registration file, which you received with the program.

However if you do not own a registration file, but you have purchased the program, please follow these instructions:



For the registration, use that ArchiCAD key, to which you would like to activate the solution (the solution will be assigned to an ArchiCAD key).

Connect your computer to the internet*.

Start the ArchiCAD program.

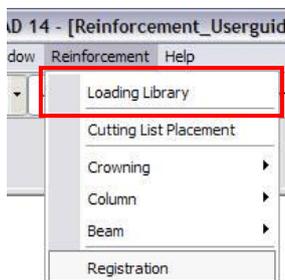
Choose „**Reinforcement/ Registration**” menu. The solution opens [eptar] web shop. As far as you are registered already and have bought the solution, please login. After the login, a page will appear, from where you can download the registration file for the ArchiCAD key.

Copy the downloaded registration file (ValidKeyReinforcement.txt) next to the expansion (reinforcement.apx).

Reinforcement-tool has got the possibility to work with ArchiCAD's network key protection. If you buy this type of key, you will be able to use Reinforcement with as many ArchiCADs in the network as you have purchased. If you have got the registration file before installing the product, then you can give it to the solution while installation. In this case the solution will run in full version for the first run, too. (These options are only available for Windows.)

* If, because of the possible viruses, you would not like to connect the computer to the internet, then please contact the producer.

1.3 Loading of the solution



The [eptar] Reinforcement 1.33 solution is made of the expansion and the attached library. To activate the library choose „**Reinforcement / Load Library**”.

After loading the library, you can place elements like objects, but some elements (rebars, meshes) can be attached to slabs or polygons, by choosing the elements of „**Design / Design extras / Accessories**” command group. By choosing that command, elements of [eptar] Reinforcement 1.33 library will appear, which you can connect to the slabs.

However, if you can not find these expansions in Your ArchiCAD program, please download them from the website of Graphisoft with the help of the „**Help / ArchiCAD downloads**” menu. In the opening browser, choose the „Expansions” Add-On solution. Download the file and install it to the ArchiCAD program.

Links:

ArchiCAD15:

<http://www.graphisoft.com/support/archicad/downloads/ac15/INT.html>

ArchiCAD16:

<http://www.graphisoft.com/support/archicad/downloads/ac16/INT.html>

ArchiCAD17:

<http://www.graphisoft.com/support/archicad/downloads/ac17/INT.html>

ArchiCAD18:

<http://www.graphisoft.com/support/archicad/downloads/ac18/INT.html>

The placement to elements and all the other actions shall be dealt with in the next chapters.

Chapter 2

Using the [eptar] Reinforcement 1.33 application

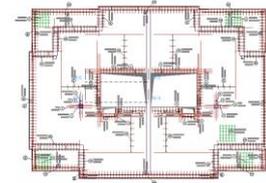
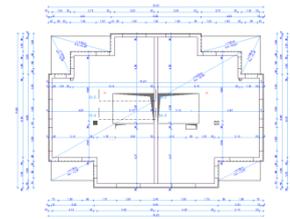
2.1 Basic usage of the program

In the Reinforcement plans, some levels appear differently than in the architectural plans. To get the necessary design for the reinforcement plan you need to change sheet options. Reinforcement plans can be created at the same time as architectural plans, so it is unworthy to change the settings of the architect, so for the reinforcement plan, please creates new view options. (Point 2.2)

After changing the view, some types of irons and webs can be placed as objects on the layout and in the 3D model (2.3. point). In the case of some elements (rebar, web) ArchiCAD expansion's applications can help in the placement (2.4. point). Complex elements (beams, columns) can be created from individual elements, and can be placed on the layout as a group, or by the detailed drawings of the complex elements, can be placed on other sheets too. (Point 2.5)

While placing the elements You can give them separate identifications, but this is not necessary, because in the end of the planning process, while the detection of iron, the program checks the number of irons, and renumbers the faulty or different serial numbered irons and optimizes the numbering (2.6 point). You can optimize the irons by the help of the interactive lists, during the planning. (Point 2.6)

To create the sectional drawings, you can place it in the sectional plan sheets or in the model space, to display the necessary irons' view drawing. (Point 2.7)

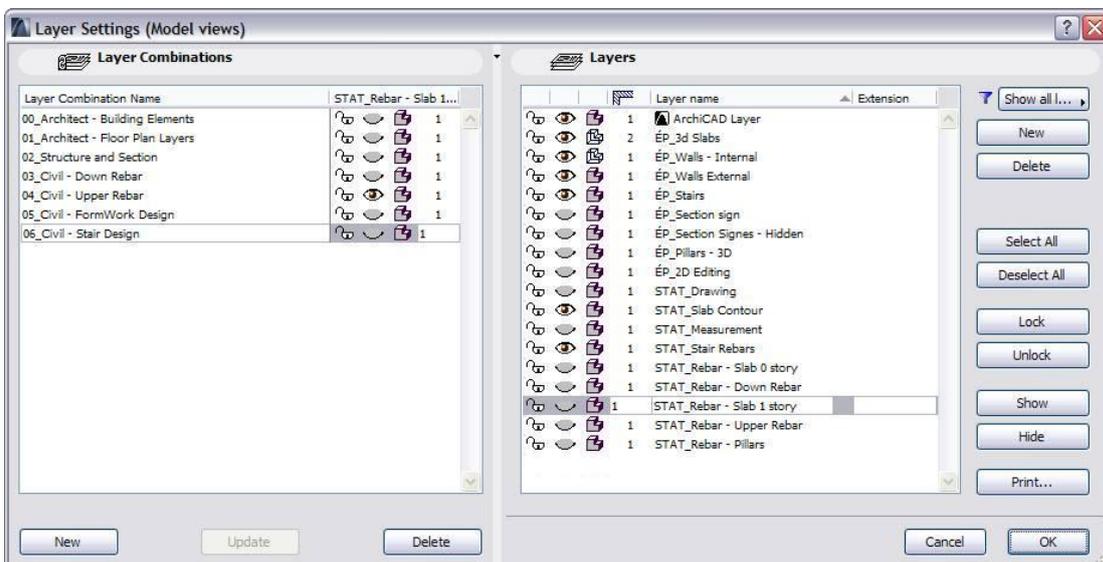


ID	Description	Count/area	Stock/lt	Length	Quantity	Description
1
2

ID	Description	Count/area	Stock/lt	Length	Quantity	Description
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50

2.2 Layout options for reinforcement plan

While creating architectural plans, on some layouts, we cut the building in a 1 meter height, and looking down from here we display the elements of the level. While creating the reinforcement plan, this kind of display shows many useless elements. Although in the reinforcement plan we need to see not just the given level's walls and apertures, but the upper level's walls and the form of the ceiling. To set these options in ArchiCAD, you need to follow the instructions given below.



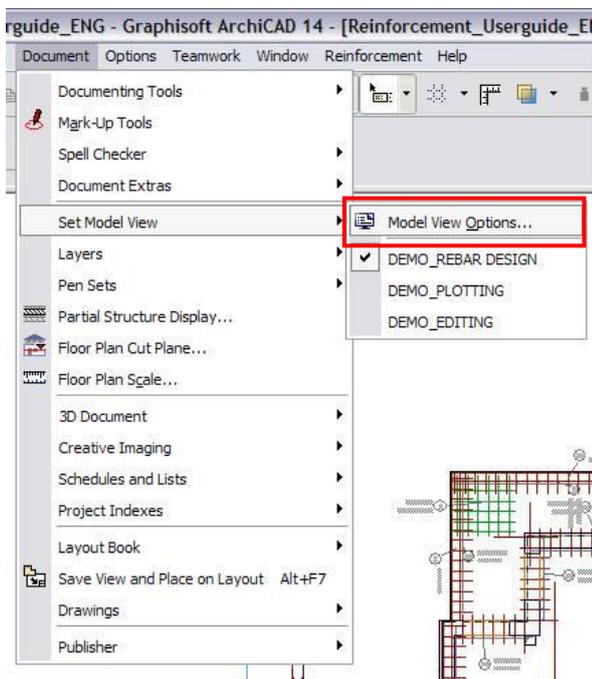
Open „**Document / Layers / Layer Settings**” menu. Among the layers, create (top right button) new layers for the reinforcement plan. We recommend the creation of these layers:

- „Reinforcement – Lower reinforcement”
- „Reinforcement – Upper reinforcement”
- „Reinforcement – Beams, wreaths”
- „Reinforcement – Columns”
- „Reinforcement – complex element container”

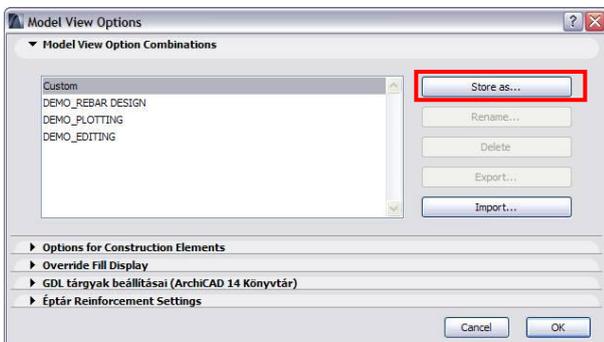
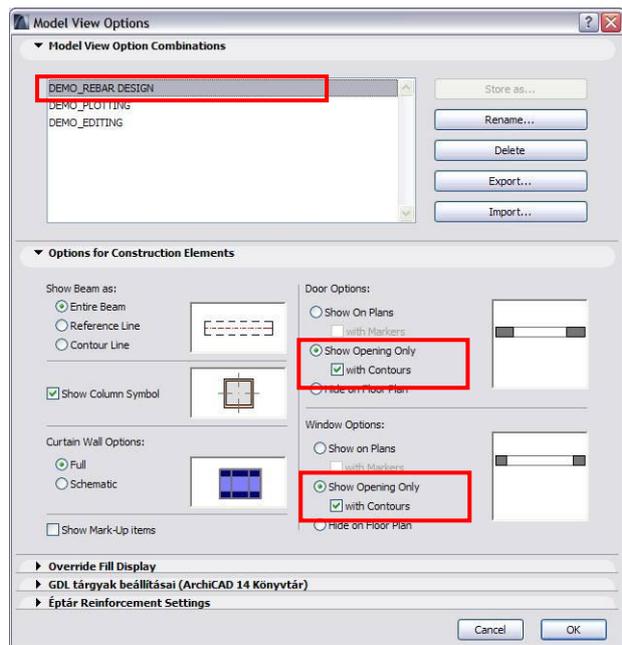
On the left side, create “**Layer Combination Settings**”, with these layer visibilities:

- „Reinforcement – 3D group”
- „Reinforcement – Complex element definition”
- „Reinforcement – Lower reinforcement”
- „Reinforcement – Upper reinforcement”
- „Reinforcement – Segment”

These layer views will help you, to be able to change between views fast and effectively, and, to be able to use the right layer-options at the creation of some sheets.

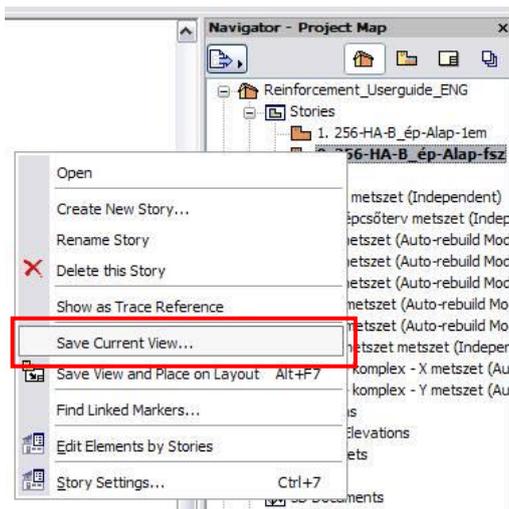


Open „**Document / Set Model View / Model View options**” window. In the appearing dialogue, choose „**04 Civil Engineer View**” type. ArchiCAD, at this kind of display format, only shows the contour of the walls and openings; however we can set an individual display format too.

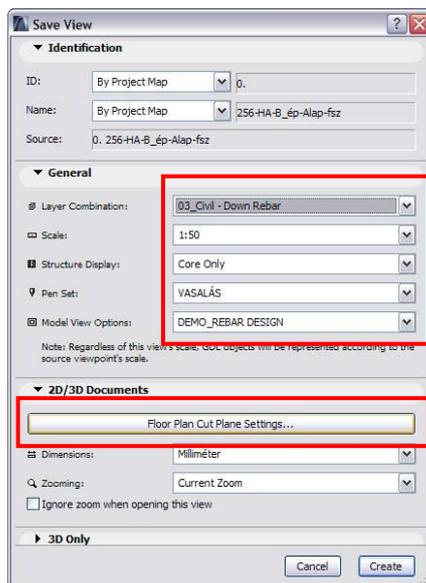


Under the „**Override Fill Display**” tab, switch on „**Override Zone Fills**”, and „**Hide Zone Stamps**” switches.

As it can be seen on the upper side of the dialogue, under the „**Set Model View**”, „**Custom**” has been chosen because of the changes made. Click the „**Store As**” button and give the „**05 Reinforcement plan**” name to the changes. Click OK and close the window.



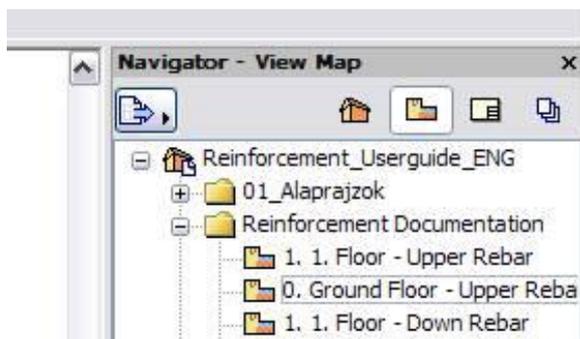
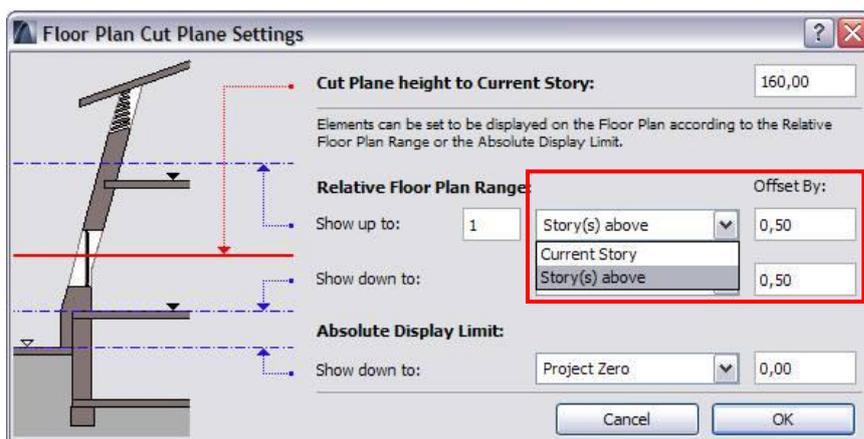
On the right side, on „Navigator” palette, under the „Project Map” tab, right click the name of the active layout. Choose the „Save Current View...” command. Give the view an easy, typical name (e.g. „Downstairs – Reinforcement plan”).



Switch „Layer Combinations” to „Reinforcement – Down rebar” type, put „scale” on 1:50, and „Model View” switch to the (just now created) „05 Reinforcement plan” type.

The setting of „Floor Plan Cut Plane Settings” is necessary because like that we can see the upper level’s walls and apertures on our layout view.

Click „ Floor Plan Cut Plane Settings” button. In the appearing dialogue, next to the „Show Up to” switch, switch the value from 0 to 0.5 meter, and „Show Down to” switch to „Current Story” 0.5.



Click the „OK” button. The view will appear in the navigator, under the „Map View” tab. We recommend you to create a separate folder for the Reinforcement plans with the name „Reinforcement plans” and to paste the created views in this library.

Unfortunately the behavior in ArchiCAD of walls and slabs are not the same, so you can see on the plan view, that the slabs over the actual story (which you are working on right now) are not visible. That is because the walls are set up to support the cutting height on the floor plan view, and the

slabs are not visible on the below story by default. Please step up to the above story and select the slabs which you want to see on your reinforcement plan. In the upper information bar, click „Floor Plan and section”, and then switch „Show on Stories” (switch) to „Home Story and one Story Down” stance.

Comment

With these steps we have set the necessary options for the reinforcement plan. We advice you, that in bigger offices the sample sheets (*.tpl) should be created in a way, that these settings are already set, so afterwards they will not have to be set again and again.

2.3 Placing of the elements and their settings

ArchiCAD can place the necessary irons and webs on the plan as objects. On the left side, under the „**Toolbar**” click „**Object**” icon. In the appearing window, look for „[eptar] - Reinforcement” library, and choose the needed element. As far as Reinforcement library is not yet active, choose „**Reinforcement / Load Library**”.

BASIC NOTES OF THE ELEMENTS

Appearance of the elements on the layout and in the 3D window can be changed globally in a separate dialogue.

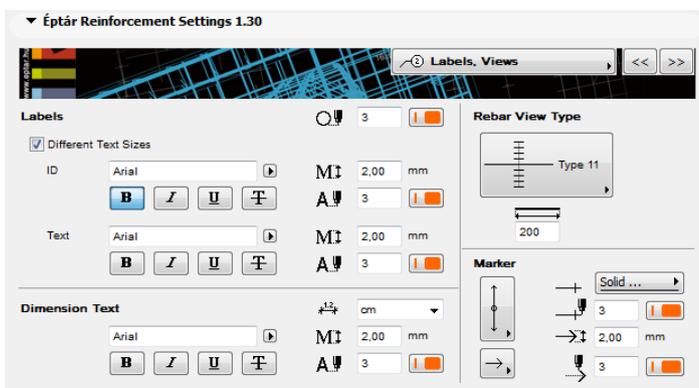
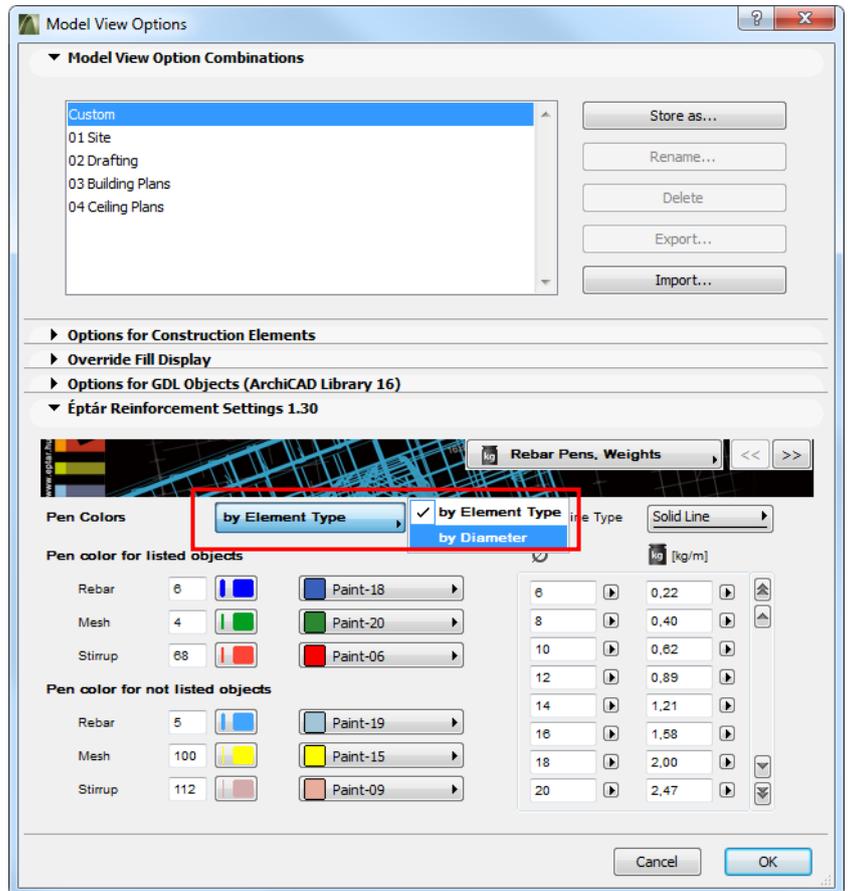
Open „**Document/ Set Model View / Model View Options...**” window. Choose „**[eptar] Reinforcement Settings**” tab (lowermost).

Here you can adjust the color of the pen for every type of element, iron, or every element not in the list, and like that, making the plan easier to read.

Also in this dialogue, you can judge the iron’s diameter of the mass associated, if it is necessary. **Whenever you change a value in the dialogue, you have to refresh the plan, to activate the changes. In the mean time the values of the iron list (weight values) change only, if the optimization of the irons rerun.**

If you want to change the pen-color of some elements in 3D or on the layout, then in the settings window of the element, in „**2D representation**” or in „**3D representation**” page, switch ON „**Object Custom Pen**” switch, and set the desired color.

If you want the program to show the Iron weight with a definite number of decimals, you have to set the display of decimals to the preferred number in the “**Angle & Font Size Decimals in Dialog Boxes**” box which can be found under “**Options /Project Preferences/ Working Units...**” .



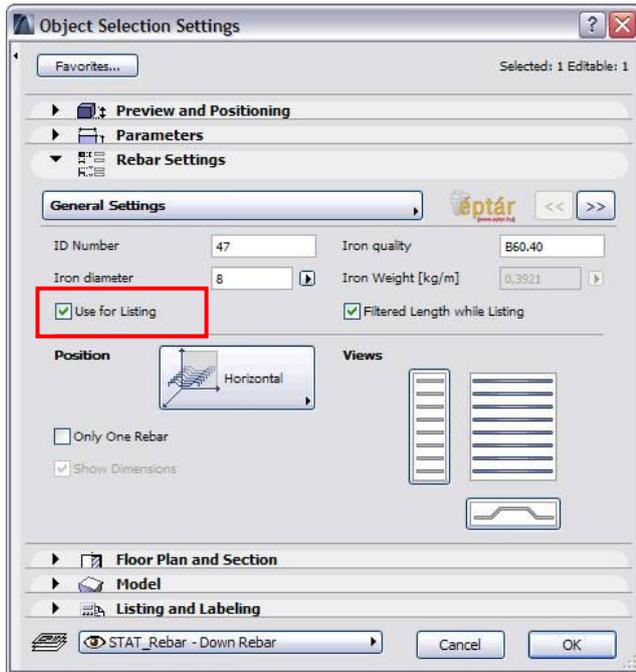
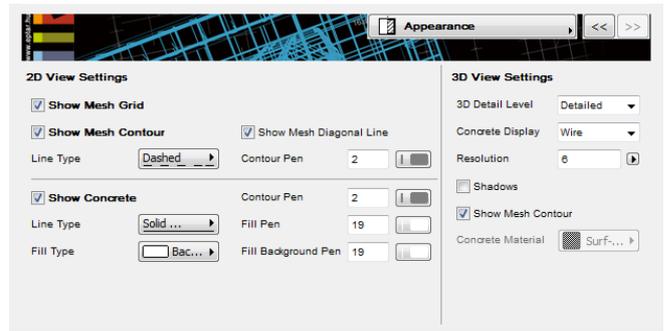
In Reinforcement 1.33 you will find further global setting options. You can define the default settings for the appearance of the views, labels and dimensions of the rebar element.

You can set the text styles of the labels the unit of dimensions on the „**Labels, Views**” page, and choose the Rebar View Type.

Settings of stirrups and meshes will be found on the “**Appearance**” page, and you can even set the 3D detailing and other settings for the whole plan.

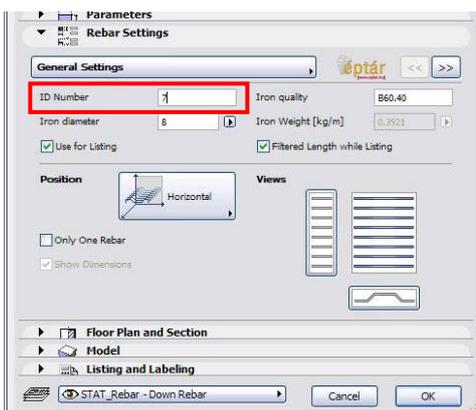
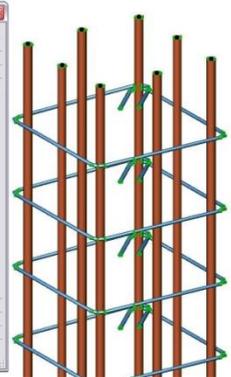
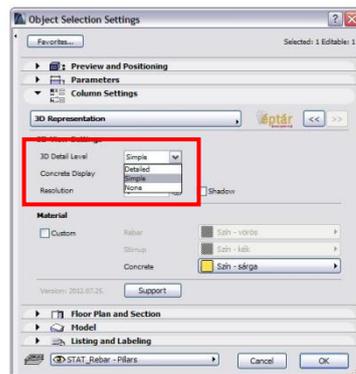
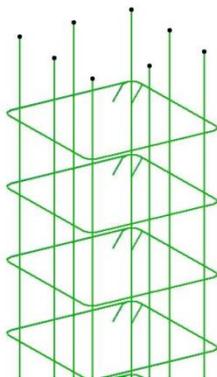
Global settings will define the default settings of each reinforcement elements that you use in the plan. In every single element, you will find the checkbox „Use Library Global”. Switch this checkbox off, to be able to modify the default settings of single elements.

We suggest that you that you create sample sheets (*.tpl) so that these settings are already set. In this case they will not have to be set again and again.



There is a switch in every element „for Listing”. Sometimes, some elements are only placed for a complex element’s creation in the model space, or maybe an explanation, and irons as outstretched views. In these times, these irons can be missed out from the optimization and listing. This switch is basically switched on, so every element gets listed. When switching off, be careful so that you don’t miss out elements what you want to show in the list, from the listing.

The appearance of the elements in the 3D window can be complex or basic. Complex view gives a chance to see possible overlaps and collisions, but can slow the progress/visualization a lot. In basic view, we replace elements with 3D lines, so like this the progress/visualization will be much faster and editing will be easier.



Every element can be granted a separate serial number. If you would like to custom name the rebars, it is possible. In this case of, creation or refreshing of detection of iron, the solution only checks, if irons with the same serial numbers, have the same properties. If you’ve made a mistake and you’ve given two different irons the same number, the program will renumber one of them. However if you do not want to „waste” your time with this, then you can leave all the custom numbers on the basic „1” and during the process, the program will rename them anyways.

UNIT SETTINGS

The Reinforcement add-on supports imperial and metric units as well. The unit setting depends on the settings used for ArchiCAD.

The unit can be set in Options/ Project Preferences/ Working Units menu. If you set Model Unit and Layout Unit to metric units, the Reinforcement add-on will be working just as usual. If you choose imperial units, you will be able to give the measures and to display all measures and list content in imperial units.

If you change the unit once you have already started working, the already placed objects will be converted. If you find that any objects' measures were not converted (this can mainly happen to the side views of the rebar on the floorplan, and the Main View in the Cutting list.) highlight the object (or all objects) and open the Object Settings dialog. If you push OK, the measures will be converted, and you can continue working with the new units.

If you use the metric units, you will be able to choose the unit of the automatic dimensions of the reinforcement objects (mm or cm).

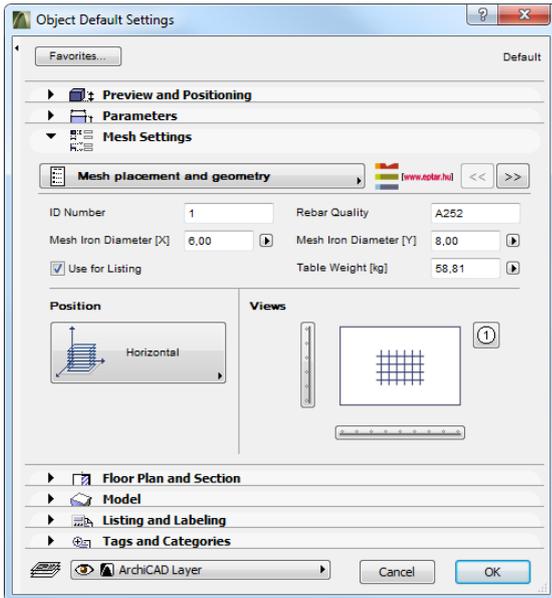
If you use imperial units, you can only use the default unit type on the automatic dimensions.

Important!

The change from metric to imperial sizes does not affect the preset global values of rebar diameters and weights; the usual mm and kg/m values are converted as is to inches and lb/feet units. These values will show a slight difference from the values that are used as standard imperial sizes.

If you want to use the add-on with the standard imperial sizes you will have to make your own value set in the global settings, to match the imperial sizes and make a template that you use with Reinforcement. Check page 10 for the description of global settings.

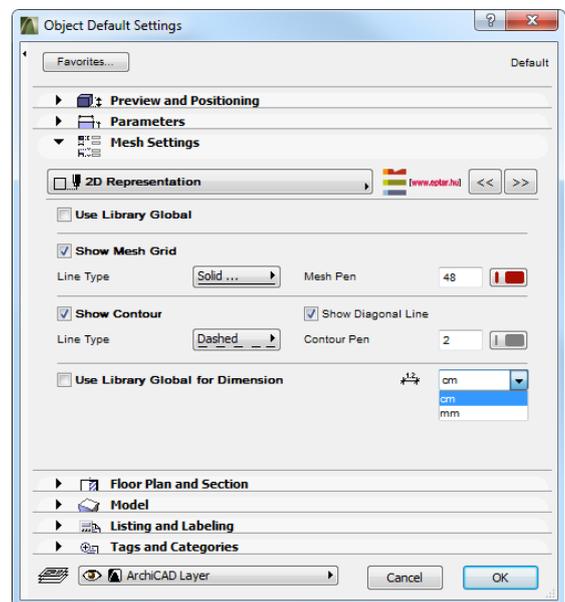
MESH OBJECT



While choosing the mesh object, in the settings window, you can set the element's maximum height in the plan, diameter-, holes values (XX x YY), the size of overlapping and the material quality of meshes too.



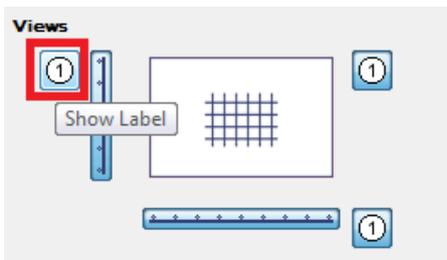
The solution automatically calculates the weight of the mesh element, but if you set up a special diameter (e.g. 9 mm) then the web's m² weight dialogue becomes active and you can give the weight value to the selected diameter and the holes value.



After setting the main parameters, on the 2nd settings page (**Layout view**) you can provide how the web will exist in the plan. You can make only the web's contour visible, make only a piece in the web's middle visible, or make the web's data visible in the plan too. Since these settings can be defined globally, you have to unmark the checkbox „**Use Library Global**” in order to be able to modify the settings.

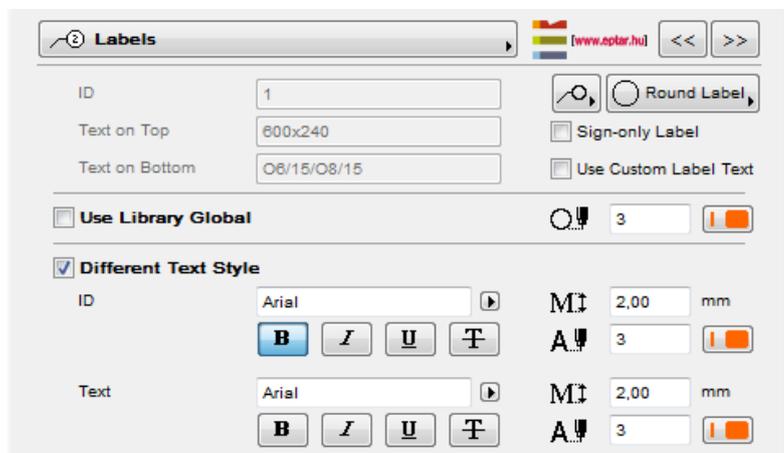
On this page you will see the unit settings parameter, as well.

Attention!
Labels will only appear on the floorplan if you have switched on the “**Show Label**” button on the “**Mesh placement and geometry**” page.

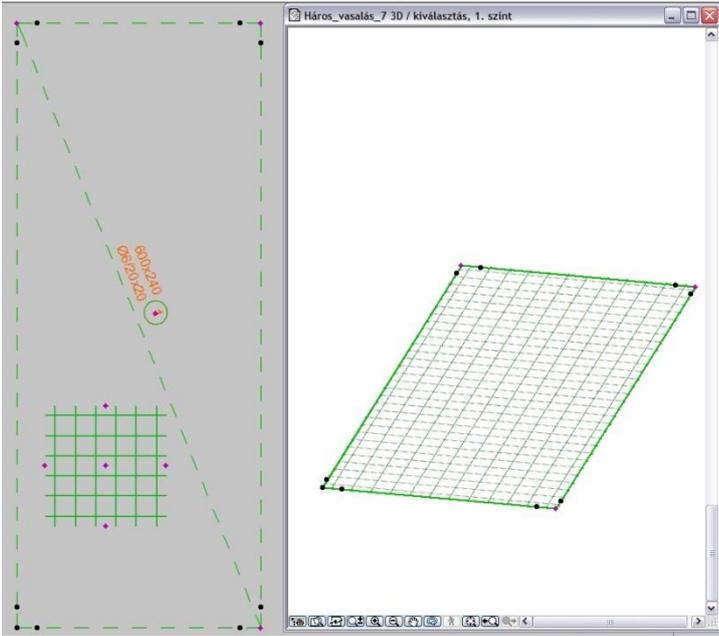


You can set the display of labels on the “**Labels**” page. If you have turned the “**Show Label**” switch on, the mesh will be labeled according to the label type you chose. Content of the label could be overwritten on this page: you can decide whether you want to see the ID, measures, diameter of the rebars on the floorplan. You can also choose the shape of the marker and add a custom text to your label.

You can also set the text style of the label, but it may be even better to set the styles globally.



If you want to use the ArchiCAD label option, you can use the „rf_label” object, what we created directly for Reinforcement solution.



By clicking the OK button, you can place a factorial valued piece of web on the layout. This piece of web can be adjusted (make it bigger or remake it) by the holder points on its heels/corners (pink). No holes can be cut out of the mesh, the solution can not let the user do that yet.

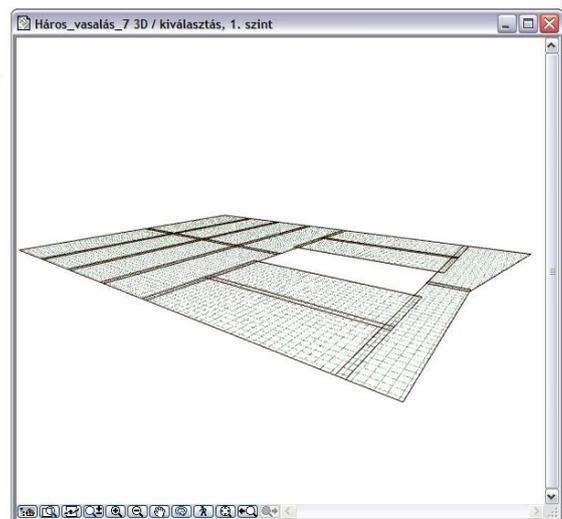
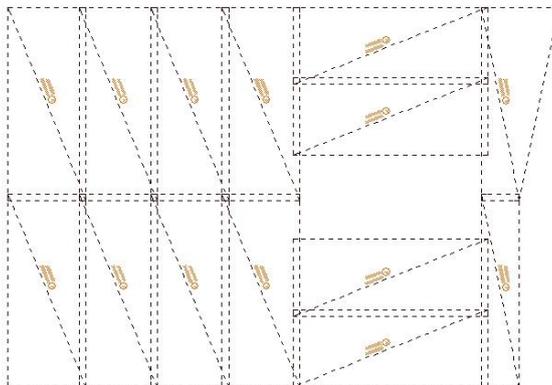
You can set the size of the mesh in the X and Y direction with the help of the hotspots on the sides of the mesh. The size of the mesh can only be reduced compared to the mesh table size. If you want to use a larger piece of mesh, set a larger **Mesh Table Size** in the “**Object Settings**” window. You can reduce the size of the mesh from the direction of only one side. The hotspot on the opposite side automatically disappears once you move the hotspot on the side. Adjusting the size of the mesh with hotspots on the side also has an impact on the position of the slanted sides and displaced corners of the mesh. Therefore we advise you to

set the size of the mesh first and then move the corner hotspots. The mesh pattern inside the placed mesh and it's size can also be readjusted by the editable hotspots on the sides of the pattern.

The pattern inside the placed mesh and it's size can also be remade by the editable hotspots on the corners. If you have switched on the label, then –based on the selected representation method – the information of the mesh will be visible on the cross-line or in the middle of the mesh pattern.

To set up the placement height of the object can be edited in the section view or in the Objects settings dialog window.

You can multiply the object with the ArchiCAD „**multiply**” command or create multiple copies of the object. There are hotspots on the corners of the mesh, these hotspots define the overlapping size of the objects. These parameters can be edited in the Object Settings dialog box with the parameters „**Table Overlapping X**” and „**Y**”. When you multiply or copy meshes, you can connect the objects to these points for the right overlapping.

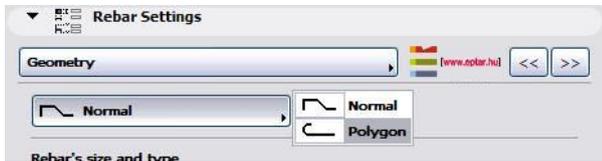


REBAR ELEMENT

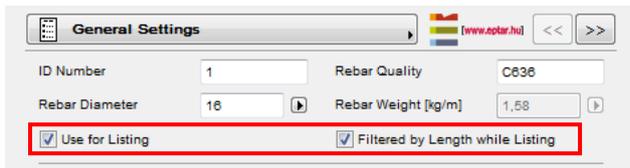
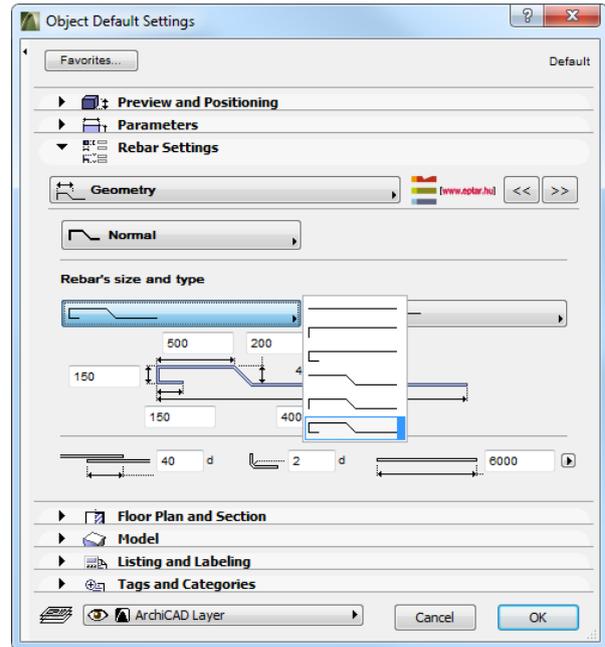
To place the rebar, choose from the dialogue, the „**rf_rebar**” object.

In the dialog (under the „**Geometry**” tab) you can set up the style of the end of rebar and the length of the different parts.

If you can not find the requested end style of the rebar you can choose the „**Polygonal**” option, which helps you define individual rebar shapes (more information in the next chapter)



You can choose two options during the rebar's geometrical definition. Solution can calculate with the full length of rebars, but you can ignore this filter and filtering the rebars regarding their end's structure. If you switch in the „**Calculate with length**” check box, during the listing process we will filter the rebars regarding the full lengths, even they have the same type of endings or not.

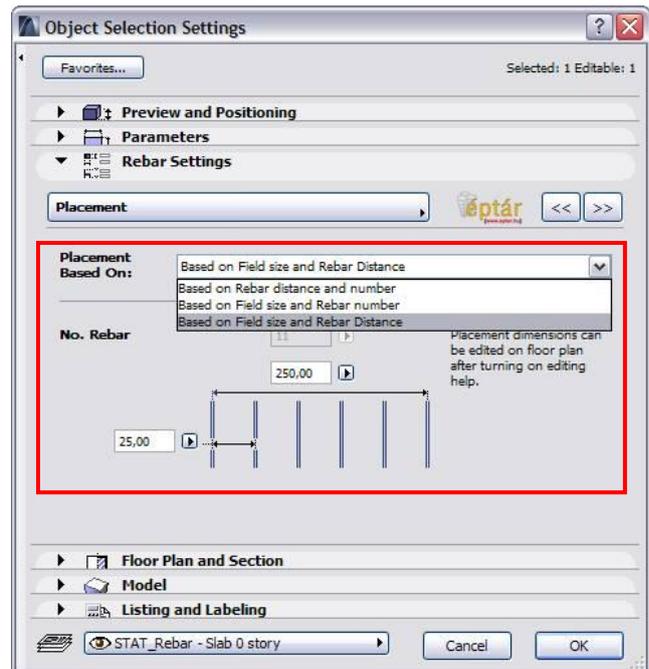


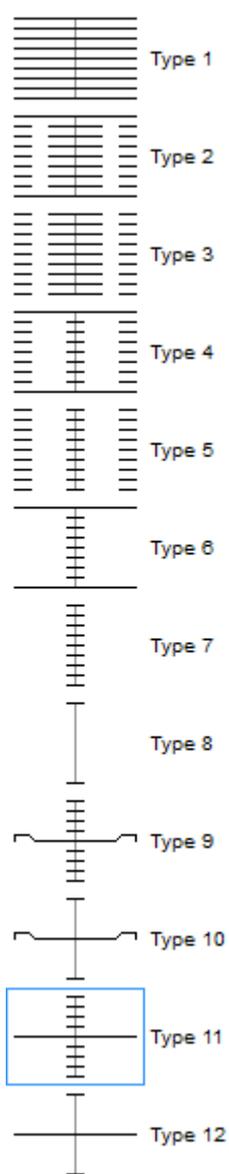
In the placement process you can select to use rebar in Horizontal or vertical position. The vertical option helps you to use the rebars to define columns or other vertical style complex elements.

In the „**Placement**” tab you can choose between different types of placement methods. You have possibilities to place the rebars based on the:

- Length and number.
- Number and distances
- Length and distances.

To place the object into the Floor plan, click „**OK**” button.



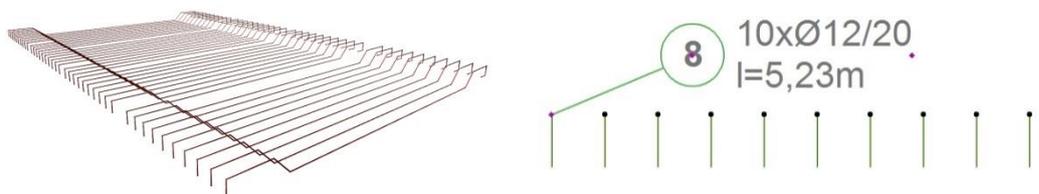


The 2D representation mode and the placement method can be defined with editable hotspots of horizontally placed rebars. Based on the hotspots position, the 2D representation can look differently:

Rebards have got 12 basic display options:

- Overview in full length
- Ladder in the middle
- Ladder and small ladders on the ends
- Ladders on the edges arrows in the middle
- Only arrows and arrowheads.
- One of the options above with lateral bar in the middle
- One of the options above with or without full length bars on the sides.

Comment: Not all types are available for the bars placed with Accessories Add-On.



The vertically placed rebar objects' 2D representation is the top view of the rebar. This 2D view is not able to modify, but you can activate the label for this version as well.

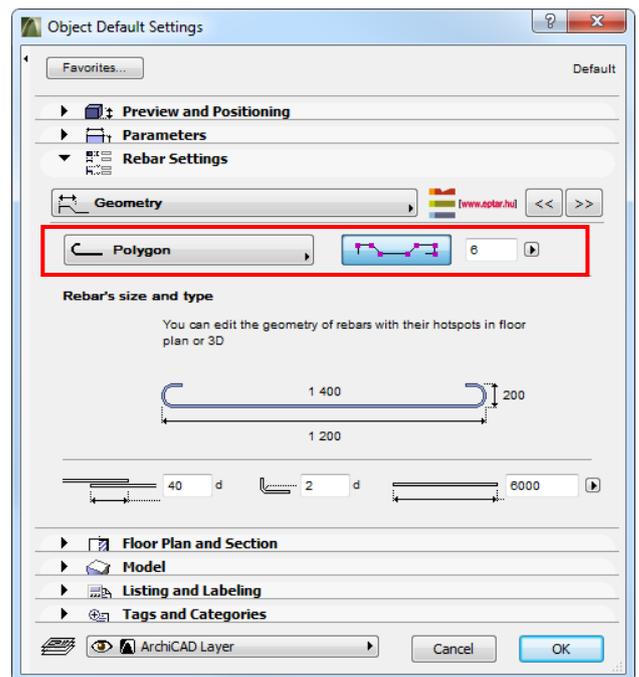
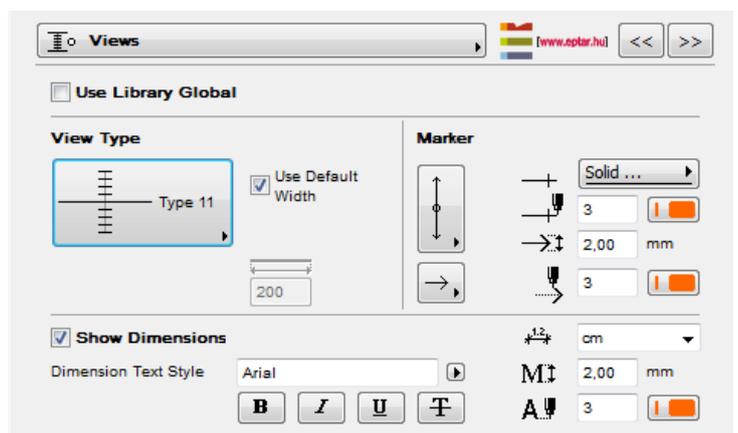
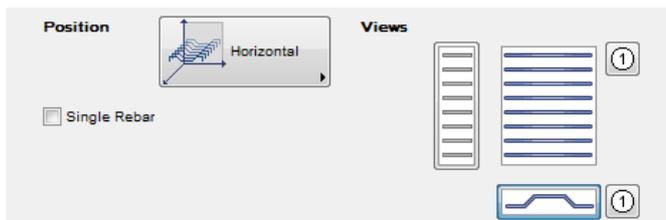
Attention!

Labels will only appear on the floorplan if you have switched on the **"Show Label"** button on the **"General Settings"** page

REBAR – POLYGON REBAR

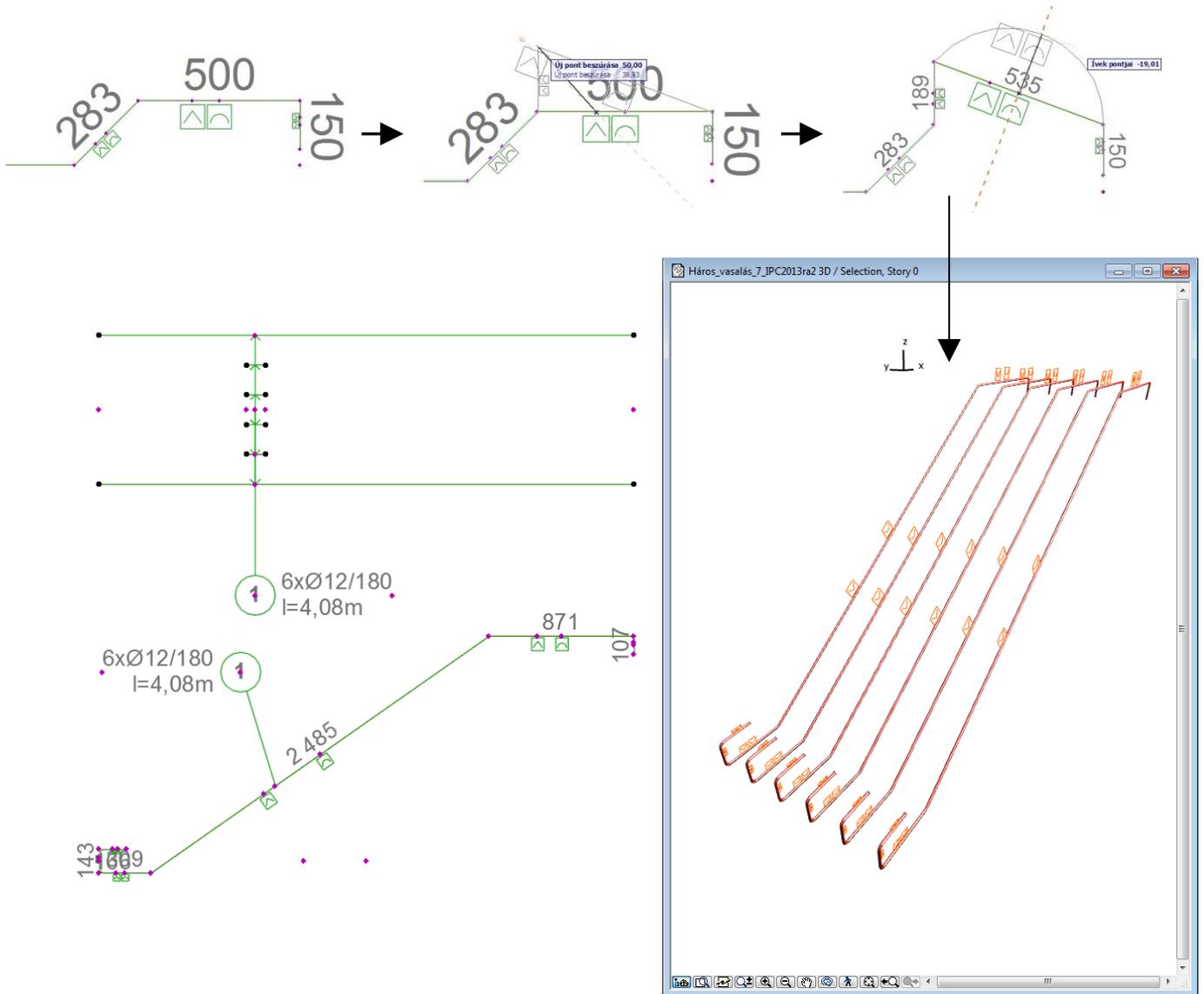
You can choose the most used end styles for the rebar, but there are situations, where construction needs more complicated shapes (maybe an arc shape rebars). In [[eptar]] reinforcement solution you can define these shapes with the „**Polygon**” type rebar.

Choose the „**rf_rebar**” object and select the „**Polygon**” style under the „**Geometry**” tab page and switch on the „**Enable Polygon Editing**” check box.

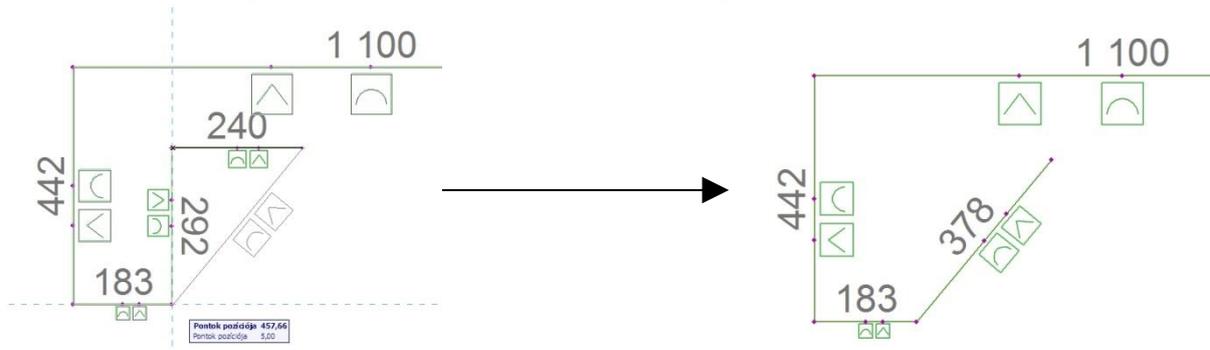


On the floor plan you can see the rebar object and the side view of it.

On every line section there are two editable hotspots (arc and new corner point). To edit these points you can insert new corner into the polyline or define arc between two corner points.



If you want to delete a corner from the polyline, just move it over the next corner point, solution will remove the line sections of 0 cm length, so the two corner points will be merged.



The polygon style rebar can be placed as a single rebar or multiplied for a length.

Note

When you create a new corner point on the Polygon path, one of the two new line sections will contains no editable hotspots. If you want to edit this line section (add new corner point), please move one of the polyline's existing corner point, and in second step move back to original position. Our solution will recalculate the middle points of the polyline and place the necessary editable hotspots on the path.

STIRRUP OBJECT

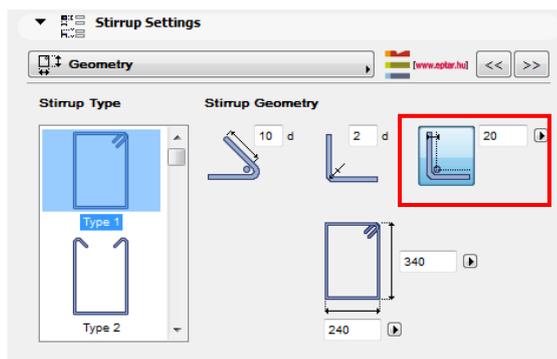
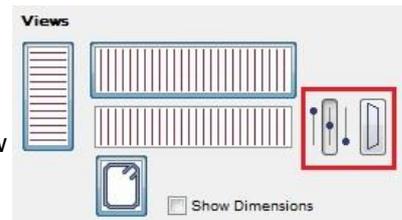
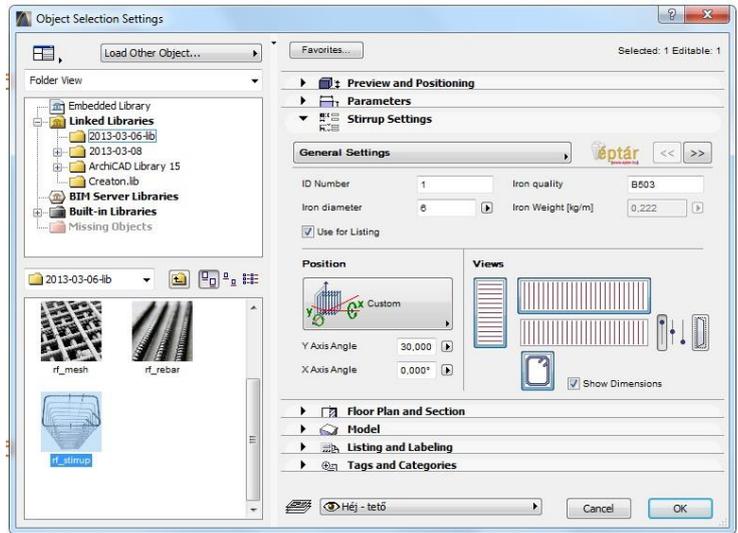
The stirrup object is a basic accessory for more complex elements. With the help of stirrups we are able to define columns (pillars), beams and crowning objects.

Select the „rf_stirrup” object in the object settings dialog.

In “Position” menu you can choose vertical (for columns), horizontal (for beams) and custom position for the stirrups. In the case of custom position you can rotate the stirrups around its X and Y axis. When you rotate around the Y, the axis of the stirrup group will turn, so the individual stirrups get a slanted position in space.

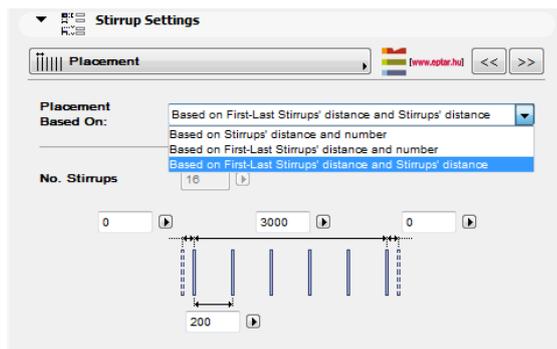
If you want to make a slanted stirrup group with vertical stirrups, we advise, that you place an individual stirrup and use the built in “Multiply” order of ArchiCAD.

In the case of custom position stirrups you have got many view options to choose from. Besides the usual vertically projected views you can choose the slanted view of the stirrup and you can also decide if you would like to see the top, middle or bottom section of the stirrups or the detailed 2D view on the floor plan view.



Select stirrup section form and set up the parameters on the „Geometry” tab page.

You can set the distance between the axis of the rebar and the side of the stirrup on this page as well. If you would like to see this on the floorplan you can turn the “Show Rebar Distance” switch on.



If you do not find the necessary predefined shape in the dialog select the „Polygon” style.

The polygon style helps to define free stirrup shape on the floor plan.

When you choose polygon based stirrup, you can choose between „open” or „closed” shapes. If you choose the open style, be careful with the iron final turnbacks during the polygon definition.

You can place stirrups in three different ways (similar to rebar):

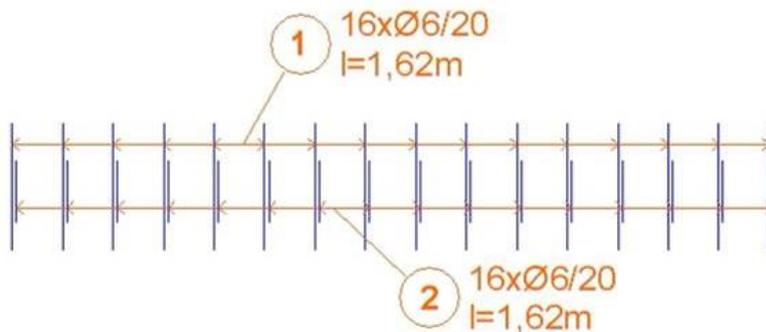
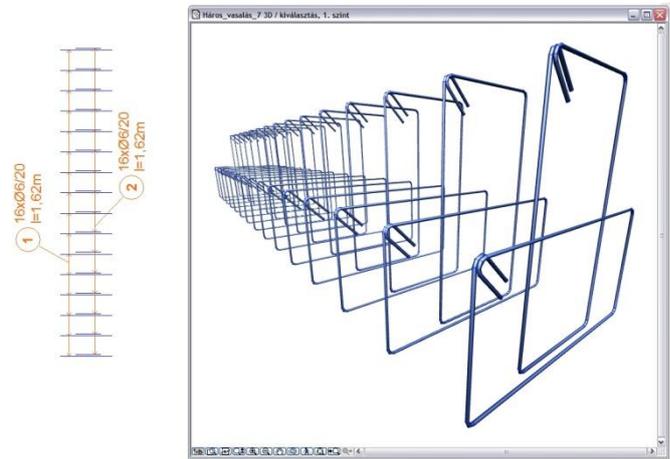
- Length and number.
- Number and distance
- Length and distance.

You can edit the distance between the stirrups with editable hotspots on the floor plan or in 3D view.

In horizontal placement 2D symbol contains the same amount of lines (top view of the stirrups) as many stirrups are on the model. In a case of vertical placement floor plan view is the section view of the stirrup.

On the floor plan you can switch on the Label for the object on the „2D Representation” tab page. You also can use the original ArchiCAD label elements, more details are about this functionality in point 2.8.

On the floor plan you can switch on the Label for the object on the „2D Representation” tab page. You also can use the original ArchiCAD label elements, more details are about this functionality in point 2.8.



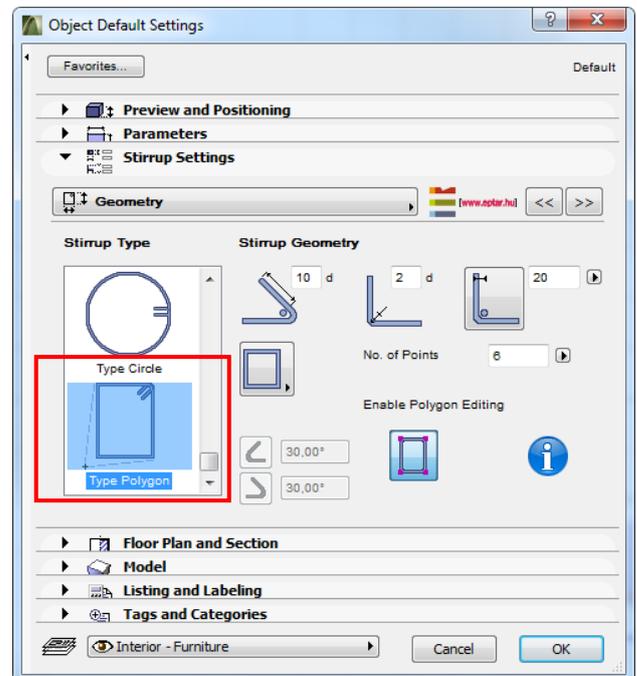
STIRRUP OBJECT- POLYGON

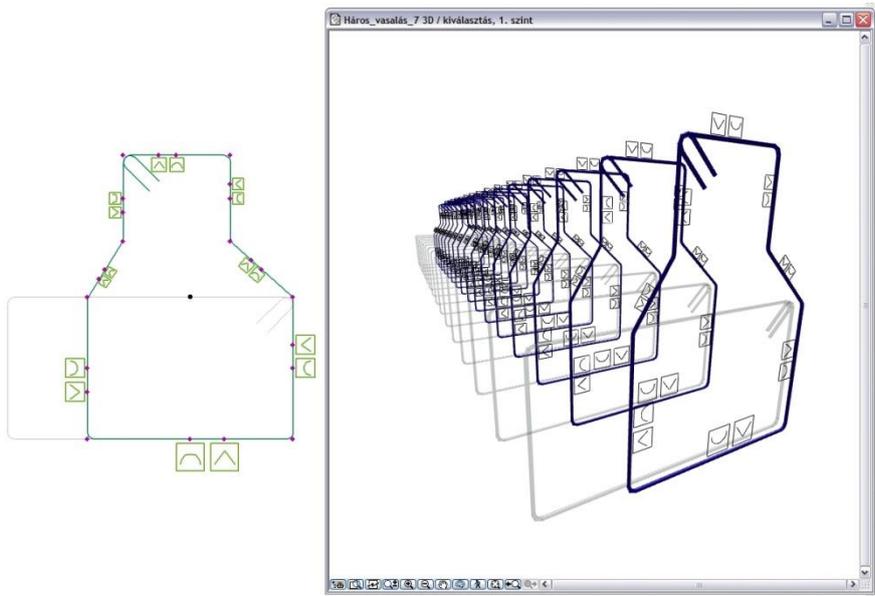
If you choose the „Polygon” shape placement into the Stirrup settings dialog, you can define free, polygon path defined stirrup object.

Select the „rf_stirrup” object. On the settings dialog on „Geometry” tab page select the „Polygon” shape. Click „OK”, place the object into the floor plan. The solution place a closed or opened stirrup polygon into the scene.

You can find two editable hotspots on every edge of the stirrup. One of these hotspot help to insert a new point into the polyline, the other defines arc instead of straight section-line.

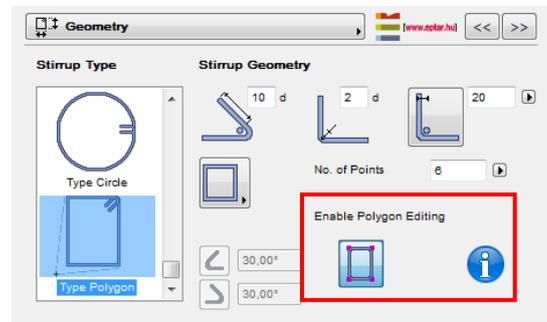
Move the editable hotspots and define the required section shape.





When you are finished the shape definition, switch off the „**Editable hotspots**” check box. The small icons will be disappeared on the 2D view and 3D model.

You can multiply the stirrup objects horizontally and vertically as a normal stirrup element. 2D and 3D representation method is the same as we listed in Stirrup placement point.



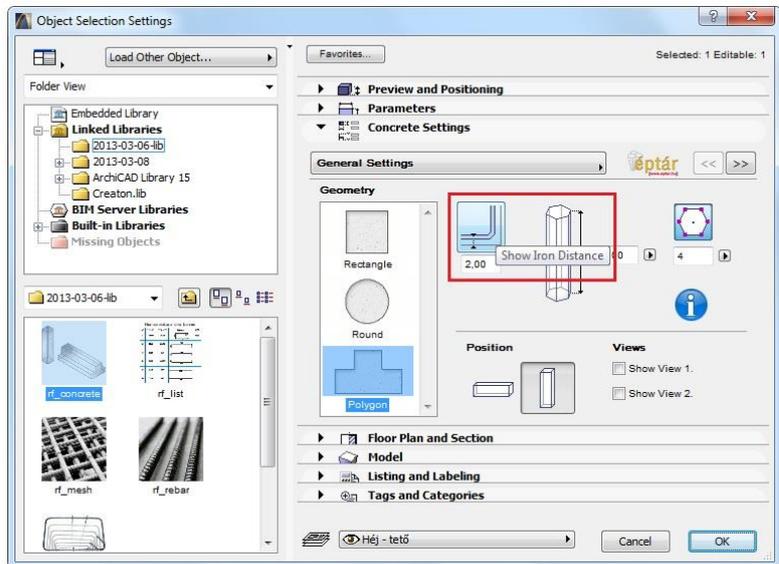
CONCRETE OBJECT

Concrete object is an auxiliary object to define complex elements. The object helps to place rebars and stirrups to the correct position regarding the defined concrete cover.

Object is placeable vertically and horizontally. You can define the shape of the object on the „**General Settings**” tab page.

The section shape can be:

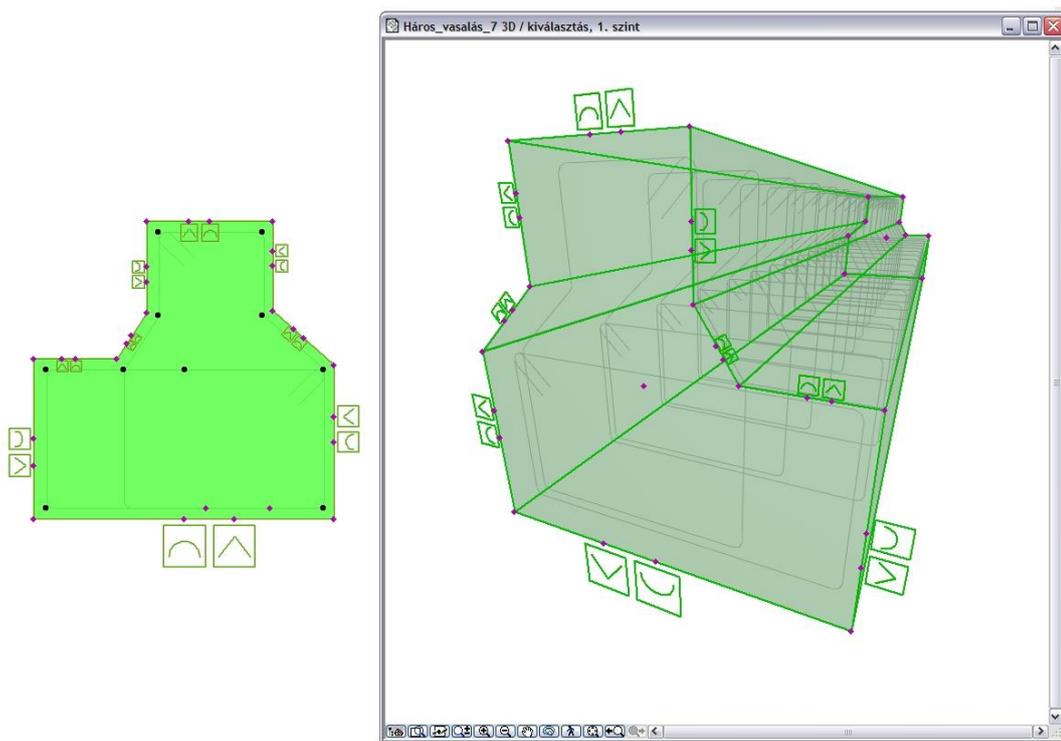
- Brick
- Circle (Cylinder)
- Extruded polygon.



Normally the columns and beams have a constant section along the full length, so one concrete object enough to define the external shape of the complex element. If you need more complex shapes, you can use more concrete elements to define the final shape.

If you set up the „**Iron Distance**” parameter in the „**Objects settings**” dialog, there will be hotspots and hotlines to define the exact placement points for rebars and stirrups. If you turn on the “Show Iron Distance” switch you will be able to see the placement lines.

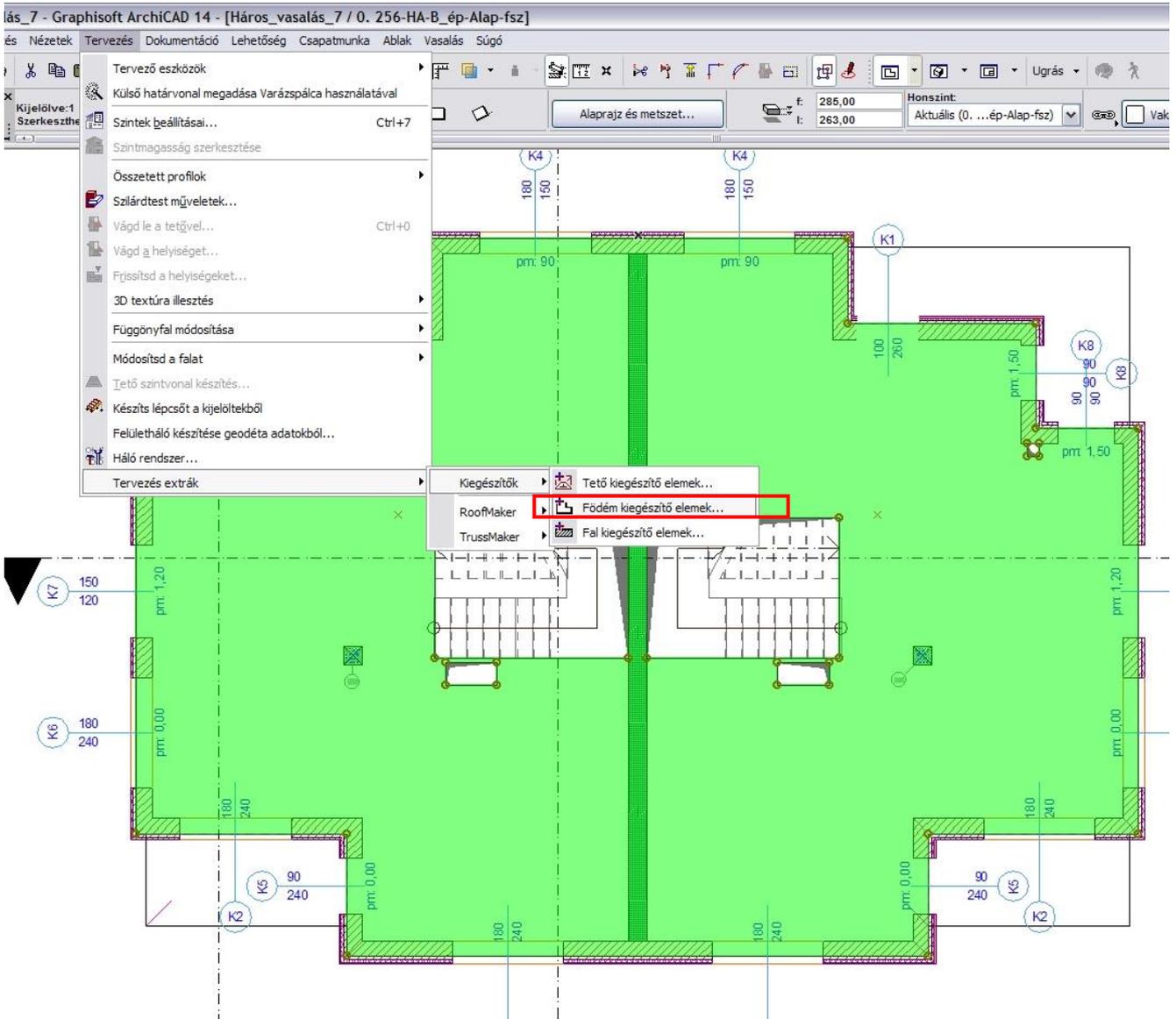
The object is easy to modify with editable hotspots on the floor plan, section- or in 3D-view.



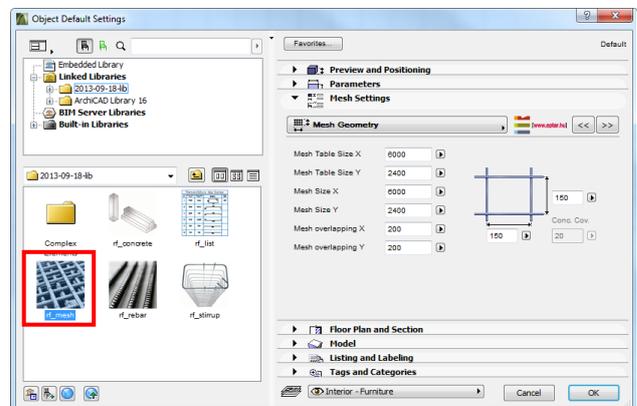
This objects is not available in the cutting list and do not count in the Rebar optimization method.

2.4 Object placement with ArchiCAD Accessory

The mesh and the rebar elements are available to connect closed polygons on the floor plan as well. This ArchiCAD process helps for the users to place rebars and meshes to a non-perpendicular or/and not rectangular shaped concrete building structures.

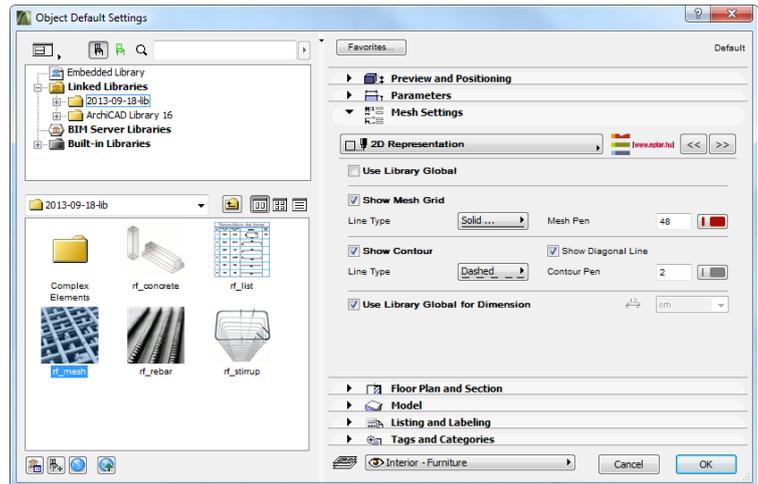


Select the ArchiCAD slab (only one) object to fill out with rebar or mesh elements. Use „**Design / Design Extras / Accessories / Slab Accessory**” command. Select the rebar or mesh object in the dialog box and set up the necessary parameters as we defined in the earlier chapter. Click on „OK” button, the ArchiCAD automatically place the rebar or mesh elements into the polygon shape slab with the parameters what you defined.

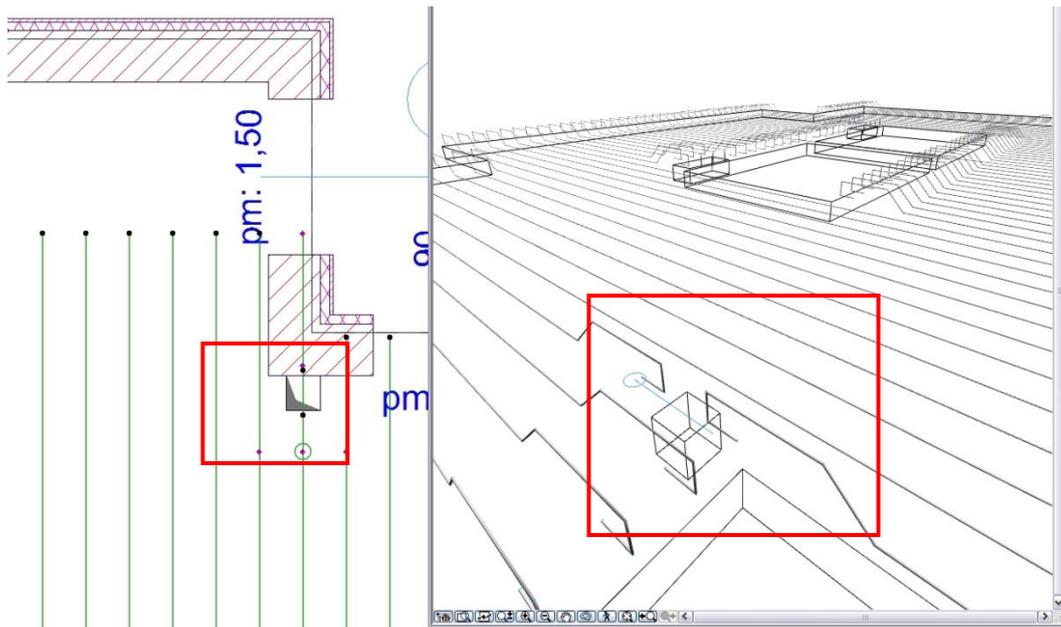


To change the placement direction of the rebar / mesh objects, select the object and switch on the „**Enable polygon editing**” or „**Show Sections**” check box in the Object settings dialog. Click on the „OK” button. On the floor plan you can see the editable hotspots, which helps you to define the placement direction and the placement origin.

Set up the necessary parameters. Switch of the „**Enable Polygon Editing**” check box.



If you define an ArchiCAD slab with holes on the surface the solution can support it, so the rebars and mesh elements do not cross the holes. Please, be circumspect when you define small holes for ArchiCAD slabs (for tubes, mechanical stuffs), because the software do not hesitate to finish a rebar if it cross the hole and place a new one after the hole. It can cause problems, if you are not careful enough.



The rebars and meshes - which has been placed with ArchiCAD Accessory – will be grouped, you can not mix the styles of the rebar or meshes in one placed group.

In this 1.0 version the mesh do not support to calculate with overlapping and do not calculate the optimal placement direction. We are on it to create a new update which helps for you to support this functionality.

2.5 Create Complex structures

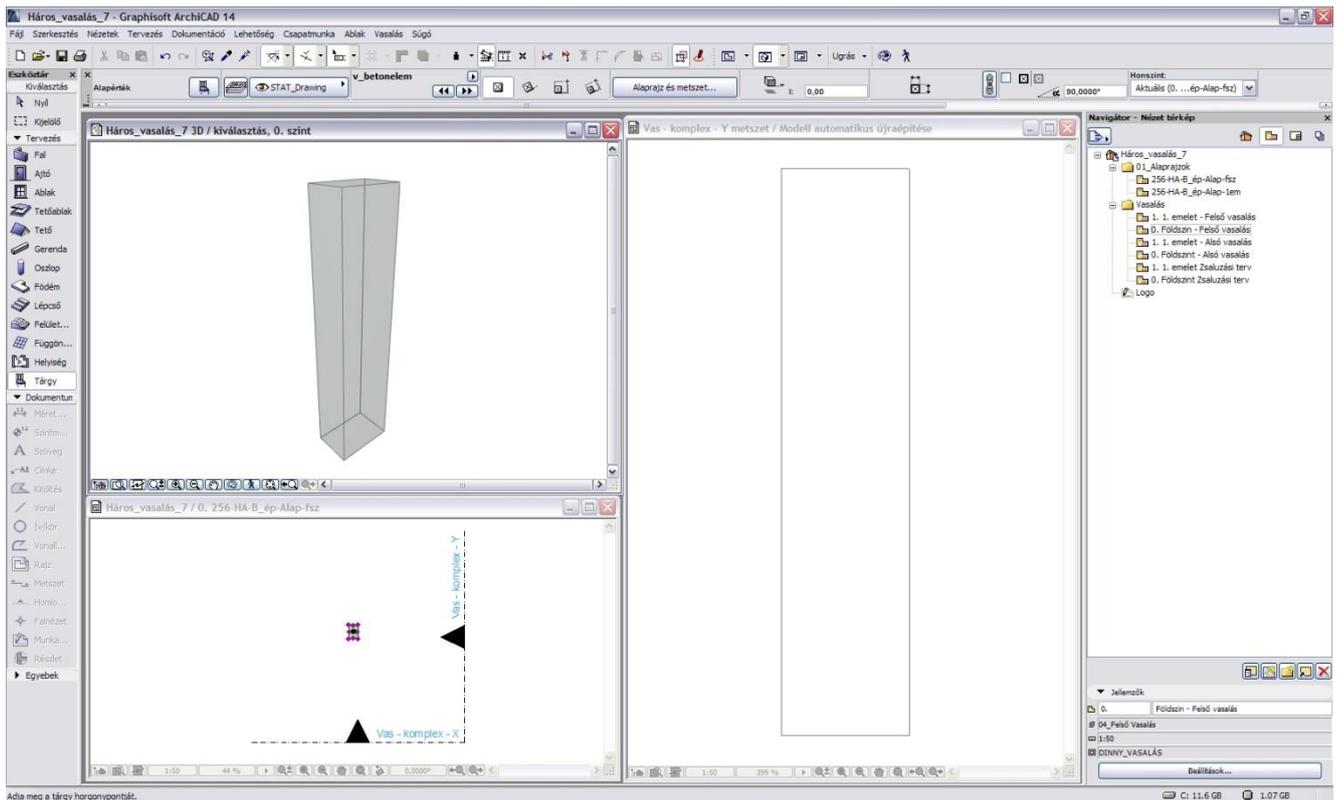
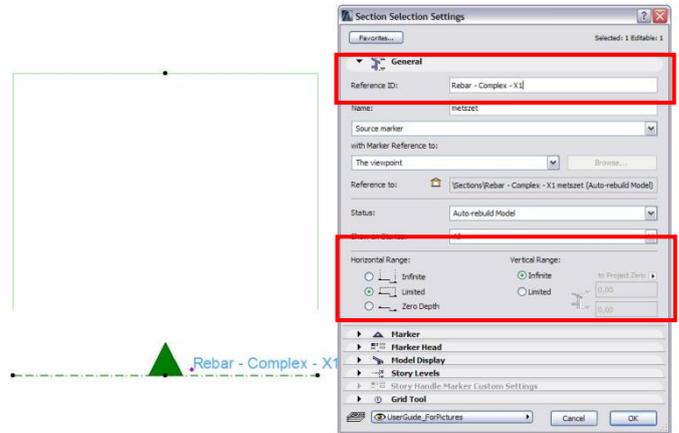
Columns, Beams and Crowning are complex elements in our solution. These structures can be defined with a group of basic objects (rebars, stirrups).

Working Area

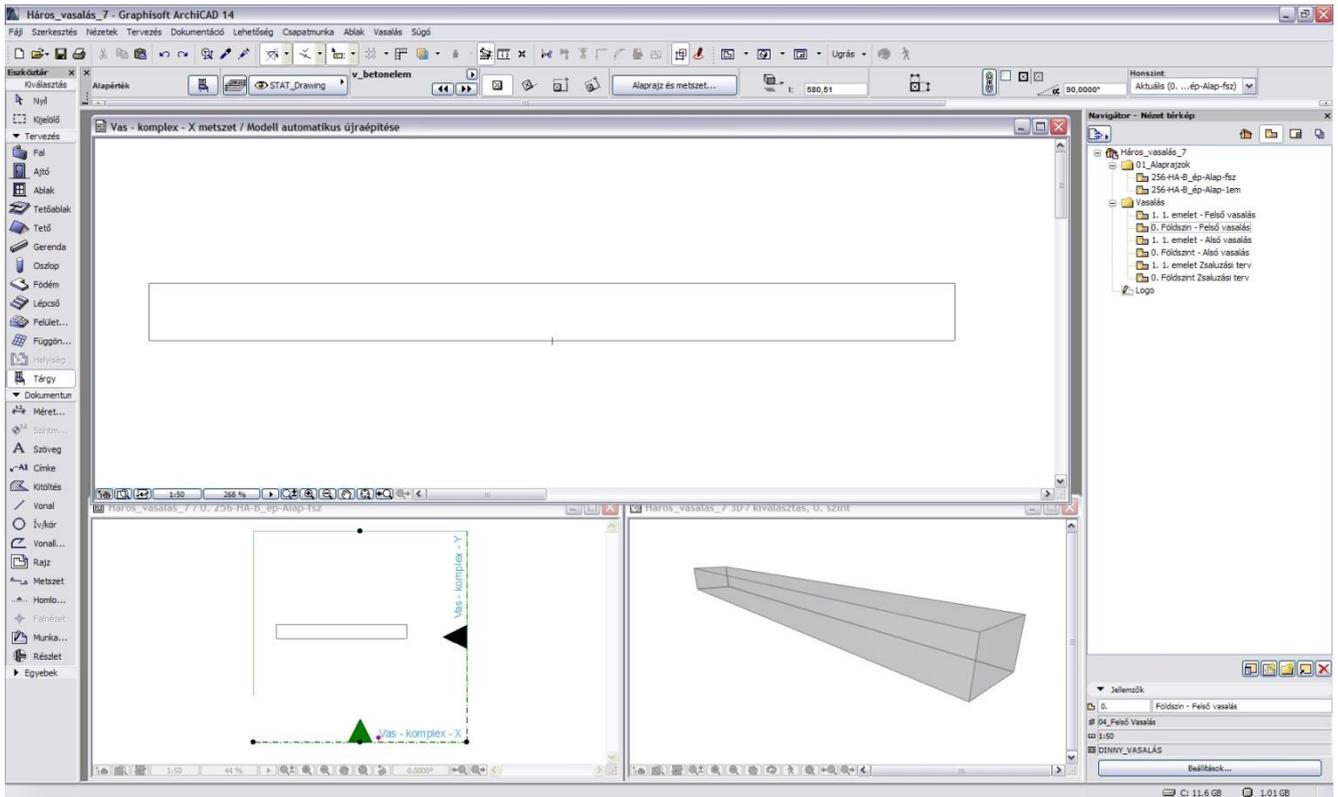
To define columns and beams we need an environment where we can edit side and front views of these structures parallel with the floor plan edit. For this working process, please define a new section on the floor plan, which is out of the working area of the building structures. Rename the section window to „**Vas – complex – X**”. Change the section line length to 4-5 m short. Select the section and open the „**Section Selection settings**” dialog. Change the „**Horizontal range**” to „**Limited**”. Press the „**OK**” button and change the view distance to ~5m.

Move the copy of the section on the floor plan and rotate it with 90 degree, these two sections will represent the front and side view. Change the name of the copied section to „**Vas – complex – Y**”.

To create a column we suggest to use a similar window sets like this one.



For beam reinforcement definition we suggest a similar window sets like this one:



COLUMN DEFINITION

In our system one column contains at minimum 1 stirrup and 2 rebars. Fewer objects are not enough to define column structure.

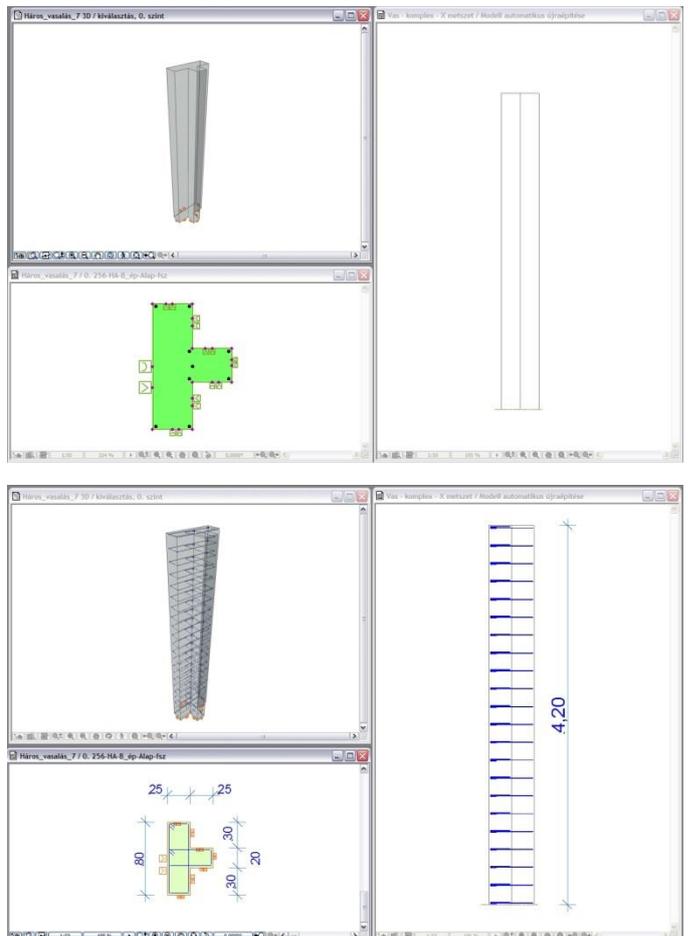
Place one or more „**rf_concrete**” object on the floor plan somewhere in the horizontal range of the sections „**Vas - komplex - X**” and „**Y**”.

Define the concrete external surface of the column with these objects.

Select all the „**rf_concrete**” objects and set up the concrete cover („**Iron Distance**”) parameter value.

Select the „**rf_stirrup**” objects from the library. Open the „**Object Settings**” dialog and choose the „**Vertical**” placement option on the „**General**” tab page. Choose shape for stirrup and set up the parameters. Click „**OK**” button and place the stirrup object inside of the „**rf_concrete**”. Connect the stirrup corners to the hotlines and hotspots, which define the safety concrete covers.

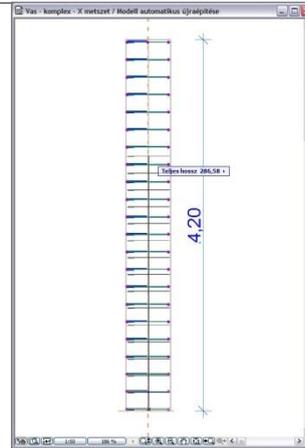
If you choose polygon style stirrup, please define the correct shape with the editable hotspots on the floor plan view, connect the polygon corners to the hotspots and hotlines of the „**rf_concrete**” object. Click on the opened Section window, the stirrups will be visible immediately in the bottom of column.



Use the editable hotspots to define the correct height of the stirrup group and change the stirrup's distance, if it is necessary (for more information about stirrup placement see 2.3. point).

Unfortunately there is no possibility to copy or place 3D elements in the section window, if you need more stirrup group in the column, go back to the floor plan, copy the existing stirrup object and click on section window again. The new object will be available and you can move it to the right position, set up the geometrical parameters with the editable hotspots.

When you are ready with the placement of all the stirrups, you can define the rebars.

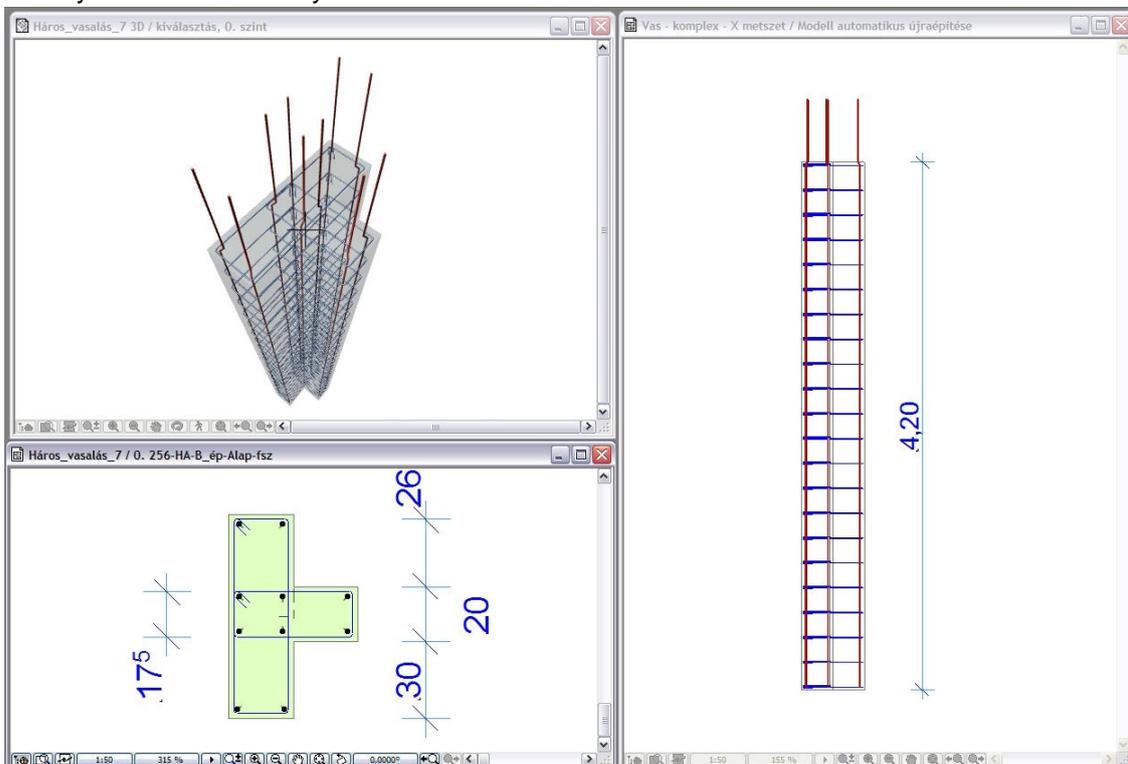
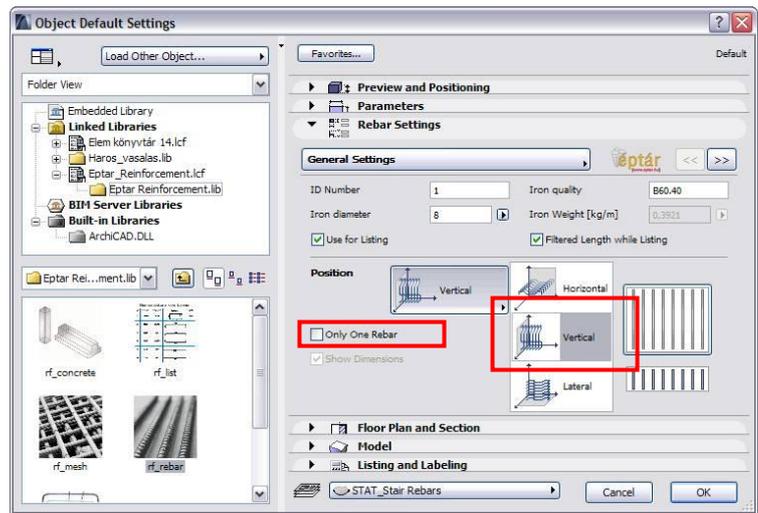


Select the „rf_rebar” object from the library. On the User Interface select the „Vertical” placement option and switch on the „Only one rebar” check box.

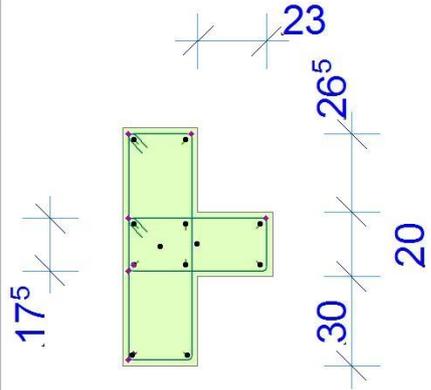
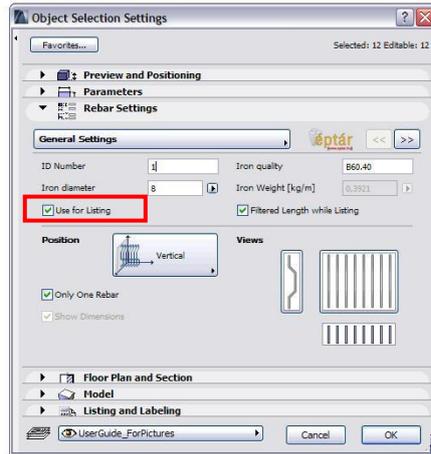
Set up the right style of the endings in the „Geometry” tab page and change the parameters. Click on „OK” button. Place the rebar object inside the stirrup contour and rotate it to the right direction. Click on the Section window and change the values with the editable hotspots of the rebar, if it is necessary.

If you want to place more pieces of rebar with the same parameters, go back to the Floor Plan view and multiply, copy them to the right position.

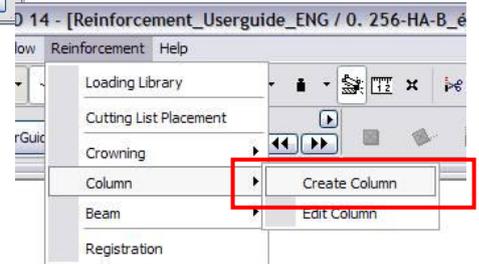
If you need rebars with different parameter sets for the column, repeat the process above as many time it is necessary.



When you are ready with the column objects rebar structure, select all the rebar, stirrup, concrete elements in the Column and make sure, the „Use for Listing” parameter is checked in.



Select all the rebars, stirrups and concrete elements in your construction on the floor plan (section view is not supported for this command) and choose „Reinforcement / Column / Create Column” command.



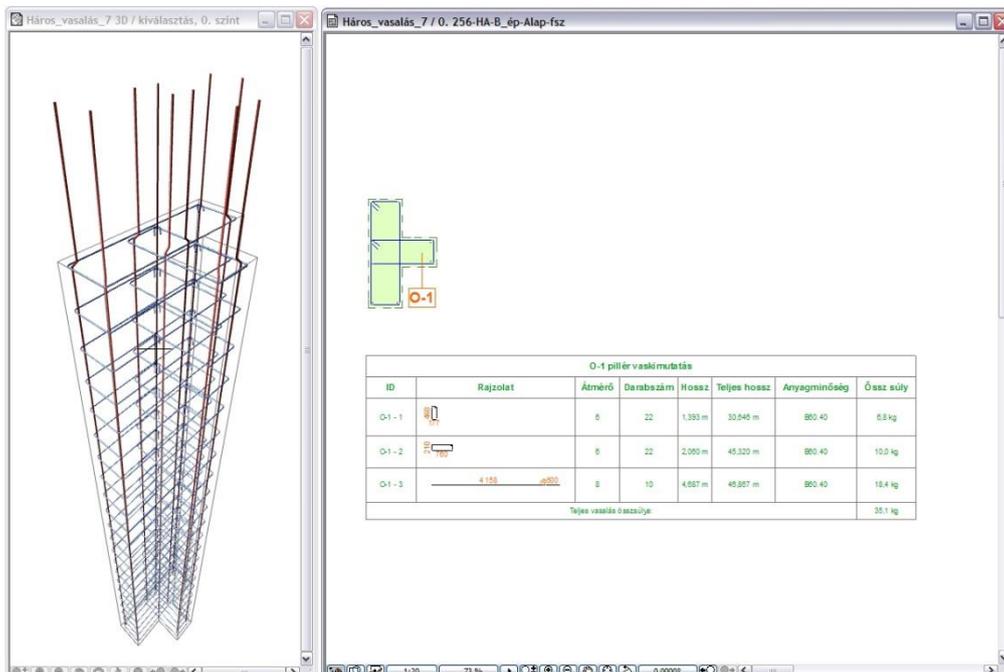
In the opening dialog fill out the name (ID) of the new column (if you choose a name, which already existing, application will ask you to choose a different name). After the definition you can place this column with this name to the model. The rebar calculation table will contains also this name for the components (rebars, stirrups) of this column.



In the dialog, you can see a check box with a name „With List”. If you switch on this check box (default), you can place an individual rebar table about this construction on the floor plan (useful for detail drawings). Click the „OK” button.



Place the column object into the floor plan, at least one copy of it (You can copy this column to multiply it on your plan. It is also suggested to save this object to the „Favorites”). If you requested a cutting list you also can place it into the floor plan. In the column, what you created, all the components’ (rebars, stirrups) ID will contains the column’s name and an individual, internal ID behind. In this moment user can **not** change this names (e.g. If you used **O-1** as a name of the column, the rebars and stirrups number will looks like: **O-1-1, O-1-2, O-1-3...**).



When you place the column into the floor plan, the solution generates an independent detail window. The name of the detail window is the same as the name of the column. In the further working process you can find here the components of the columns, even more, you can create a detail drawing from this column structure in this independent detail window. Use this detail window to change the properties, components or to create a new column structure based on this basic structure (more information about the column changes see 2.6. point).

We suggest you to copy the cutting list into the detail window. Open the „**Vas – complex – X**” section window and copy the front view from the window to the detail window, over the section view. If you need the side view as well, open the „**Vas – complex – Y**” section and copy the side view near to the front view.

To create a full detail drawing from the column reinforcement structure use the ArchiCAD default labels (**rf_label**) or switch on the checkbox in the „**Object Settings**” dialog on the „**2D representation**” tab page. The information content and the shape of the label are also possible to set up in the „**Object Settings**” dialog.

One of the possible detail drawings of columns:

O-1 Column List of Iron							
ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
O-1-1		6	22	1,393 m	30,646 m	B80-40	6,8 kg
O-1-2		6	22	2,060 m	45,320 m	B80-40	10,0 kg
O-1-3		8	10	4,687 m	46,867 m	B80-40	18,4 kg
Total Weight							35,1 kg

We suggest you to save your column into the „**Favorites**”.

Select the column object on the floor plan, open the „**Object Settings**” dialog and push the „**Favorites...**” button on the top of the dialog.

In the next dialog write a name which helps you to recognize your column from other favorite objects. We suggest you to use the same name as you used for the column originally (e.g. **O-1 - Column**).

BEAM DEFINITION

In our system one beam contains at minimum 1 stirrup and 4 rebars. Fewer objects are not enough to define beam structure.

Place one or more „**rf_concrete**” object on the floor plan somewhere in the horizontal range of the sections „**Vas- complex –X**” and „**Y**”. Define the concrete external surface of the beam with these objects. Select all the „**rf_concrete**” objects and set up the concrete cover („**Iron Distance**”) parameter value.

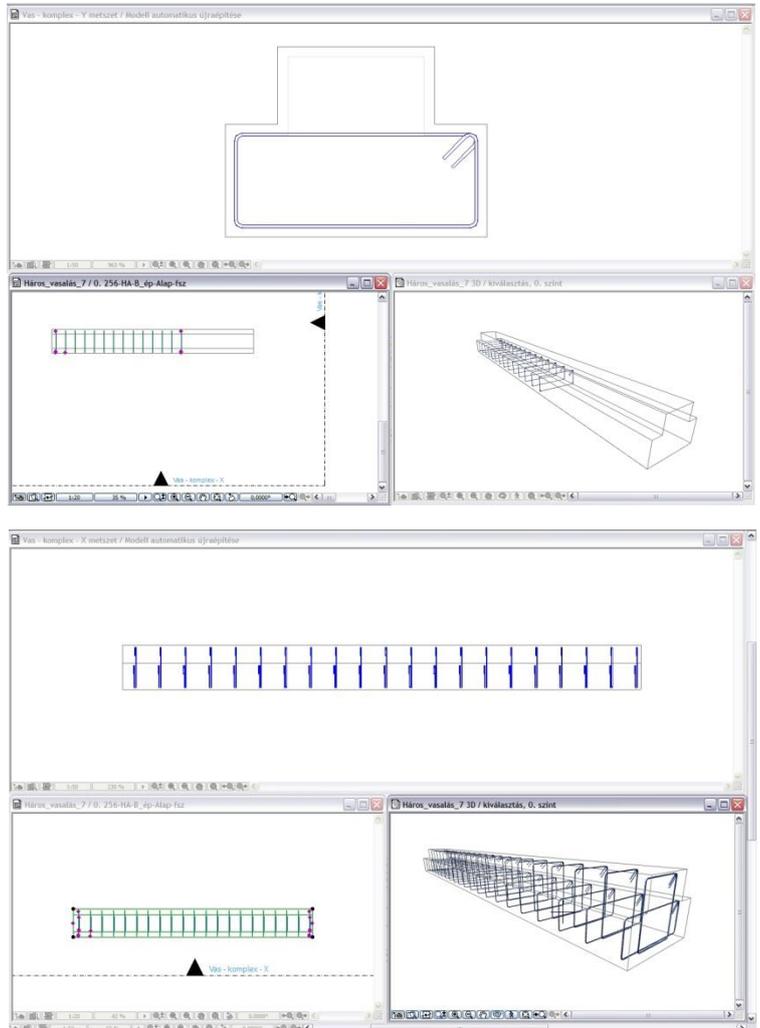
Select the „**rf_stirrup**” objects from the library. Open the „**Object Settings**” dialog and choose the „**Horizontal**” placement option on the „**General**” tab page. Choose shape for stirrup and set up the parameters. Click „**OK**” button and place the stirrup object inside of the „**rf_concrete**”. Connect the stirrup corners to the hotlines and hotspots, which define the safety concrete covers.

If you choose polygon style stirrup, please define the correct shape with the editable hotspots on the section view, connect the polygon corners to the hotspots and hotlines of the „**rf_concrete**” object. Click on the floor plan window.

Use the editable hotspots to define the correct length of the stirrup group and change the stirrup’s distance, if it is necessary (for more information about stirrup placement see 2.3. point).

Place stirrup objects inside of the beam contour which can define the stirrup structure what you need. Define the positions, shapes and stirrup distances with editable hotspots.

When you are ready with the placement of all the stirrups, you can define the rebars.



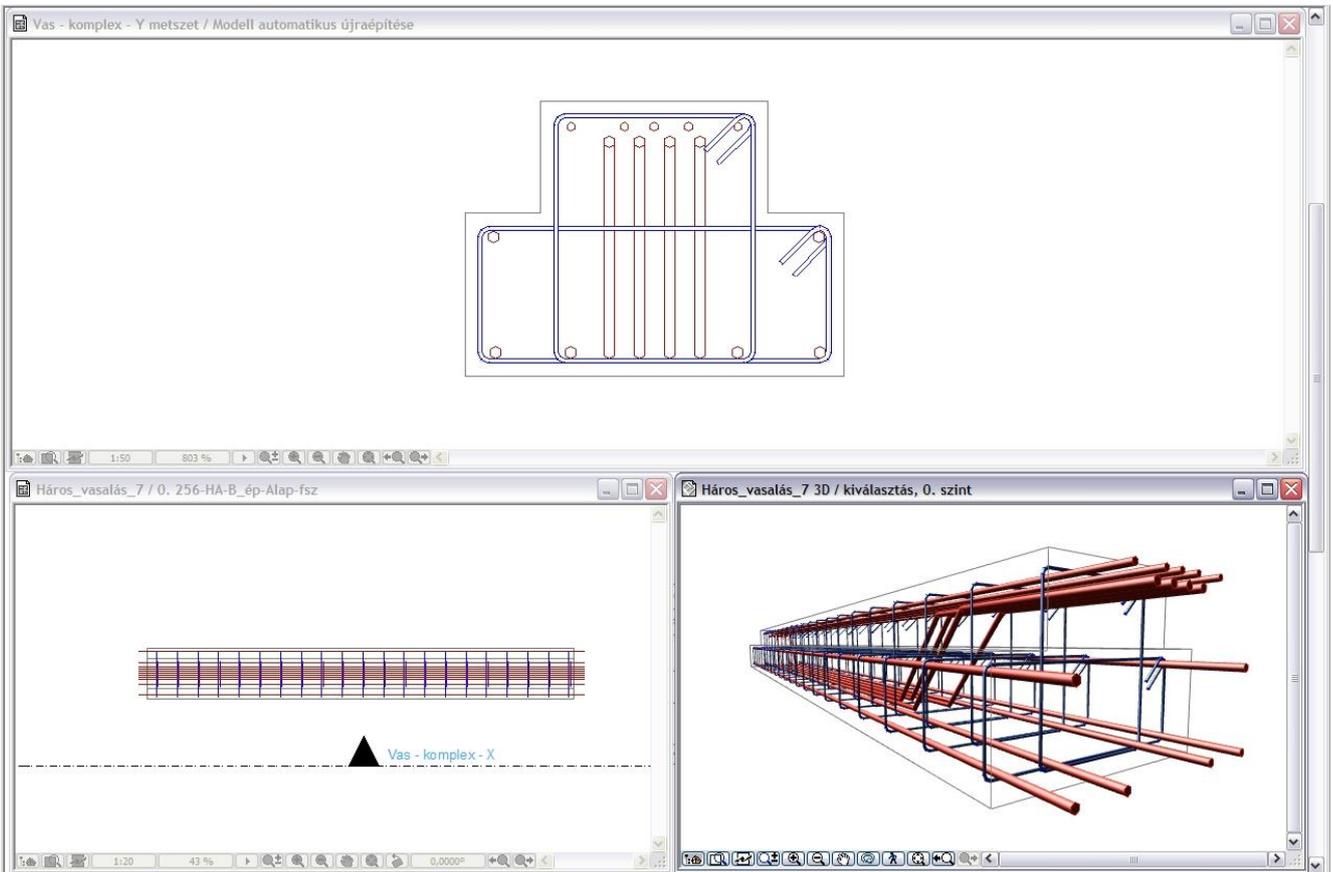
Select the „**rf_rebar**” object from the library. On the User Interface select the „**Horizontal**” placement option and switch on the „**Only one rebar**” check box.

Set up the right style of the endings in the „**Geometry**” tab page and change the parameters. Click on „**OK**” button. Place the rebar object inside the stirrup contour. Click on the Section window and move the rebar on the right position.

Unfortunately there is no possibility to copy or place 3D elements in the section window, if you need more rebars in the beam, go back to the floor plan, copy the existing rebar object and click on section window again. The new object will be available and you can move it to the right position, set up the geometrical parameters with the editable hotspots.

If you need rebars with different parameter sets for the beam, repeat the process above as many time it is necessary.

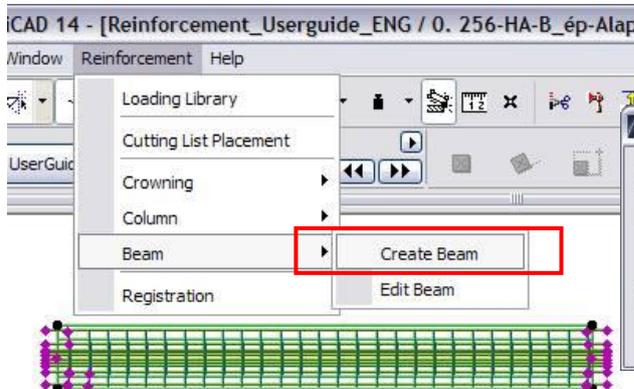
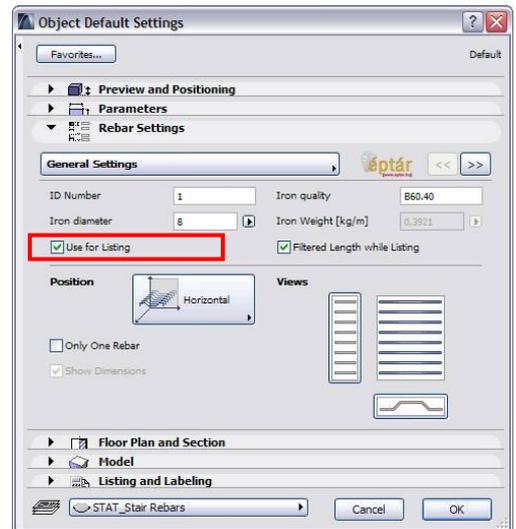




When you are ready with the beam objects rebar structure, select all the rebar, stirrup, concrete elements in the beam and make sure, the „Use for Listing” parameter is checked in.

Select all the rebars, stirrups and concrete elements in your construction on the floor plan (section view is not supported for this command) and choose „Reinforcement / Beam / Create Beam” command.

In the opening dialog fill out the name (ID) of the new beam (if you choose a name, which already existing, application will ask you to choose a different name). After the definition you can place this beam with this name to the model. The rebar calculation table will



contains also this name for the components (rebars, stirrups) of this column.

In the dialog, you can see a check box with a name „With List”. If you switch on this check box (default), you can place an individual rebar table about this construction on the floor plan (useful for detail drawings). Click „OK” button.

Place the beam object into the floor plan, at least one copy of it (You can copy this beam to multiply it on your plan. It is also suggested to save this object to the „Favorites”). If you requested a cutting list you also can place it into the floor plan.

In the beam, what you created, all the components' (rebars, stirrups) ID will contains the beam's name and an individual, internal ID behind. In this moment user can **not** change this names (e.g. If you used **G-1** as a name of the beam, the rebars and stirrups number will look like: **G-1-1**, **G-1-2**, **G-1-3...**).

When you place the beam into the floor plan, the solution generates an independent detail window. The name of the detail window is the same as the name of the beam. In the further working process you can find here the components of the beam, even more, you can create a detail drawing from this beam structure in this independent detail window. You can use this detail window to change the properties, components or to create a new beam structure based on this basic structure (more information about the beam changes see 2.6. point).

We suggest you to copy the cutting list into the detail window. Open the „**Vas – complex – X**” section window and copy the front view from the window to the detail window, over the top view. If you need the section view as well, open the „**Vas – complex – Y**” section and copy the side view near to the side view.

To create a full detail drawing from the beam reinforcement structure use the ArchiCAD default labels (**rf_label**) or switch on the checkbox in the „**Object Settings**” dialog on the „**2D representation**” tab page. The information content and the shape of the label are also possible to set up in the „**Object Settings**” dialog.

One of the possible detail drawings of beam:

ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
G-1-1		8	21	1,292 m	27,134 m	B60-40	0,0 kg
G-1-2		8	21	1,393 m	29,253 m	B60-40	0,5 kg
G-1-3		12	5	4,400 m	21,999 m	B60-40	19,4 kg
G-1-4		16	6	4,400 m	26,399 m	B60-40	41,4 kg
G-1-5		16	2	4,632 m	9,263 m	B60-40	14,5 kg
G-1-6		16	2	4,632 m	9,263 m	B60-40	14,5 kg
Teljes vasalás össz súlya:							102,3 kg

CROWNING DEFINITION

In our system one crowning contains stirrup systems and rebars. Crowning object is able to stretch on the floor plan instead of the beam element which is a fix sized object.

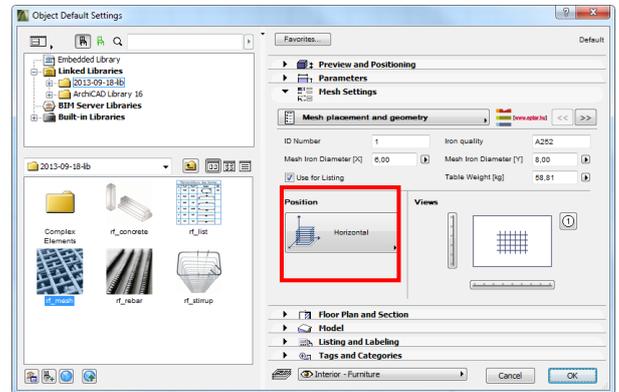
The crowning system needs to contain at minimum one stirrup and two rebars.

Place one or more „rf_concrete” object on the floor plan somewhere in the horizontal range of the sections „Vas- complex –X” and „Y”.

Define the concrete external surface of the crowning with these objects. The crowning system should be built in a **vertical position**, so you can define the section view of the **crowning** on the floor plan (looks like a column).

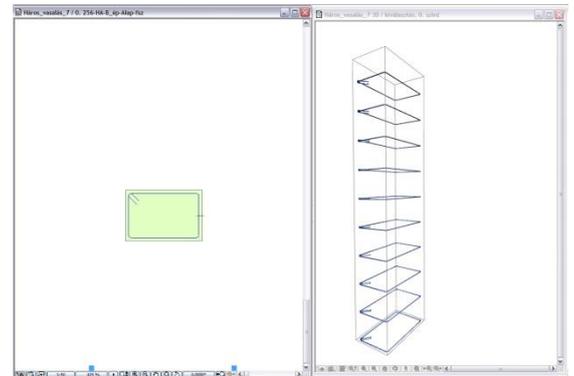
Select all the „rf_concrete” objects and set up the concrete cover („Iron Distance”) parameter value.

Select the „rf_stirrup” objects from the library. Open the „Object Settings” dialog and choose the „Vertical” placement option on the „General” tab page. Choose shape for stirrup and set up the parameters. Click „OK” button and place the stirrup object inside of the „rf_concrete”. Connect the stirrup corners to the hotlines and hotspots, which define the safety concrete covers. If you choose polygon style stirrup, please define the correct shape with the editable hotspots on the floor plan view, connect the polygon corners to the hotspots and hotlines of the „rf_concrete” object.

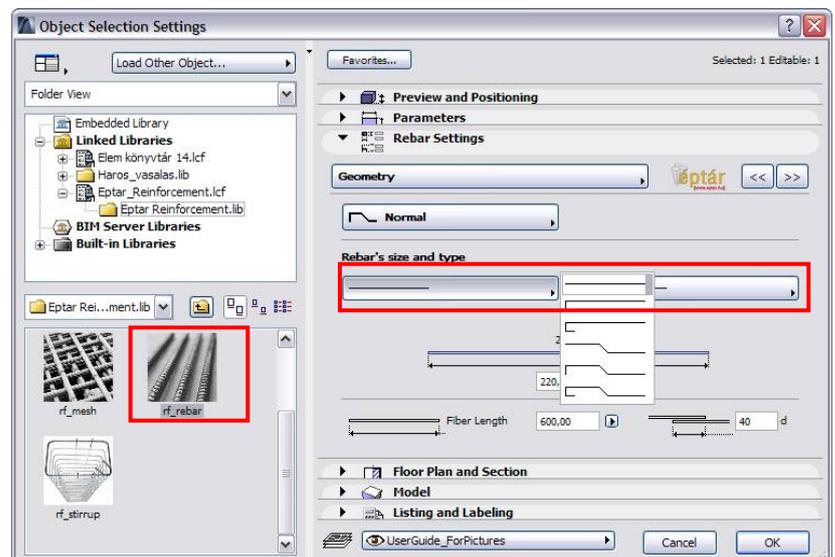


note:

In the crowning definition you can place more stirrup groups in the field, but the software will only distinguish the shape of the stirrup groups, the placement position and the distance between the stirrups will be the same, independently from your design in the 3D model. Use more stirrups only to define more complex stirrup (crowning) sections.

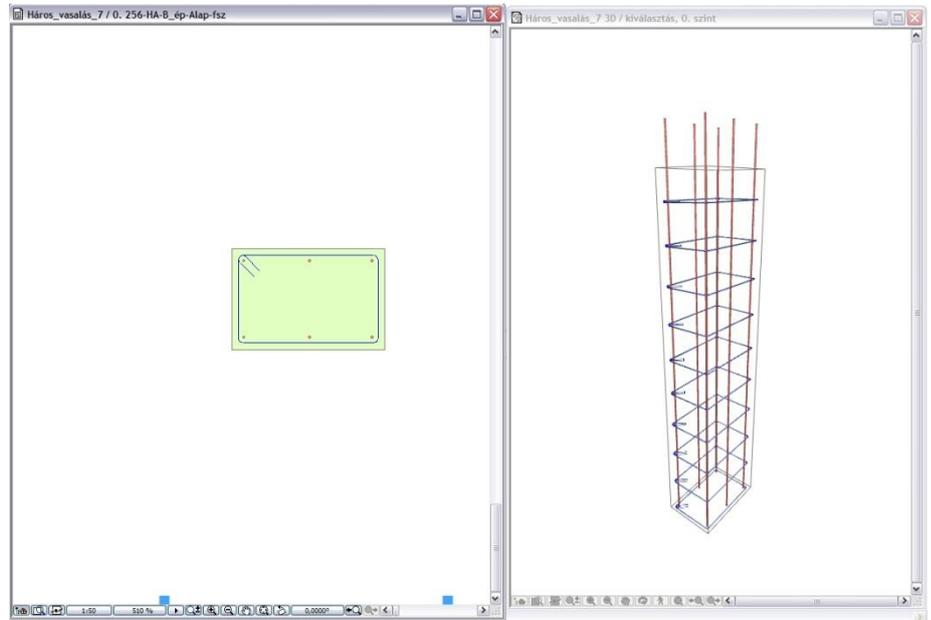


Select the „rf_rebar” object from the library. On the User Interface select the „Vertical” placement option and switch on the „Only one rebar” check box. Set up the „Straight” style of the endings in the „Geometry” tab page. Click on „OK” button. Place the rebar object inside the stirrup contour. Click on the Section window and move the rebar on the right position with the right length.



If you need more rebars in the crowning, go back to the floor plan view and multiply your rebar object on the floor plan to the right positions.

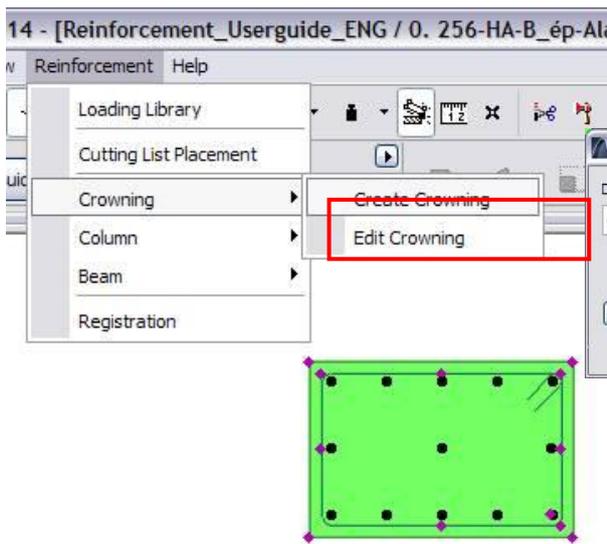
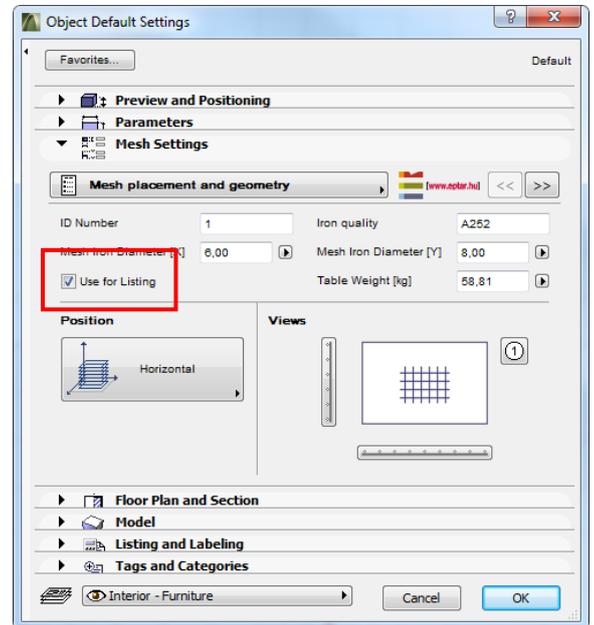
If you need rebars with different parameter sets for the beam, repeat the process above as many time it is necessary.



When you are ready with the beam objects rebar structure, select all the rebar, stirrup, concrete elements in the crowning and make sure, the „Use for Listing” parameter is checked in.

Select all the rebars, stirrups and concrete elements in your construction on the floor plan (section view is not supported for this command) and choose „Reinforcement / Crowning / Create Crowning” command.

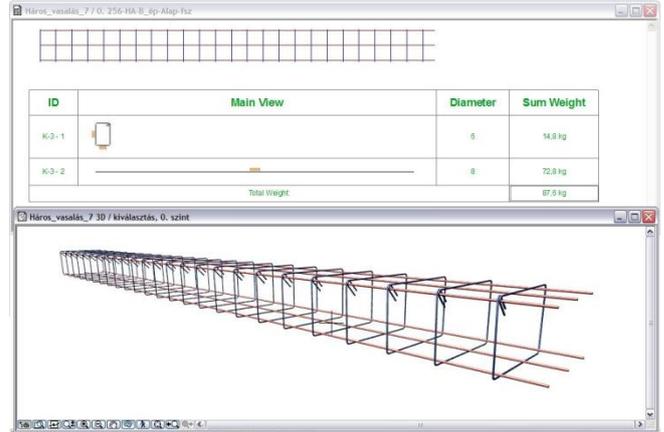
In the opening dialog fill out the name (ID) of the new crowning (if you choose a name, which already existing, application will ask you to choose a different name). After the definition you can place this crowning with this name to the model. The rebar calculation table will contains also this name for the components (rebars, stirrups) of this crowning.



In the dialog, you can see a check box with a name „With Cutting List”. If you switch on this check box (default), you can place an individual rebar table about this construction on the floor plan (useful for detail drawings). Click „OK” button.

Place the crowning object into the floor plan, at least one copy of it (You can copy this crowning to multiply it on your plan. It is also suggested to save this object to the „Favorites”). If you requested a cutting list you also can place it into the floor plan.

In the crowning, what you created, all the components' (rebars, stirrups) ID will contains the crowning name and an individual, internal ID behind. In this moment user can **not** change this names (e.g. If you used **K-1** as a name of the crowning, the rebars and stirrups number will looks like: **K-1-1, K-1-2, K-1-3...**).

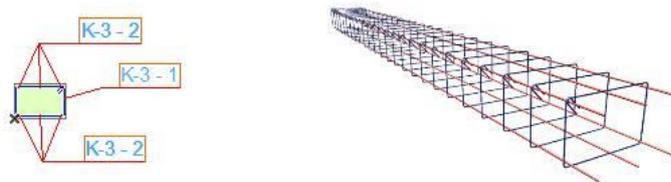


When you place the crowning into the floor plan, the solution generates an independent detail window. The name of the detail window is the same as the name of the crowning. In the further working process you can find here the components of the crowning, even more, you can create a detail drawing from this crowning structure in this independent detail window. You can use this detail window to change the properties, components or to create a new crowning structure based on this basic structure (more information about the crowning changes see 2.6. point).

To create a full detail drawing from the beam reinforcement structure use the ArchiCAD default labels (**rf_label**) or switch on the checkbox in the „Object Settings” dialog on the „2D representation” tab page. The information content and the shape of the label are also possible to set up in the „Object Settings” dialog.

On the floor plan (3D model) the crowning object is one group of rebars. These objects contain editable hotspots on the edges. With the help of these hotspots, you can edit the length and the rebar’s overhanging. You can also use the standard ArchiCAD labels or the implemented function to show the name of the complex element. In this case the labels will show the crowning name (**K-1**) and the lengths of the full construction (rebars).

One of the possible detail drawings of crowning:



ID	Main View	Diameter	Sum Weight
K-3-1		6	14,8 kg
K-3-2		8	72,8 kg
Total Weight:			87,6 kg

2.6 Edit Objects

EDITING METHODES IN GENERAL

The changes of any kind of the objects on the 3D model do not has influence to the already placed cutting lists. If you edit any objects on the model or change some parameters on it, we suggest to you to refresh the cutting lists with the command „Reinforcement / Cutting lists placement”.

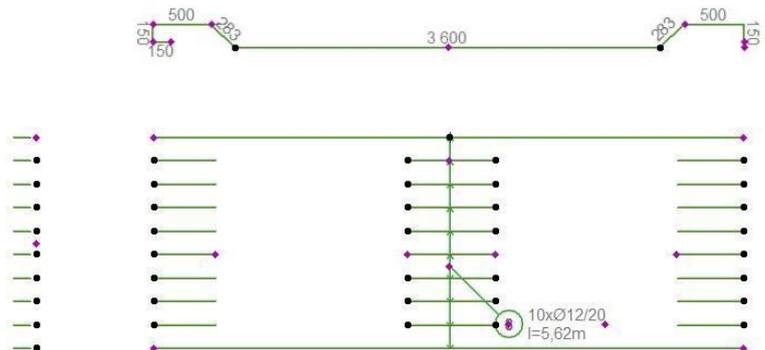
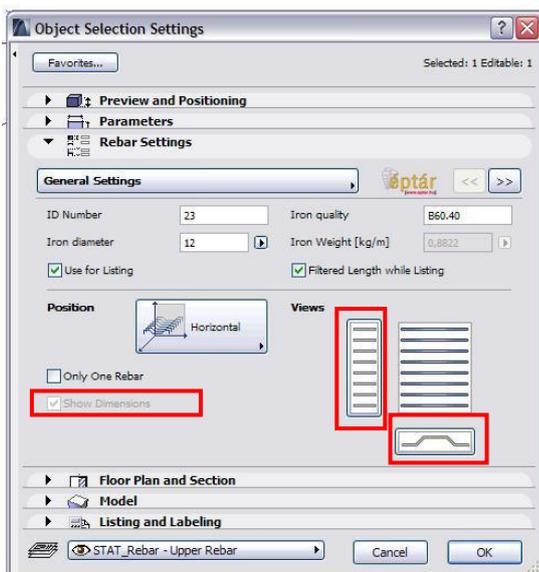
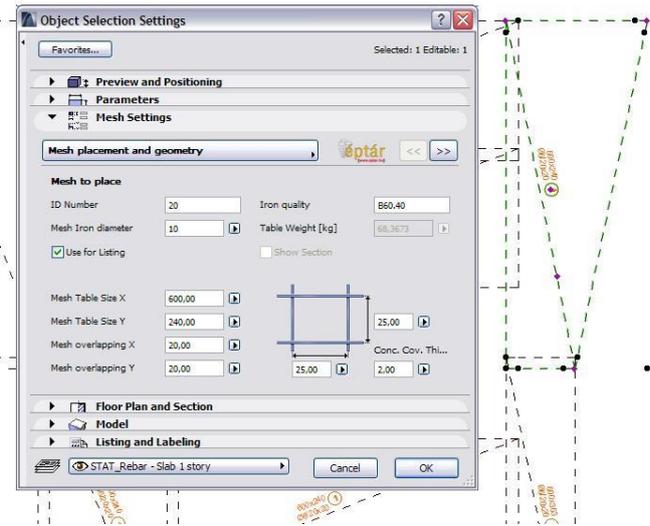
During the refresh of the cutting lists be careful, and consider if you want to refresh all the visible objects on the floor plan or just a few selected ones.

EDIT BASIC OBJECTS

To change the objects placed on the floor plan (3D model), select the object. If the object has editable hotspots, you can change some geometrical data with it.

To change more parameters in the library part, please open the „Object Settings” dialog and change the requested values. Click „OK” button to apply the changes on the object.

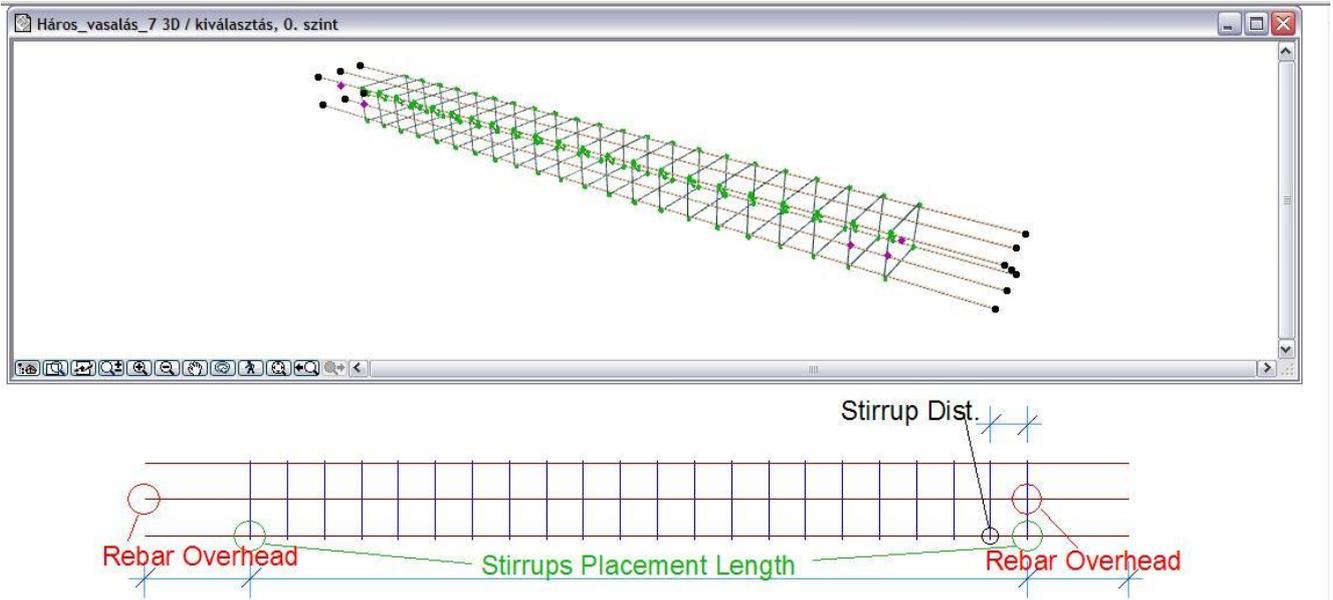
To edit rebar and stirrup objects, it would be helpful if you see the side view (section view) of the element. To see these views open the „Object Settings” dialog, select „General Settings” tab page and switch on the requested views with the icons on the bottom. Click the „OK” button, the external views will be available soon on the floor plan with editable hotspots. These external views are able to move, rotate with the help of the hotspots and you can edit some geometrical values of the object with it. These side views can help a lot when you create a section view of a construction (see more in the 2.8. point).



EDIT COMPLEX ELEMENTS

Edit Crowning object

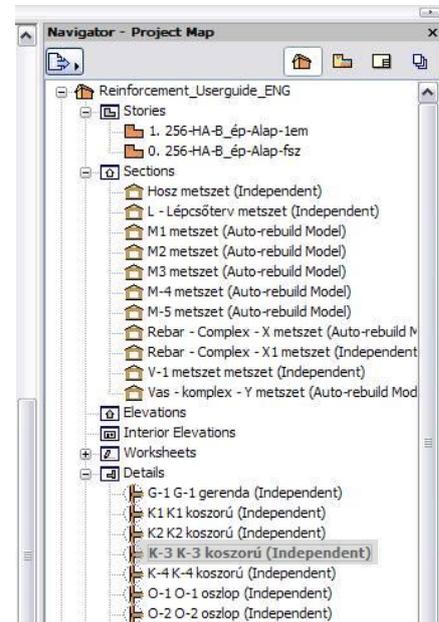
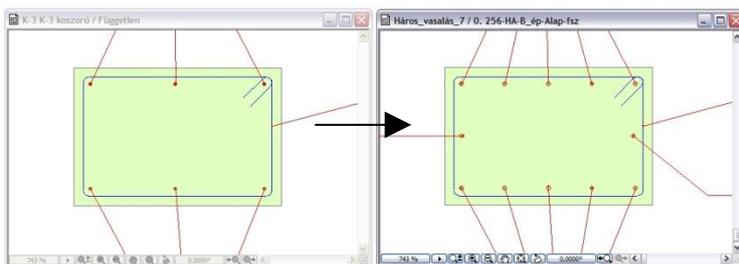
You can edit crowning object on the floor plan and 3D views with the editable hotspots. You can change the full length of the construction, the stirrups' distance, and the rebars overhang from the last stirrup element (necessary for corners).



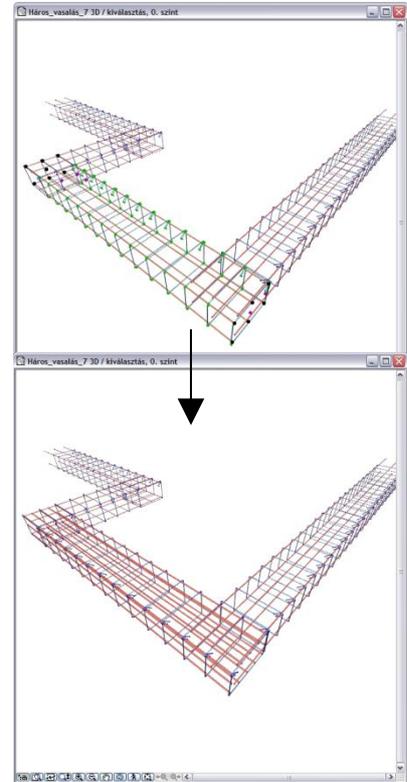
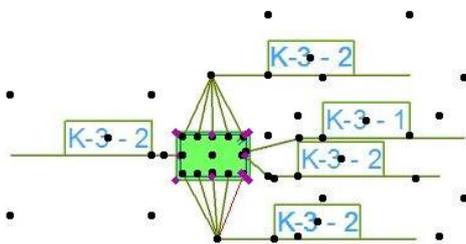
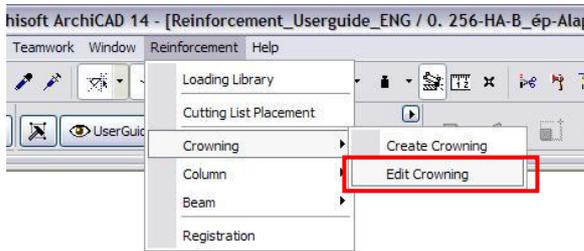
You can check the components of the crowning objects in the „**Objects Settings**” dialog under the „**General Settings**” tab page, but these components are not available to change in this view, you can edit the components only with the help of the command „**Reinforcement / Crowning / Edit Crowning**”.

Select the „**Detail Window**” with the name of the crowning to edit (e.g. K-1). Copy the section view components from the detail window to the Floor plan.

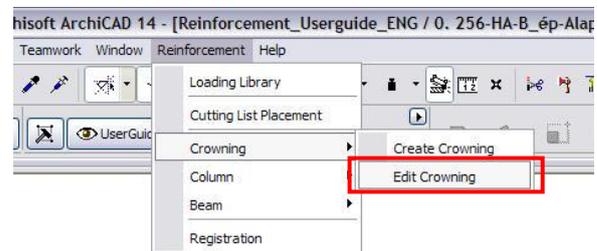
Change the necessary parameters, delete components or add new ones into the construction.



If you do not want to change only a few types of elements on the model, select the objects to change and select the edited crowning components as well. Click on the „**Reinforcement / Crowning / Edit Crowning**” command. The solution will refresh the selected crowning objects with the selected components (basic elements). In this case you created a new crowning structure, but there are some untouched crowning on the model, which are not affected, so you can not overwrite the original crowning number, you have to define a new name.

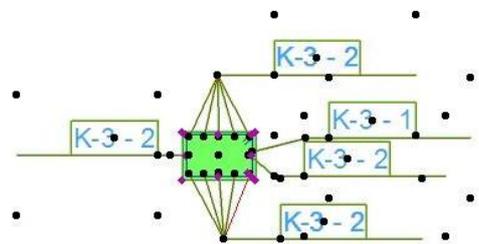
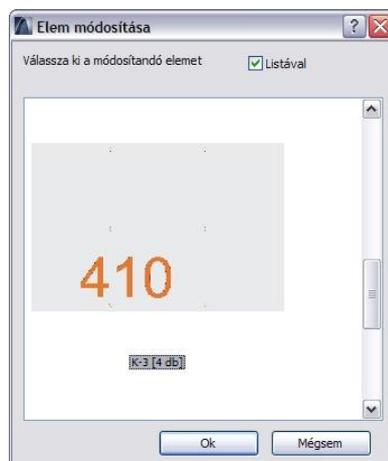


If you want to change all the crowning in the model, select only the crowning components on the floor plan. Click on the „**Reinforcement / Crowning / Edit Crowning**” command. Solution opens a dialog, which shows the available crowning composites on the model. You have to select the number of the crowning structure, which you want to refresh. Click „OK”, the solution will update all the crowning objects (structures) on the floor plan, which has the same number what you selected. Solution also updates the detail windows as well.



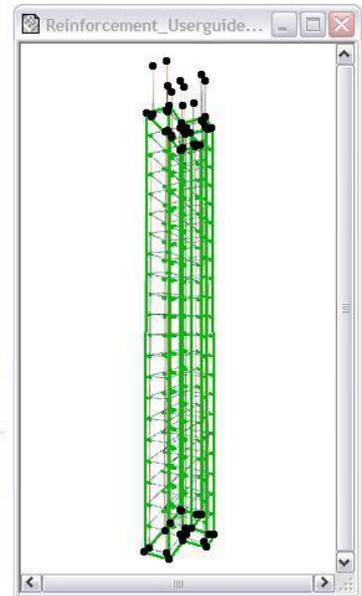
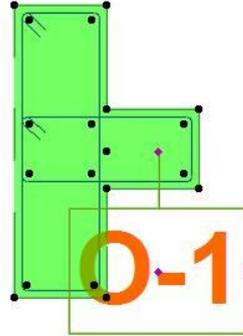
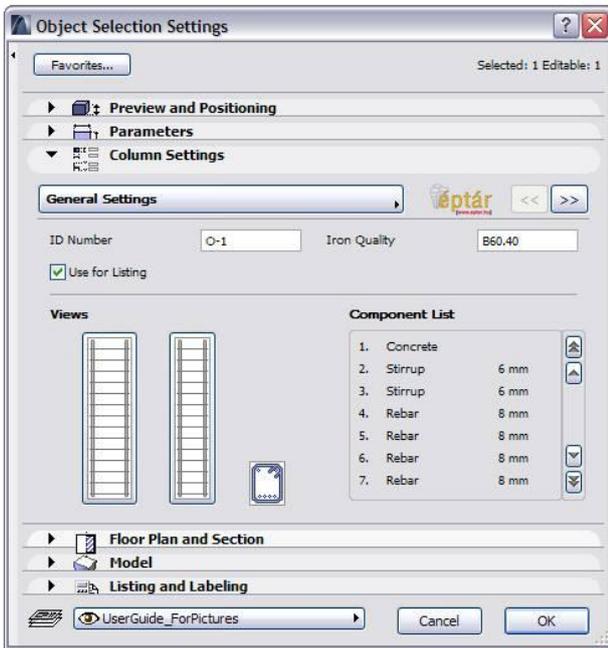
The application does not refresh Side views in the „Detail drawing” window, so we suggest deleting these views and tables and placing them again.

When you update the crowning, the cutting lists, which contain this type of crowning, will be not updated automatically. We suggest you to update the cutting lists after these types of changes.



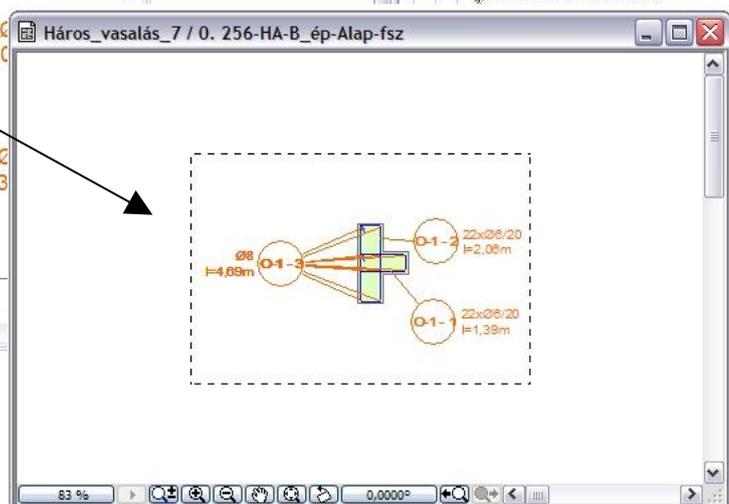
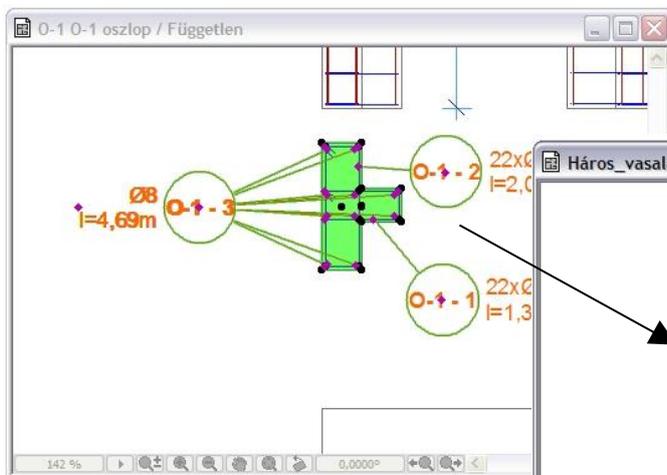
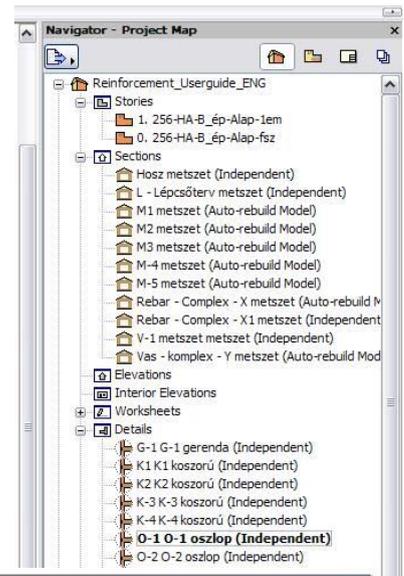
Edit Column object

Columns elements are not possible to edit on the floor plan or on 3D model view. In the „**Objects Settings**” dialog you can change only the pen colors, listed or not listed check box, ID name and some 2D representation mode.



You can check the components of the column objects in the „**Objects Settings**” dialog under the „**General Settings**” tab page, but these components are not available to change in this view, you can edit the components only with the help of the command „**Reinforcement / Crowning / Edit Crowning**”.

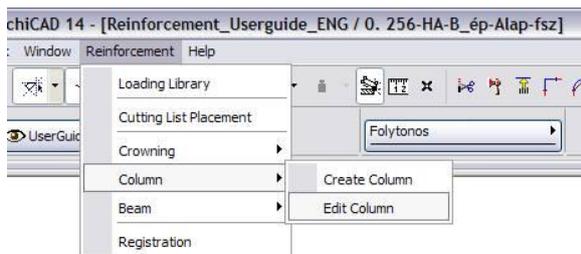
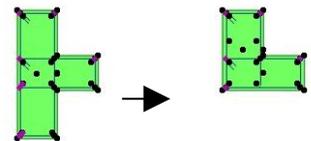
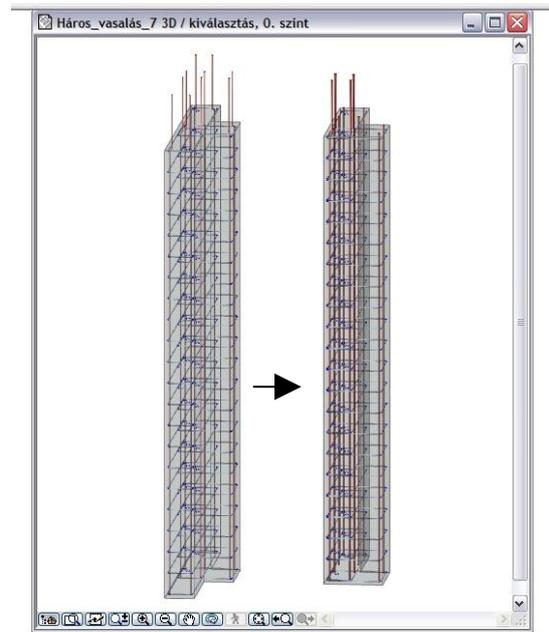
Select the „**Detail Window**” with the name of the column to edit (e.g. O-1). Copy the section view components from the detail window to the Floor plan.

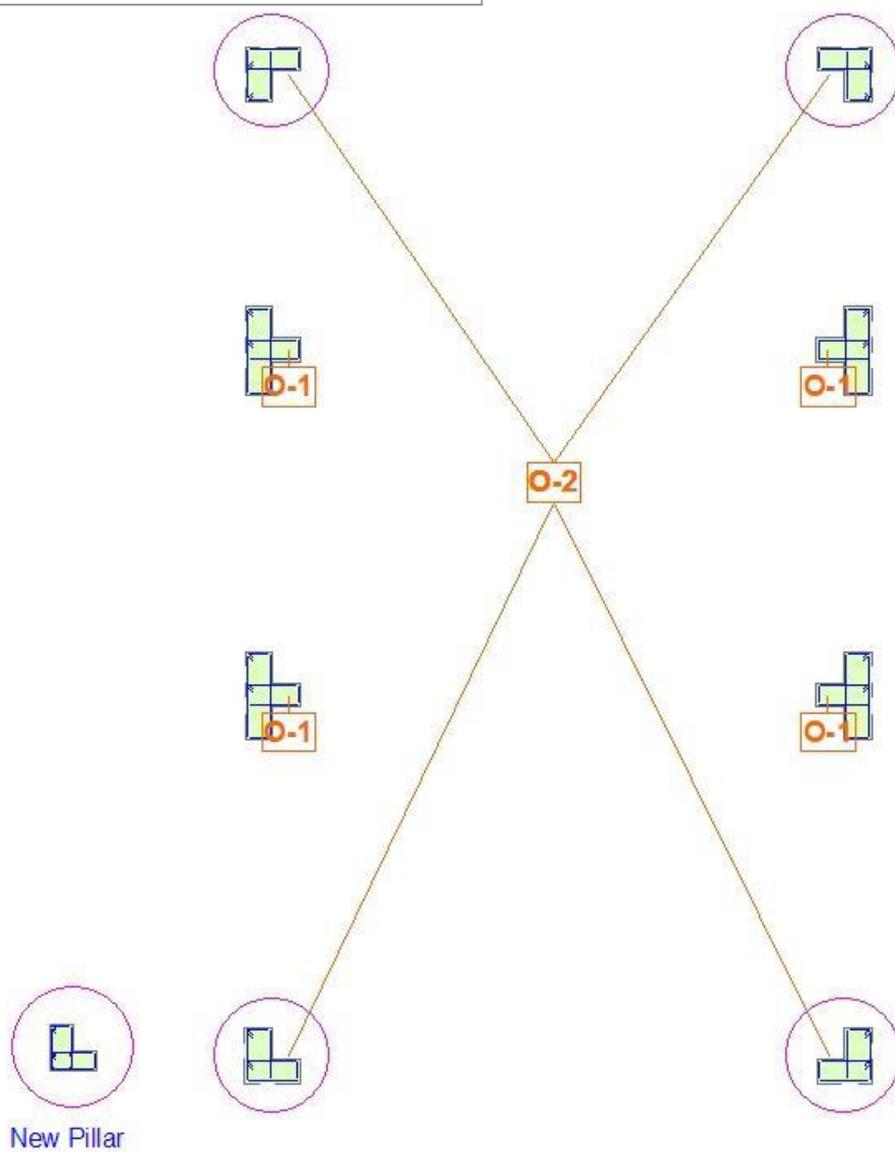
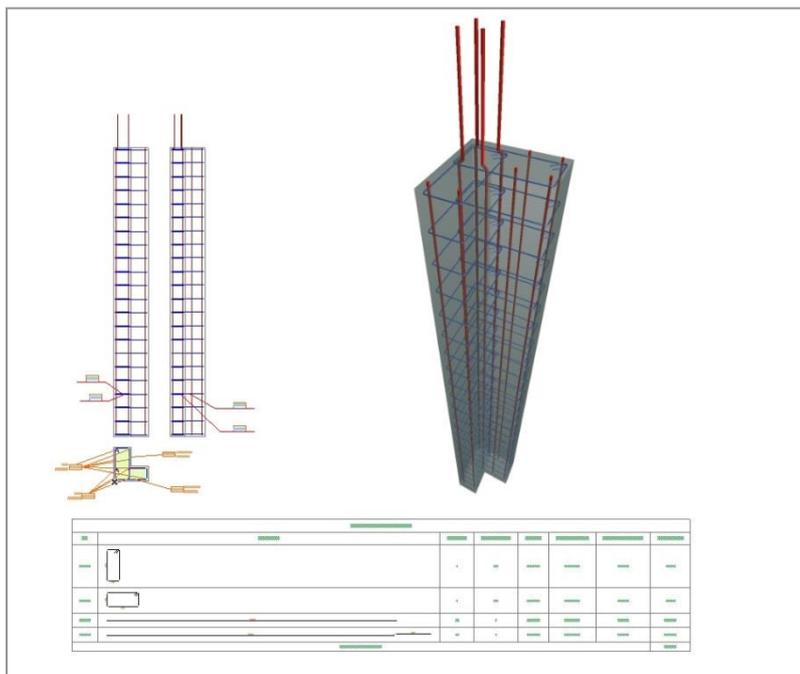


Change the necessary parameters, delete components or add new ones into the construction.

If you want to change only a few pieces of elements on the model, select the objects to change and select the edited column components as well. Click on the „**Reinforcement / Column / Edit column**” command.

The solution will refresh the selected column objects with the selected components (basic elements). In this case you created a new column structure, but there are some untouched columns on the model, which are not affected, so you can not overwrite the original column number, you have to define a new name.

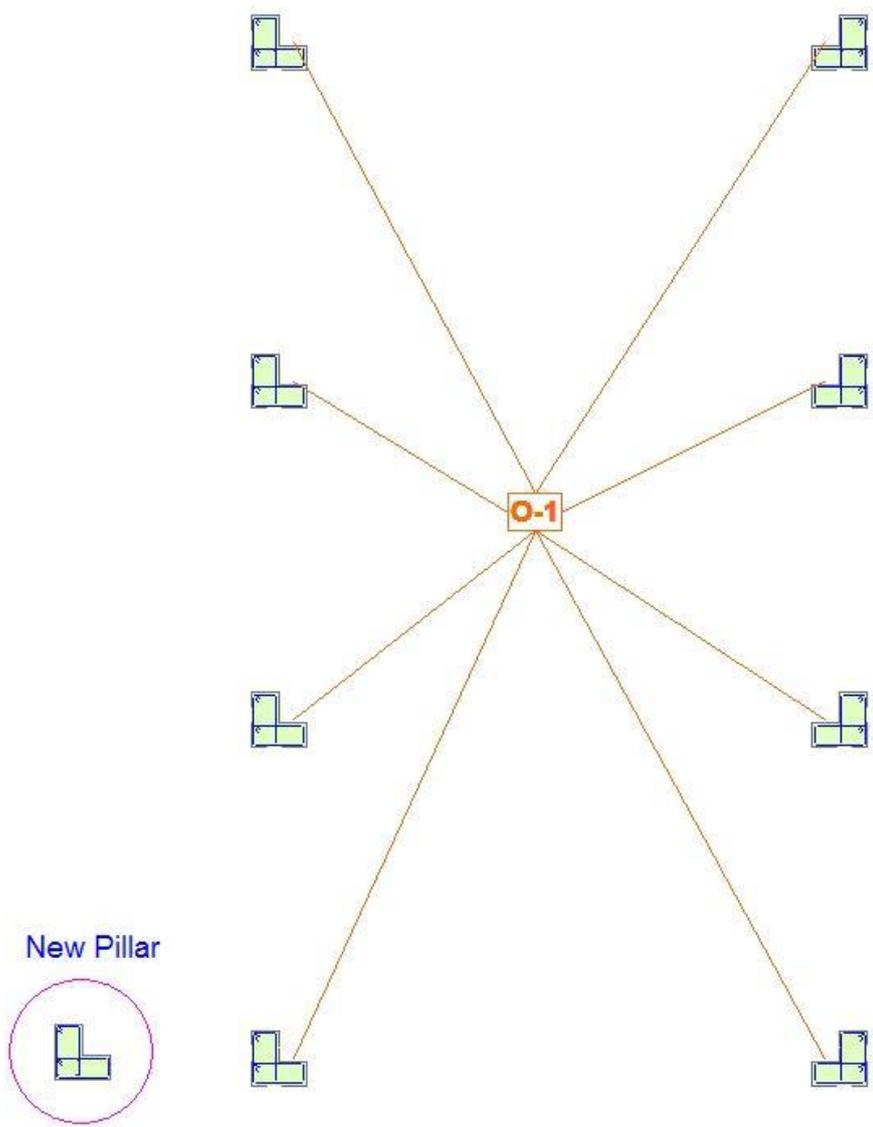
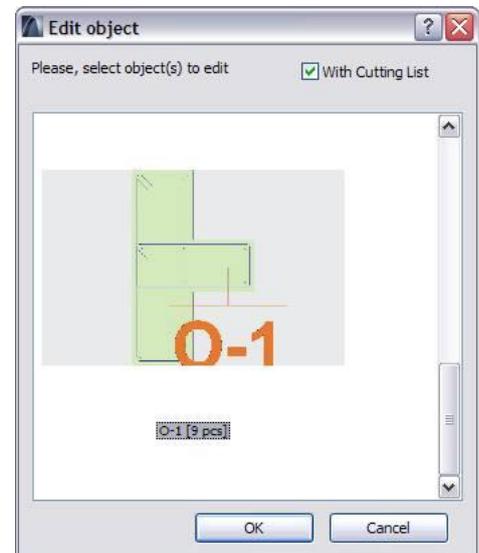




If you want to change all the columns in the model, select only the column components on the floor plan. Click on the „**Reinforcement / Column / Edit Column**” command. Solution opens a dialog, which shows the available column composites on the model. You have to select the number of the column structure, which you want to refresh. Click „OK”, the solution will update all the column objects (structures) on the floor plan, which has the same number what you selected. Solution also updates the detail windows as well.

The application does not refresh Side views in the „Detail drawing” window, so we suggest deleting these views and tables and placing them again.

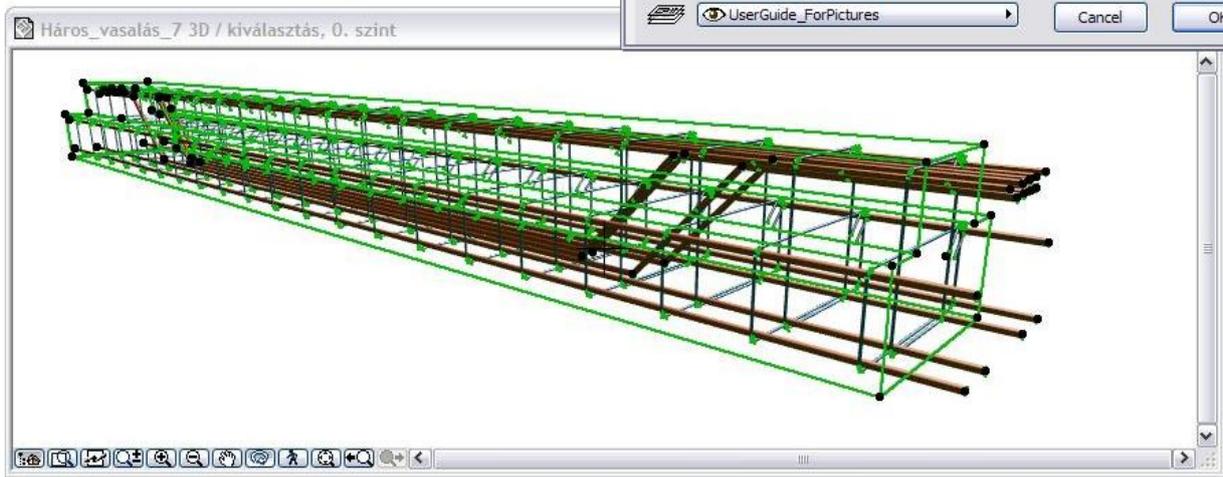
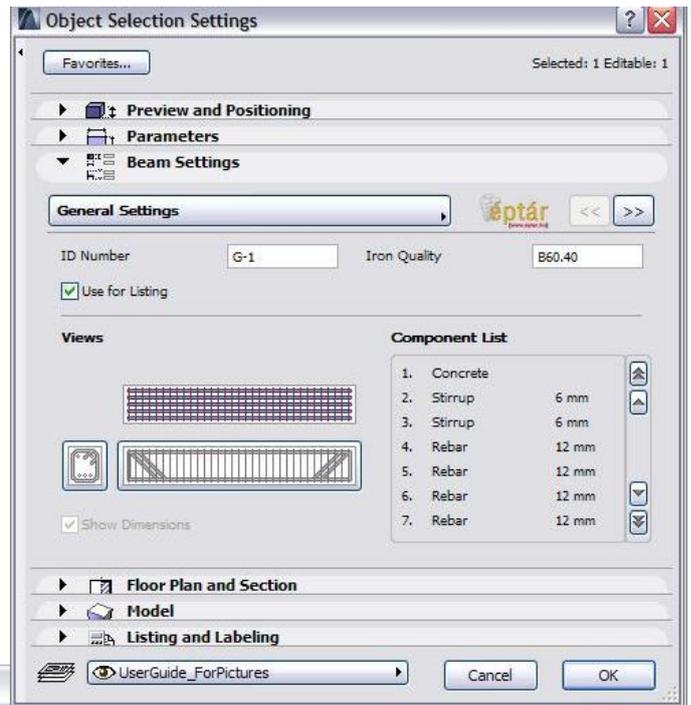
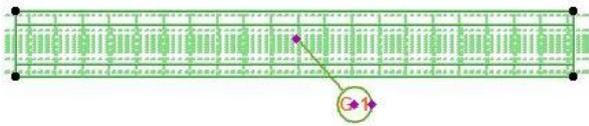
When you update the columns, the cutting lists, which contains these types of columns will be not updated automatically. We suggest you to update the cutting lists after these types of changes.



Edit Beam objects

Beams elements are not possible to edit on the floor plan or on 3D model view. In the „**Objects Settings**” dialog you can change only the pen colors, listed or not listed check box, ID name and some 2D representation mode.

You can check the components of the beam objects in the „**Objects Settings**” dialog under the „**General Settings**” tab page, but these components are not available to change in this view, you can edit the components only with the help of the command „**Reinforcement / Beam / Edit Beam**”.



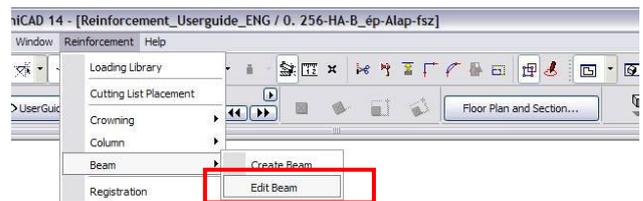
Select the „**Detail Window**” with the name of the column to edit (e.g. G-1). Copy the section view components from the detail window to the Floor plan.

Change the necessary parameters, delete components or add new ones into the construction.



If you want to change only a few pieces of elements on the model, select the objects to change and select the edited beam components as well. Click on the „**Reinforcement / Beam / Edit beam**” command.

The solution will refresh the selected beam objects with the selected components (basic elements). In this case you created a new beam structure, but there are some untouched beams on the model, which are not affected, so you can not overwrite the original beam number, you have to define a new name.



If you want to change all the beams in the model, select only the beam components on the floor plan. Click on the „**Reinforcement / Beam / Edit Beam**” command. Solution opens a dialog, which shows the available beam composites on the model. You have to select the number of the beam structure, which you want to refresh. Click „OK”, the solution will update all the beam objects (structures) on the floor plan, which has the same number what you selected. Solution also updates the detail windows as well.

The application does not refresh Side views in the „Detail drawing” window, so we suggest deleting these views and tables and placing them again.

When you update the columns, the cutting lists, which contains these types of columns will be not updated automatically. We suggest you to update the cutting lists after these types of changes.

2.7 Delete Elements

To delete any rebar construction from the 3D model, you can use the ArchiCAD default process. Select the object on any views and click „DEL” button or select „**Edit / Delete**” command.

On the other hand the deleted objects will not be deleted from cutting list until you do not regenerate them. To clear the deleted objects from the cutting list, please use the „**Reinforcement / List Placement**” command.

2.8 Use ArchiCAD Label element

You can place default ArchiCAD label elements connected to the rebar objects.

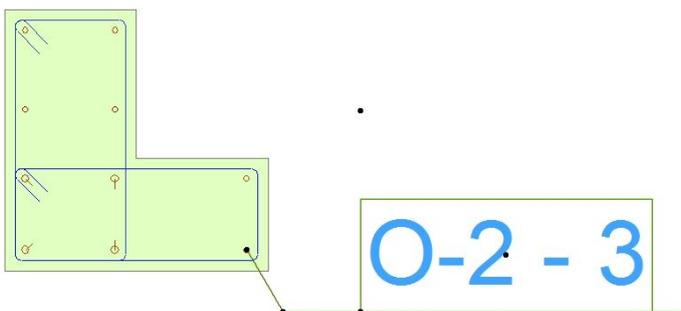
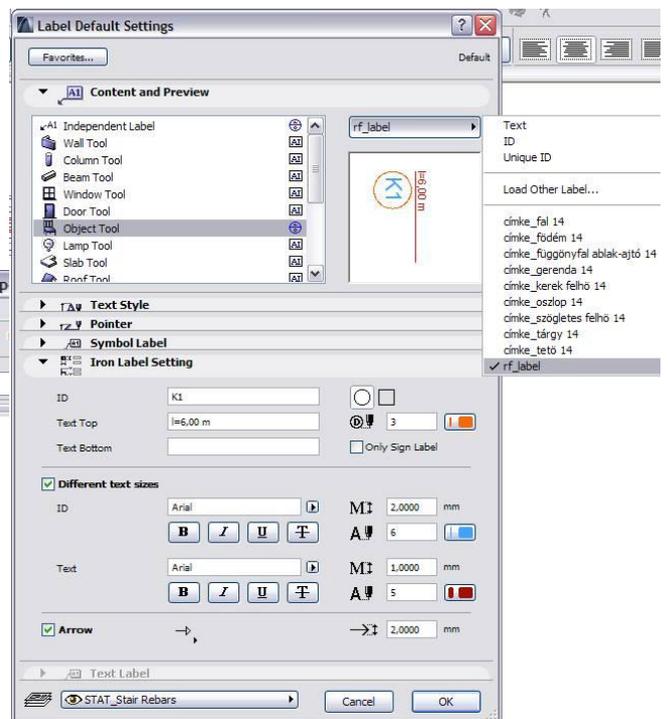
Select the „**Label**” () object from the toolbar. Open „**Label Settings**” dialog and select the „**rf_label**” type for objects. Click on „OK” button.

On the working area select the „**Label connection to objects**” placement method.



Click on a hotspot or hotline of the object on the floor plan to place and connect the label to the object. The label will be placed in the floor plan connected to the object you selected. If you want to move the label or change the text position of it, select the label object and move it's hotspots to the right position.

This Label will be connected to the rebar objects until you delete it, label will update at all parameter changes and label will move with the object together.



2.9 Create Section drawings

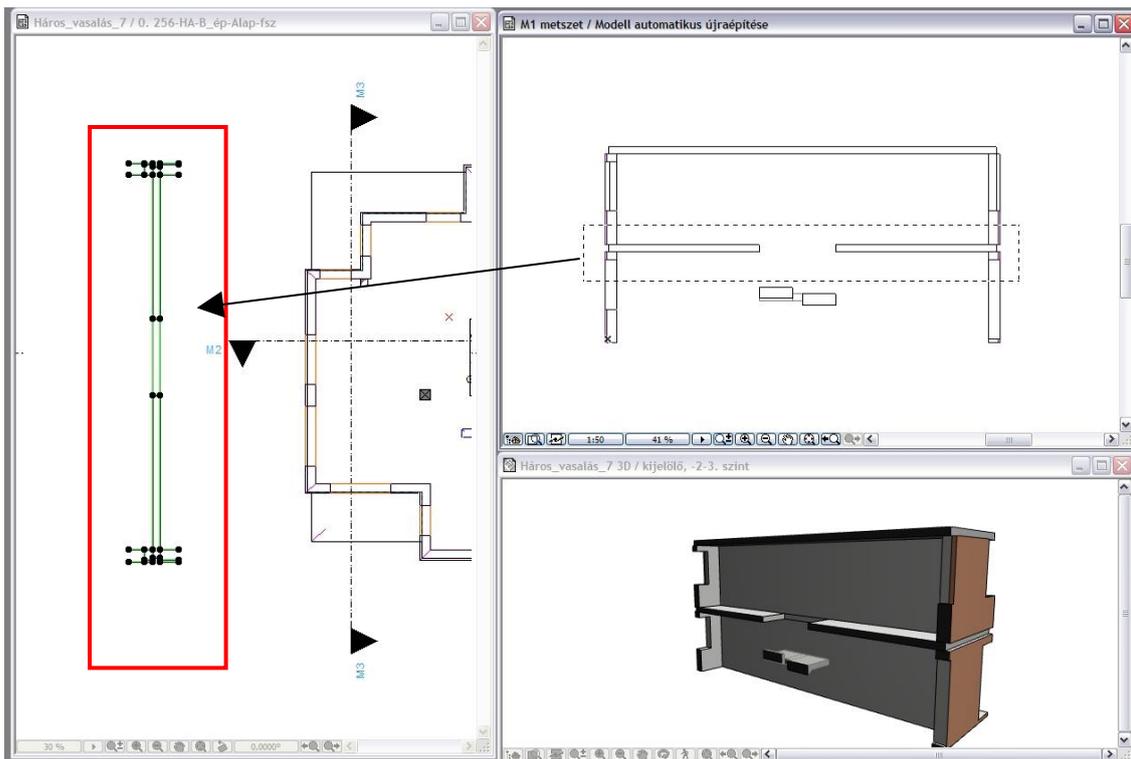
You can create section drawings on two different ways:

- To place the model section view on the floor plan and move the side views of the rebar objects,
- To work on the section view of the model, pick up the parameter sets of rebar objects and place 2D objects to the drawing.

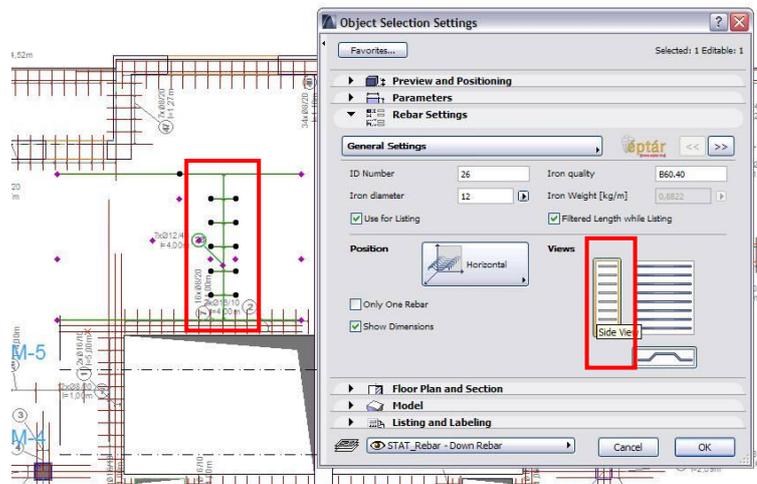
Both processes have benefit and disadvantage, depending on what is your preferred working methods you can use both of them.

Create section view on the floor plan

Create an ArchiCAD section line on the floor plan, where you want to define the section. Open the section view and mark the working area, which shows your affected slab. Copy the contour from the section window and place it to the floor plan, to the right position (in section **Ctrl+C**, on floor plan **Ctrl+V**).



Select one of the rebar objects which cross the section line on the floor plan. Open the „**Object Settings**” dialog and switch on the side view icon on the „**General Settings**” tab page.

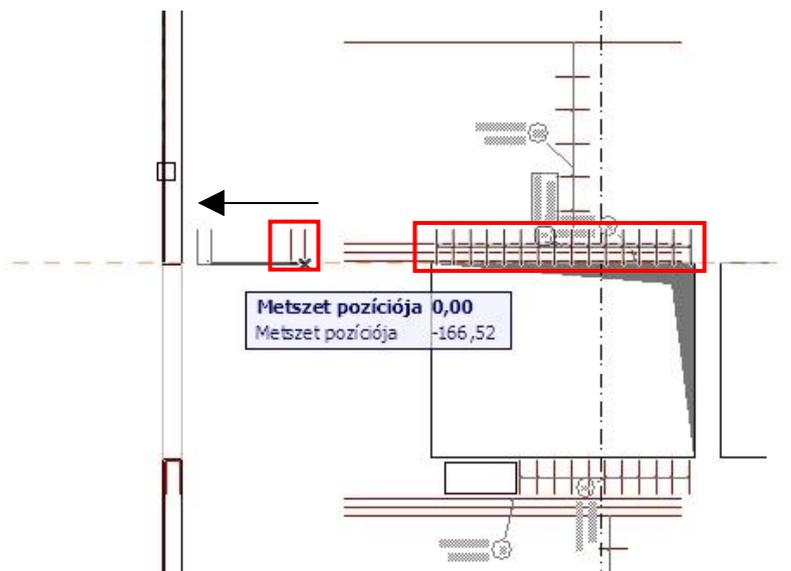


With the help of editable hotspots move the section view of the object to the right position and rotate it to the right direction if it is necessary.

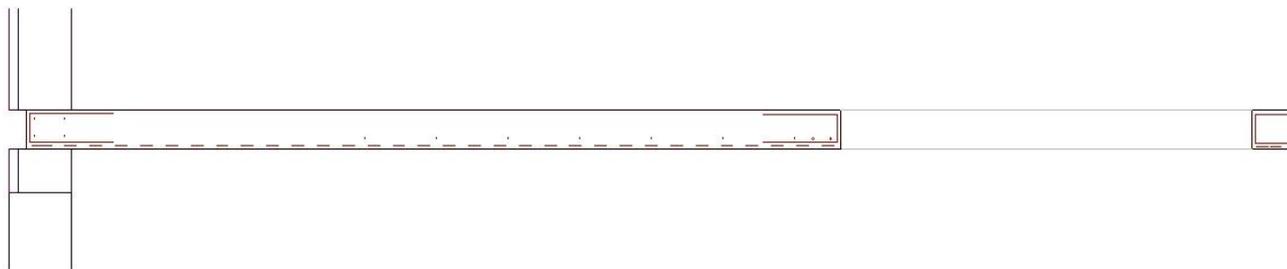
Select a mesh object in the cross section line, open the „**Object Settings**” dialog and switch on the „**Show Section**” check box. Select the direction for the section (X - Y) and click „OK”.

With the help of editable hotspots move the section view of the object to the right position and rotate it to the right direction if it is necessary.

In version 1.0 the mesh which has been placed by Accessory Add-On has no such a section view (check box is grey). In this case draw a single line to represent the mesh on section.

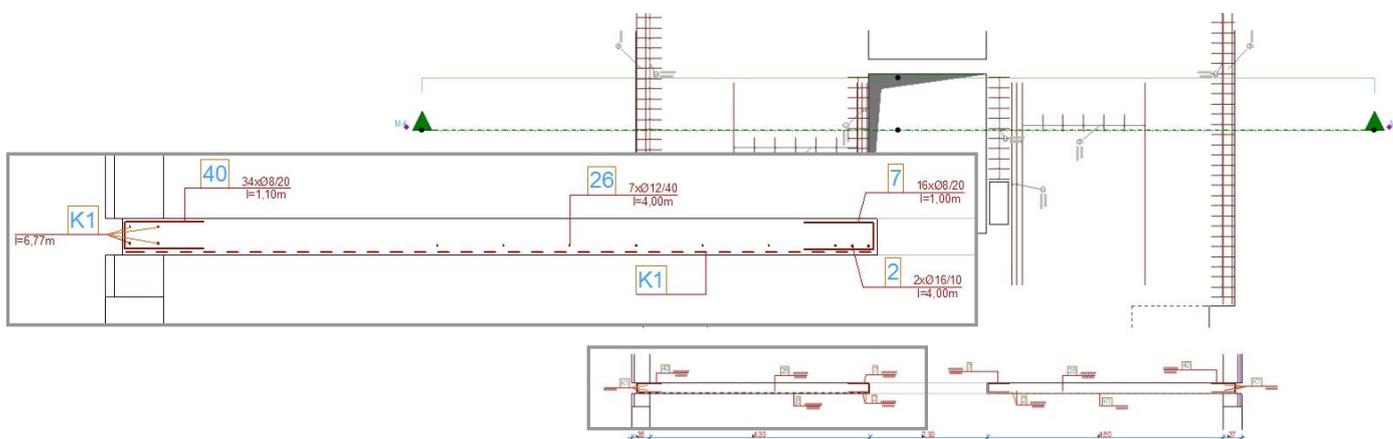
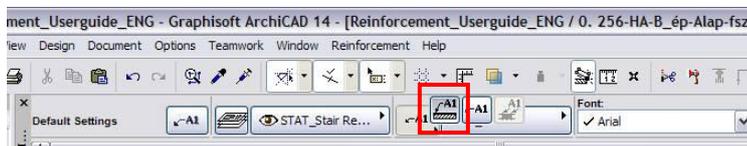


Use the process above to all objects, which is affected in the section view.



To place labels for the section drawing use ArchiCAD label objects. Select the „**Label**” (x-A1 Label) object from the toolbar. Open „**Label Settings**” dialog and select the „**rf_label**” type for objects. Click on „OK” button.

On the working area select the „**Label connection to objects**” placement method. Click on a hotspot or hotline of the object on the floor plan to place and connect the label to the object. The label will be placed in the floor plan connected to the object you selected. If you want to move the label or change the text position of it, select the label object and move it’s hotspots to the right position. This Label wills connected to the rebar objects until you delete it, label will update at all parameter changes and label will move with the object together.



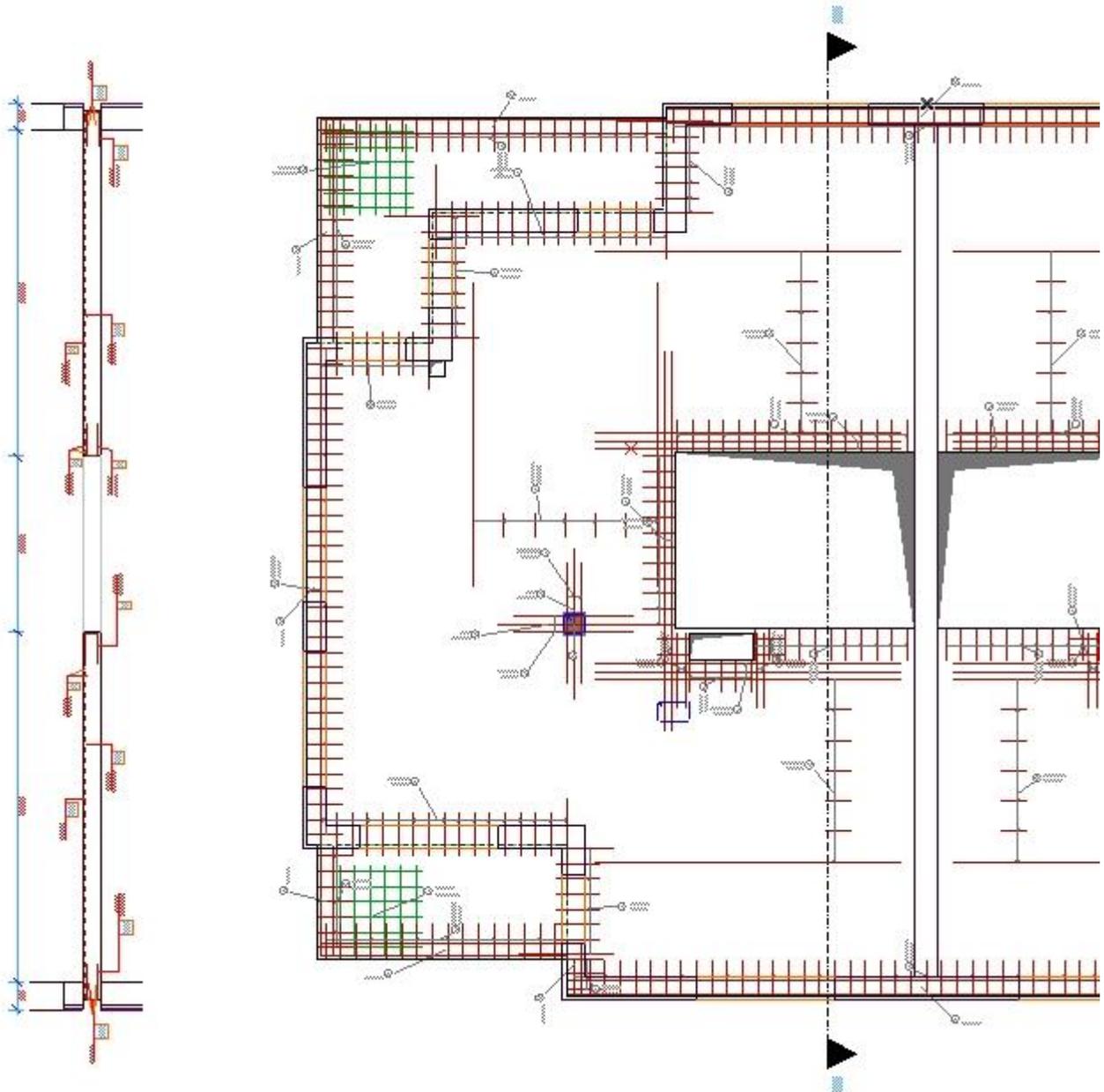
You can do the same label placement for all objects in the section, but you can place independent labels as well.

Benefits, disadvantages:

The benefit is to see all the changes in the section rebars, what you make on the model.

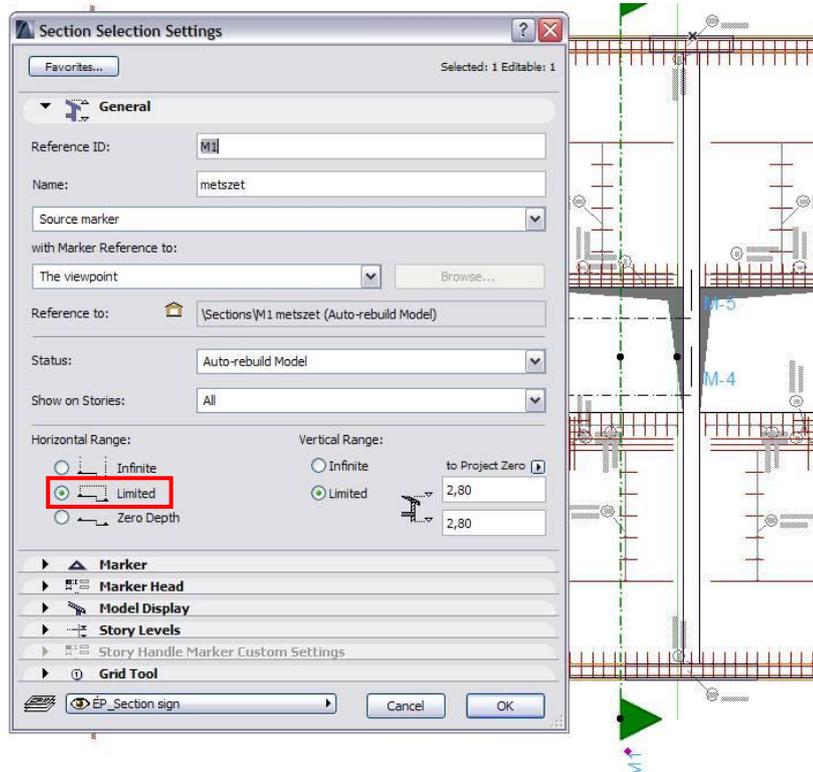
On the other hand, if you move or copy the objects on the model side, the section views will move or multiplied with the objects together, which can cause a confused section drawing.

If you use this section generation method, we suggest you to generate the section drawing at the last phase of the planning process, when you are not plan to change the structures in the model.



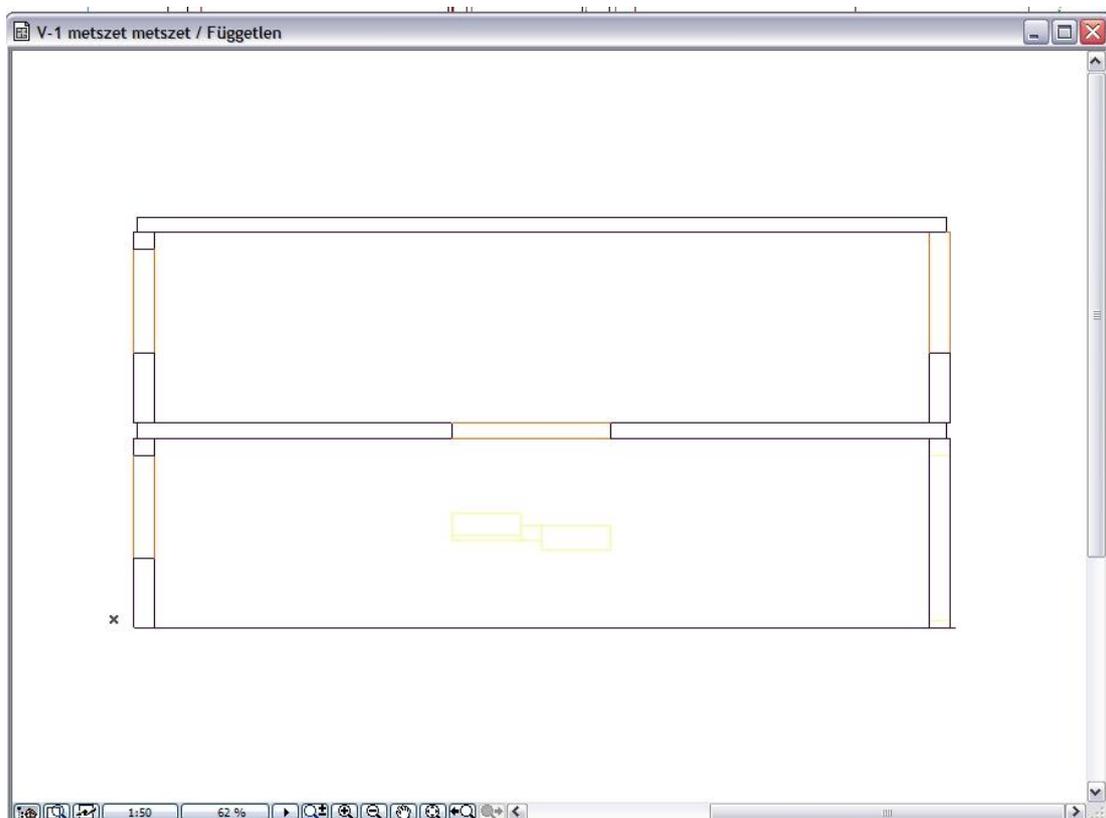
Define section view in Section Window

Create an ArchiCAD section line on the floor plan, where you want to define the section. Rename it to „**M-1**”. Open the „**M-1**” section window. Set up the Section window „**Horizontal Distance**” properties to „**Limited**”. Set up app. 2 m distance on the floor plan.

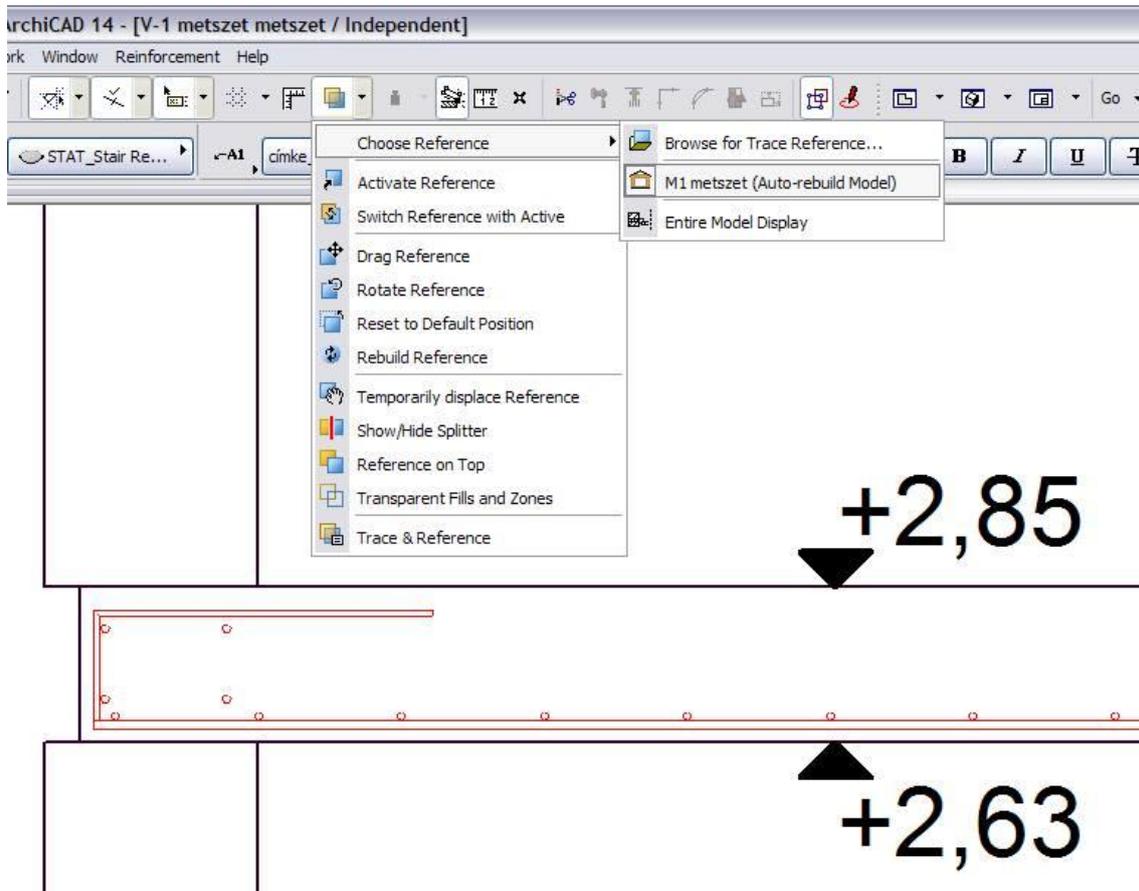


Select and copy the contour lines from the Section Window.

Create an Independent Section Drawing, rename it to „**V-1**”. Place the copied section contour into the „**V-1**” section window.

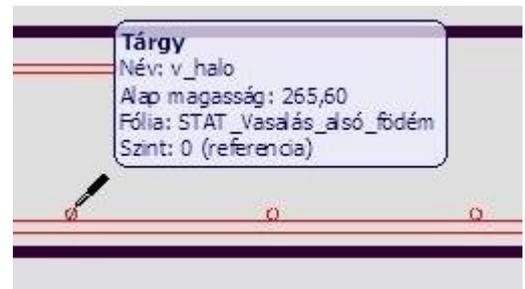


Set up the „M-1” Section as a „Trace” view in Section „V-1”. Open the „M-1” section view and set up the „Layer Sets” to show the slabs in the model in „Wireframe” mode. If you do well, the rebar objects will be visible in the „M-1” section.

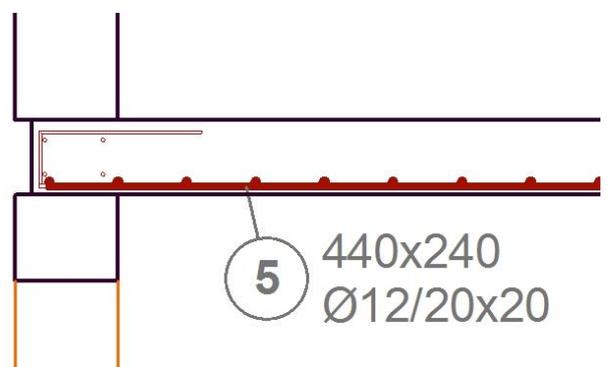


Open the „V-1” section window. On the section view you can see the contour lines of the structures and the 3D section models of the rebar structures (based on the trace drawing). Move your mouse over one of the rebar object and click on it, when you push the **Alt** button (**Alt + Click**).

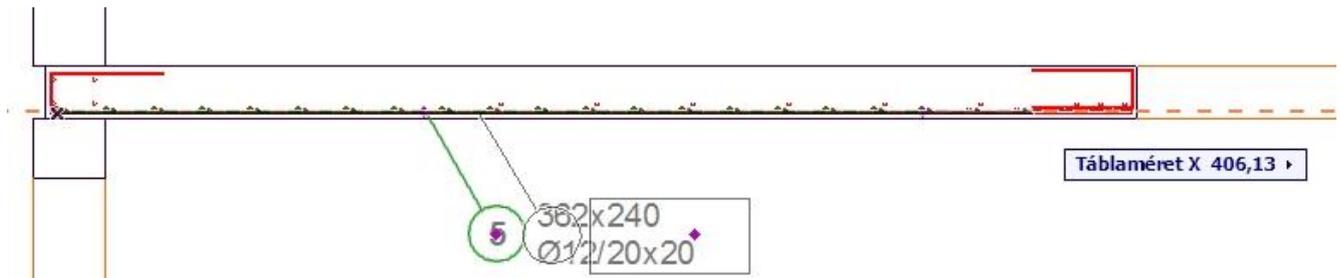
With this process you could copy the parameter settings of that element. Click on the section window (object placement). The section drawing of the earlier selected object will be placed into the section drawing, unfortunately not the right place, but somewhere around it.



Click on the „**View / Zoom / Show all**” command. Find the object placed in the last step. Move the object to the right position and rotate it if it is necessary. Set up the label to fit to your requirements.

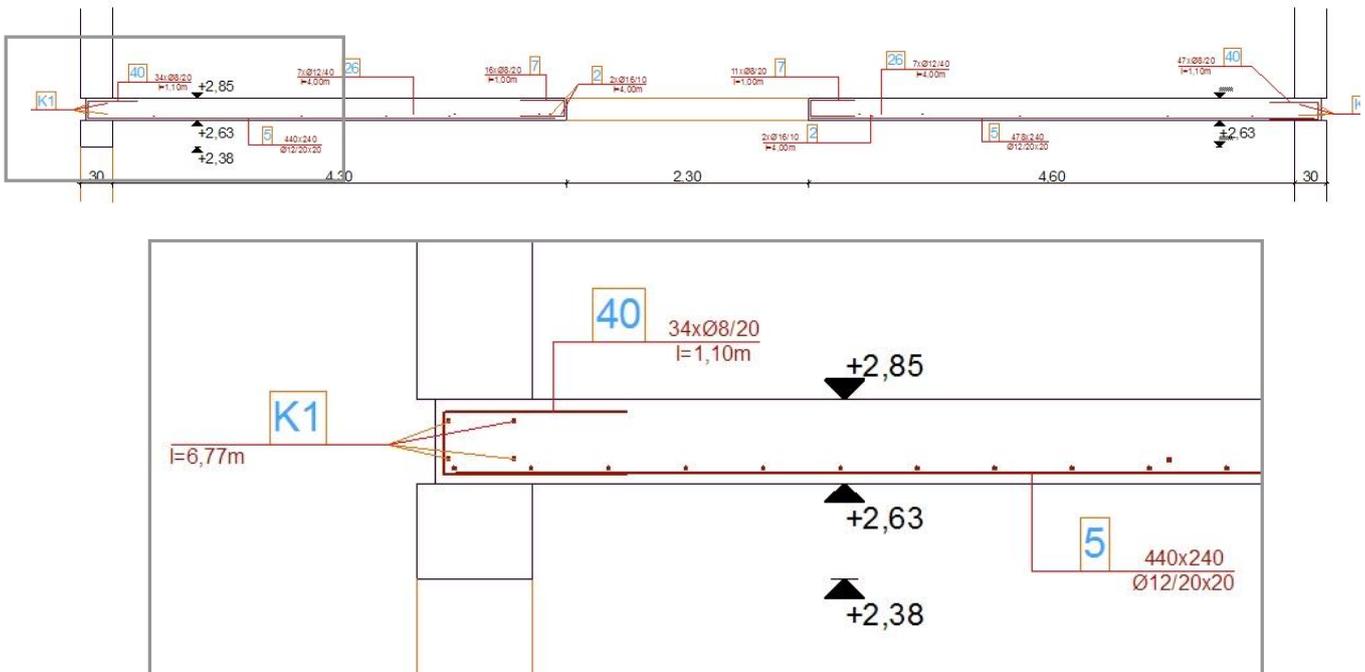


If you choose mesh objects, maybe the mesh section view will be shorter or longer, as the original object section view. In this case, you can edit the length of the mesh's section view with the hotspots on the edges.



Repeat the process above until you do not placed all the elements into the section view, which is necessary to show.

To place labels for the section drawing use ArchiCAD label objects or the objects own represented labels. For more information about the label placement and configuration see 2.8. Points.



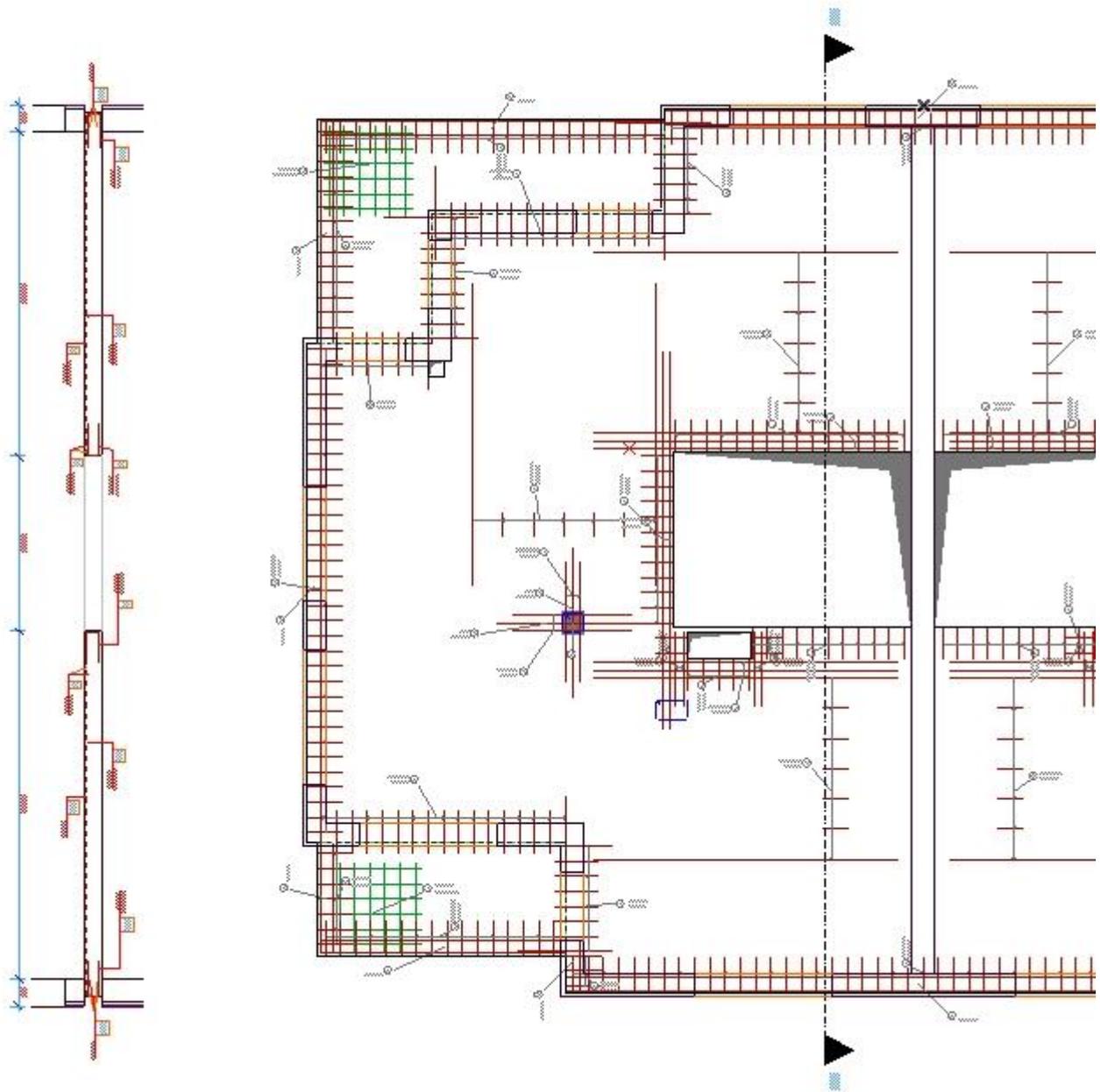
Benefits, disadvantages:

In this solution you follow the ArchiCAD working methods and you generate the section view in the section window, so the section documentation is not in the model space.

On the other hand, the section is not generated from the model directly, if you change anything in the model space, the section view will be not updated automatically, you have to check the changes in all case.

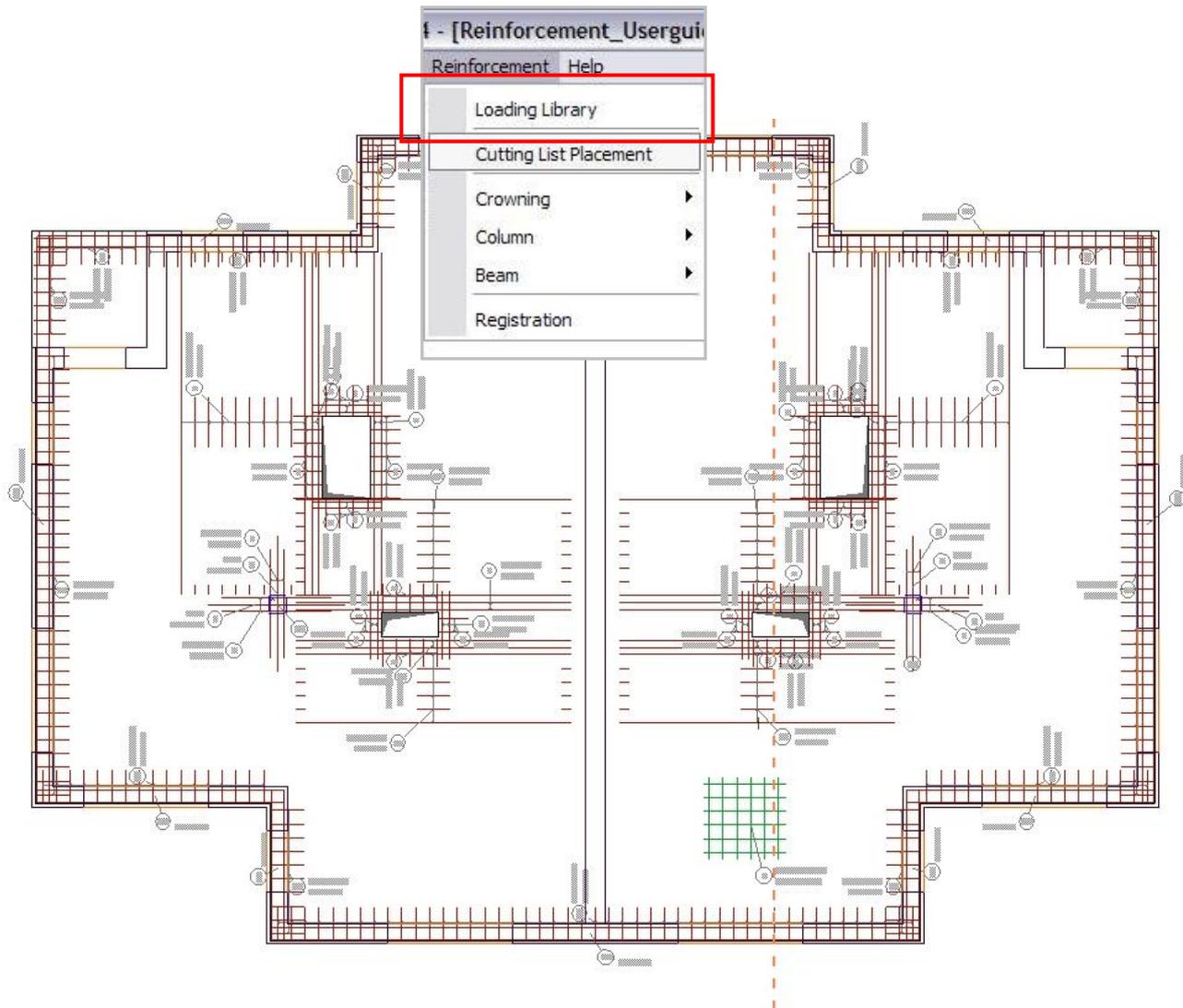
Suggestion:

If you use this section generation method, we suggest you to generate the section drawing at the last phase of the planning process, when you are not plan to change the structures in the model.



Chapter 3.

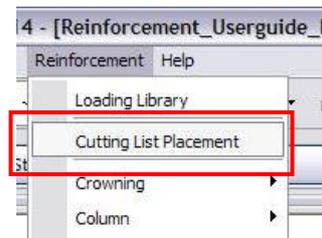
3.1. Cutting list placement and ID check



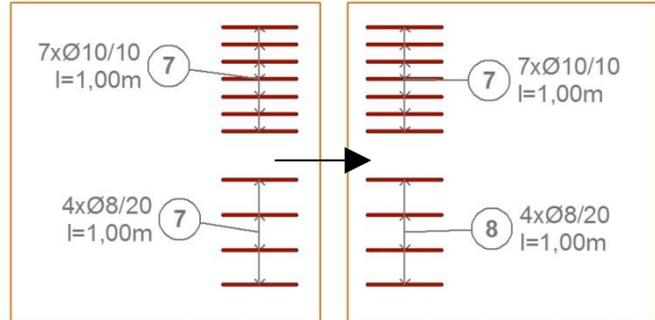
This reinforcement solution helps for the users to create the necessary reinforcement drawings and models, even it generates cutting lists and checks the ID mistakes as well. In the 2.5. point (complex structures) we already write about the cutting list definition, which is available to place into the floor plan. You can place the same cutting list for several construction drawings as well.

Create Cutting list

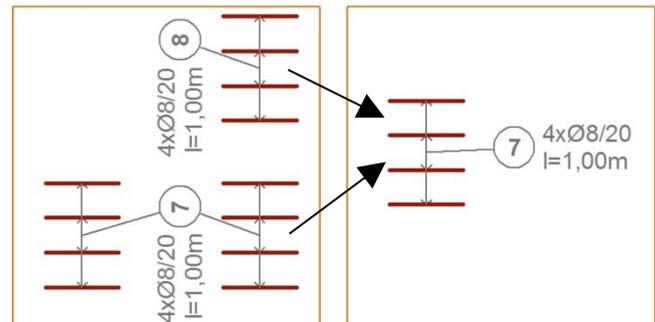
Select the floor plan view which you want to collect in one cutting list. Switch on the necessary layers and hide the unnecessary object layers. Choose the „**Reinforcement / Cutting List placement**” command. Solution will check and collect all the rebars, meshes and complex objects, control the IDs, parameter sets and place a list on the floor plan which contains all the different type rebars in the **visible layers on the active floor plan view**.



In the case, if you placed objects with same ID, but the parameter values (type, diameter, shape, and length) are different, than solution will change the ID number for one of them.



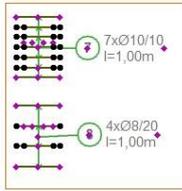
In the case, if you placed objects with different IDs, but the parameter values (type, diameter, shape, and length) are the same, than solution will collect them and use them for the same ID. The used ID will be that one, which ID has been placed more on the floor plan.



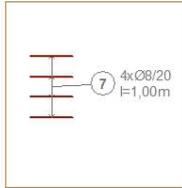
After the automatic calculation and ID check you have to define the position of the cutting list on the floor plan. If you have a cutting list on the floor plan already, the software do not place a new one, just refresh the existing one.

ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1		16	4	5,000 m	20,000 m	B80,40	31,4 kg
2		16	8	4,000 m	32,000 m	B80,40	50,2 kg
26		12	42	4,000 m	168,000 m	B80,40	148,2 kg
3		12	8	1,689 m	13,515 m	B80,40	11,9 kg
4		12	4	2,089 m	8,358 m	B80,40	7,4 kg
40		8	153	1,100 m	168,300 m	B80,40	66,0 kg
5		16	1	1,000 m	10,000 m	B80,40	12,5 kg
6		16	8	1,000 m	8,000 m	B80,40	12,5 kg
7		8	96	1,000 m	96,000 m	B80,40	37,6 kg
9		16	2	4,632 m	9,263 m	B80,40	14,5 kg
K1 - 1		12	24	37,480 m	899,520 m	B80,40	104,3 kg
K2 - 1		12	32	18,080 m	578,560 m	B38,24	86,9 kg
P1 - 1		8	30	1,140 m	34,200 m	B80,40	13,4 kg
P1 - 2		16	16	26,488 m	423,488 m	B80,40	83,0 kg
B80,40 sum: B38,24 sum: Total Weight							2956,0 kg 86,9 kg 3043,6 kg

In the case, if you have more cutting list placed into the actual floor plan view the reinforcement solution will cancel the operation, do not update any of the placed cutting lists. In this situation, please select the cutting lists to update and select all the rebars, which you want to involve into the cutting list. Click the „**Reinforcement / Cutting List placement**” command again. If the selection of the all rebar and one cutting list is difficult on the floor plan, switch off the layers, which contains the unnecessary cutting lists or place them to an independent detail drawings until you run the recalculation method again.

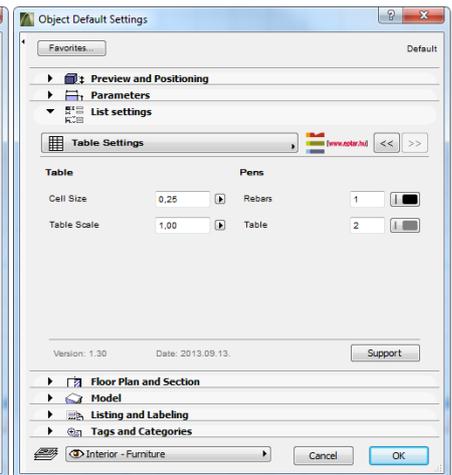
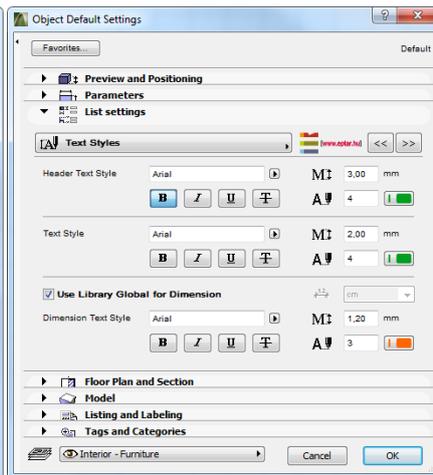
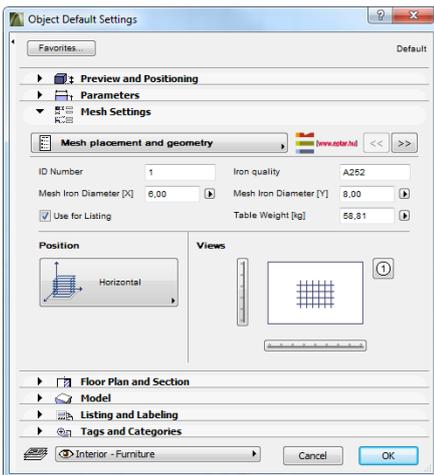


ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
7		10	7	1,000 m	7,000 m	B60.40	4,3 kg
8		8	4	1,000 m	4,000 m	B60.40	1,6 kg
Teljes vasalás össz súlya:							5,9 kg



ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
7		8	12	1,000 m	12,000 m	B60.40	4,7 kg
Teljes vasalás össz súlya:							4,7 kg

You can choose the columns of the cutting list to show on the documentation. Select the cutting list, open the „**Object settings**” dialog and switch on or off the check box in front of the different column names. If you need bigger or smaller rebar drawings, you can customize its size on the „**List Representation**” tab page.



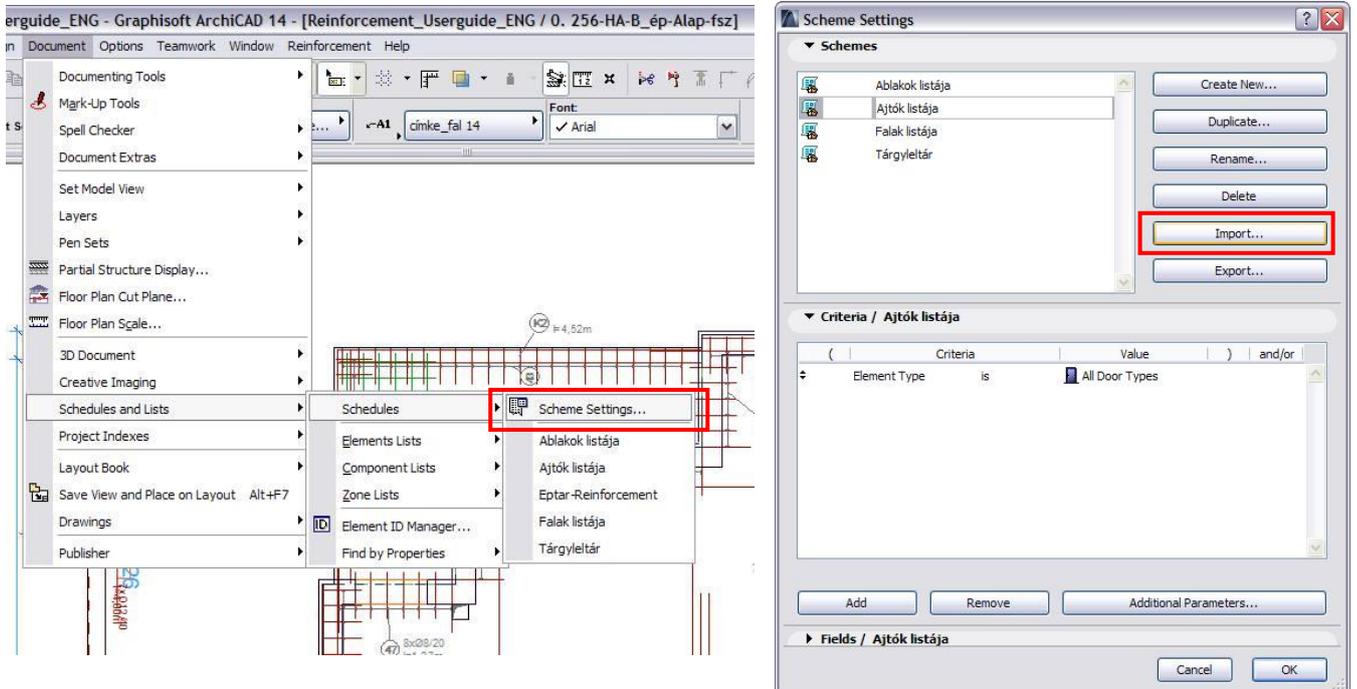
ID	Rajzolat	Darabszám	Össz súly
7		7	4,3 kg
8		4	1,6 kg
Teljes vasalás össz súlya:			5,9 kg

ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
7		10	7	1,000 m	7,000 m	B60.40	4,3 kg
8		8	4	1,000 m	4,000 m	B60.40	1,6 kg
Teljes vasalás össz súlya:							5,9 kg

3.2. Use Interactive Schedule

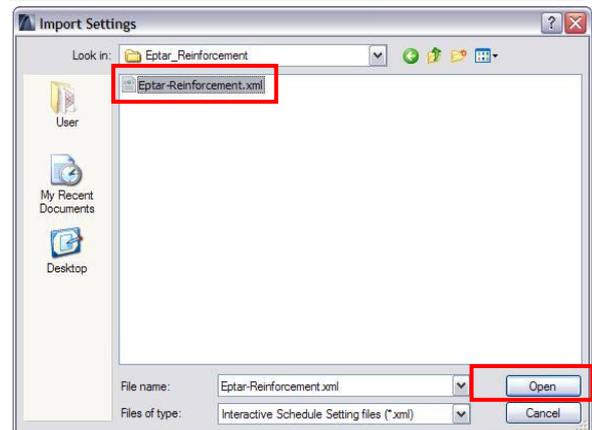
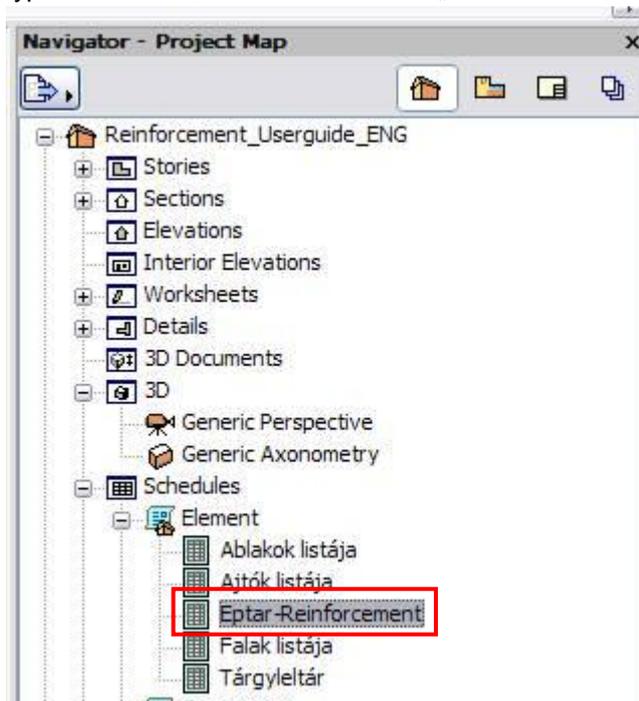
With the help of this implemented ArchiCAD listing method you also can check the ID numbers and the different rebars in your documentation.

Start the Interactive Schedule with the „**Document / Schedules and Lists / Schedules / Scheme Settings**” command.



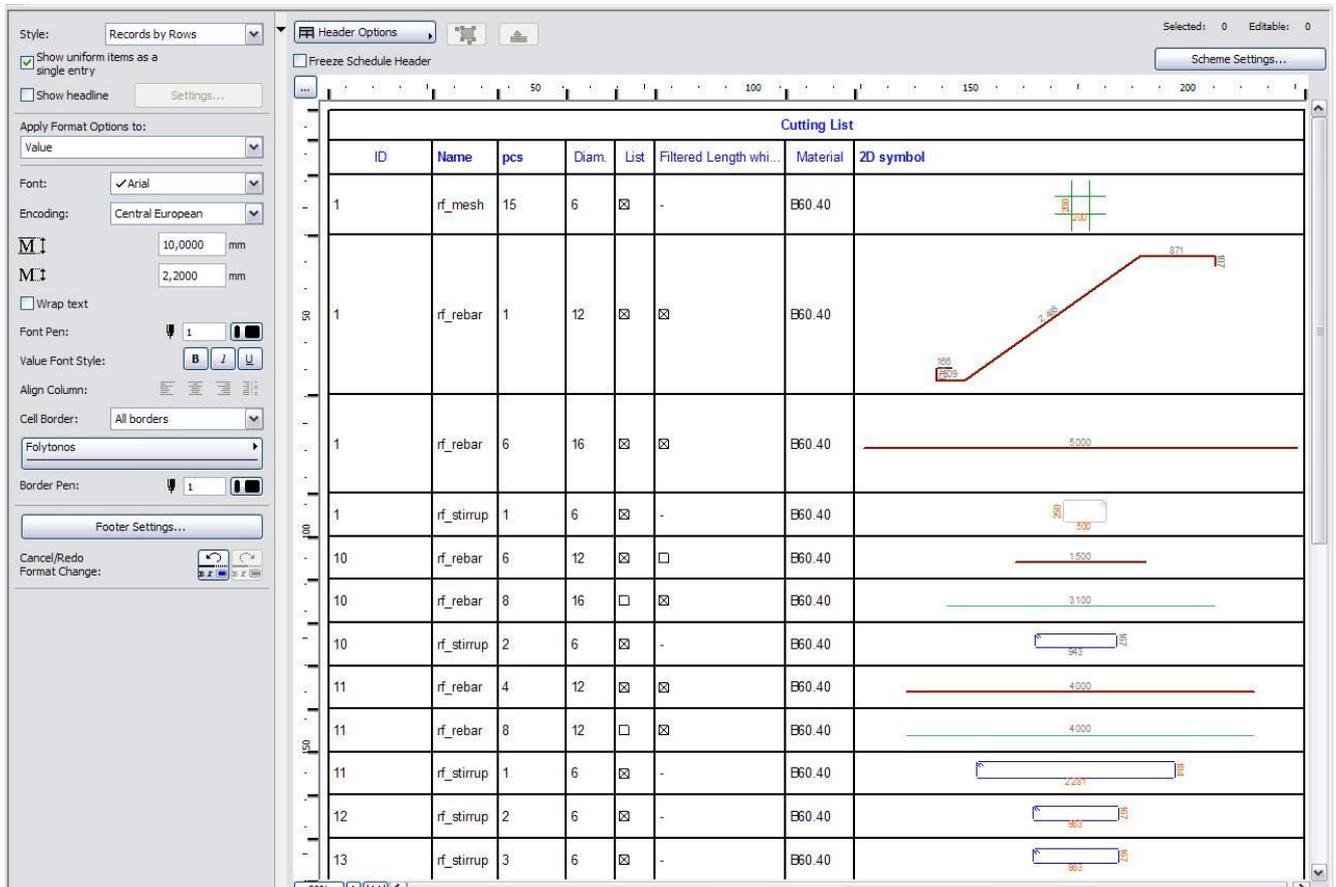
Choose the „**Import...**” button on the right side. Find the installed Reinforcement Library on your computer and the „**[eptar]-Reinforcement.xml**” file in the same location. In the most case you will find this file here: „**c:/Program Files/Graphisoft/ArchiCAD/Add-Ons/[eptar]_Reinforcement**”.

After the loading method choose the „**[eptar]-Reinforcement**” type from the schemes. Click on the „**OK**” button.

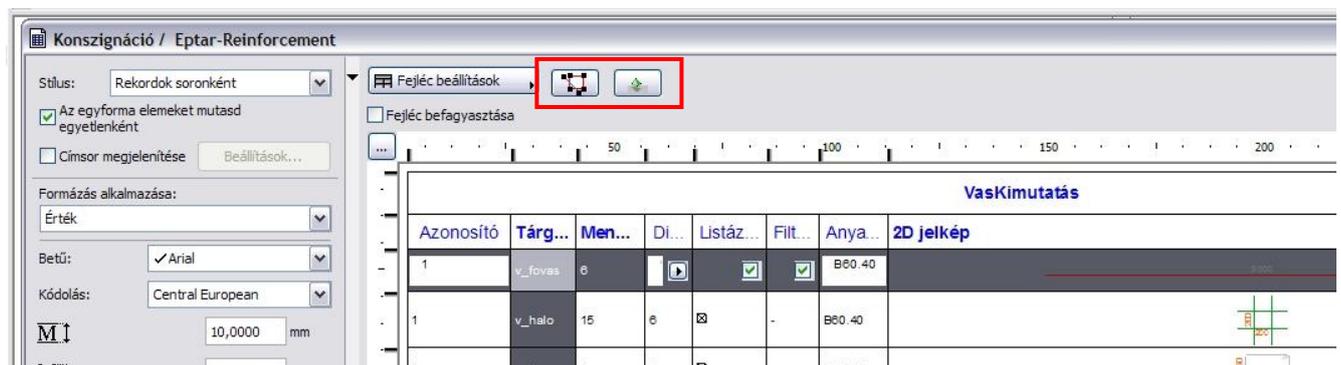


If your interactive schedule don't opens automatically, select the „**Schedules/ Element / [eptar]-Reinforcement**” in the „**Navigator**” on the right.

In the schedule window you can see all the placed reinforcement objects collected by type, diameter, shape, length and iron-quality. Unfortunately ArchiCAD do not support to ignore the hidden layers, so you will see all the rebars on the model, even they are visible on the model or not. If you want to filter the visible or not visible rebars differently, you have to define the layer filters in the settings dialog (see more in 3.3. point). For the reason of above, we suggest using this schedule to check the rebars on the model, but do not use it to create Cutting lists or summary with it, because it can cause mistakes in the calculation.



If you can not recognize an object based on the ID or the parameters in the list, you can use one of the buttons on the top of the dialog. Choose one rebar in the list and click the activated button on the header the list. ArchiCAD will show the object on the **floor plan** or in the **3D window** immediately.



In this schedule you have a possibility to change some parameters of the selected object(s). Select one object in the list and change the diameter or the Iron quality values. These changes will automatically activate for all placed objects in the model, so you can easily change the IDs or other parameters of the objects.

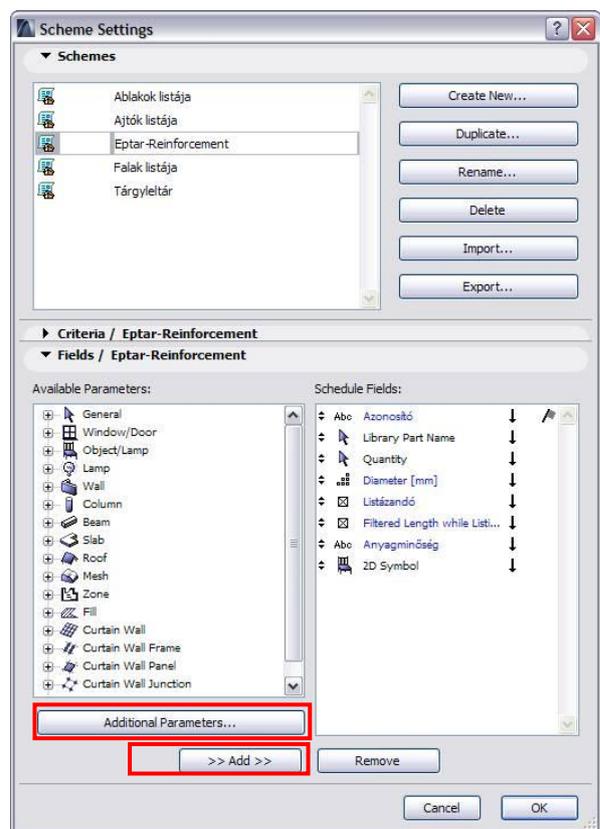
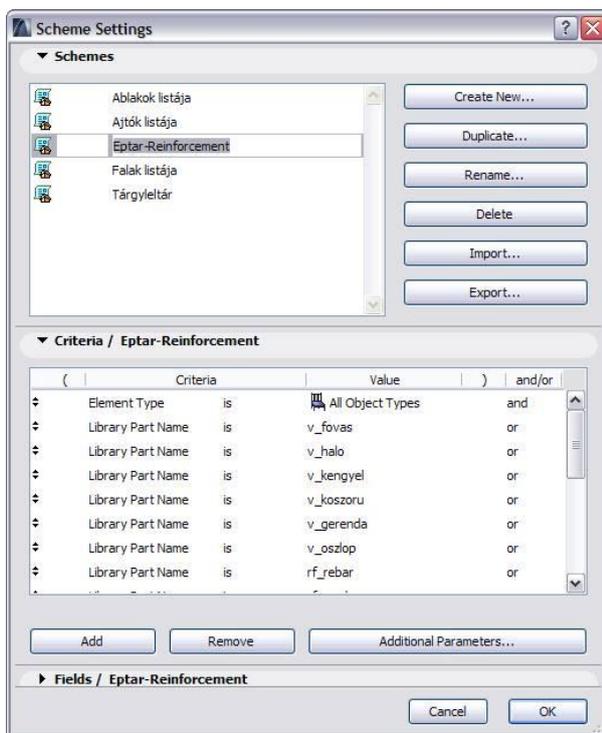
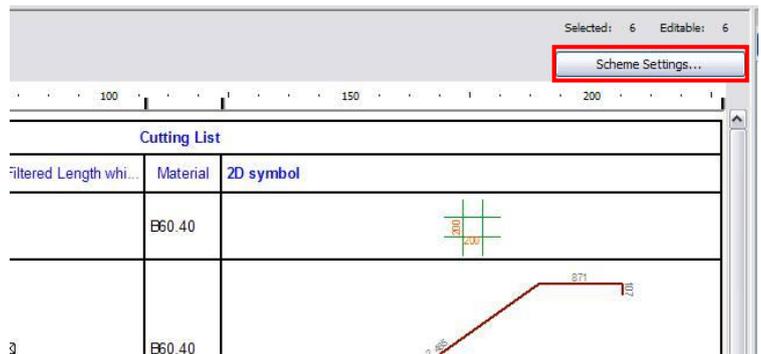
3.3. Create custom Cutting list

With the help of this schedule, you can define your own style cutting list as well.

Click on the „**Scheme Settings**” button, the setting dialog opens, where you can add or remove filters and parameters.

On the bottom parts of the dialog, under „**Fields / Éptár Reinforcement**” tab page you can see parameters on the right side which are visible in the schedule.

On the left side you can see parameters, which you can add to your list, which can use for filtering the objects in the schedule.



Under the „**Object/Lamp**” category you can find more default parameters which you can add to the list to filter and represent the objects, but you can add custom Reinforcement object parameters to the list as well. To add custom reinforcement parameters to the list use the „**Additional Parameters...**” button on the bottom. When you click on this button, ArchiCAD collect all the loaded library parts and show them in a list. Select the „**[Éptár] Reinforcement**” library and one of the library parts inside.

From the visible parameter list choose the necessary parameters and push the „**>> Add >>**” button to attach to the list on the right side.

When you click on the „**OK**” button, your selected parameters will be available immediately in the „**Schedule fields**”.

In Interactive schedule there is a priority of the parameters as well. If you move parameters on the top of the list, the list will be filtered based on that parameter first.

For more information about the interactive schedule use the Graphisoft documentation.

Chapter 4.

4.1. Technical support

If you have any problem, question, comment or request regarding the application, please contact the éptár ltd. on the next channels:

ÉPTÁR Kft.

Kata Danis

Telefon:

e-mail:

+36 1 / 225 73 55

info@[eptar].hu