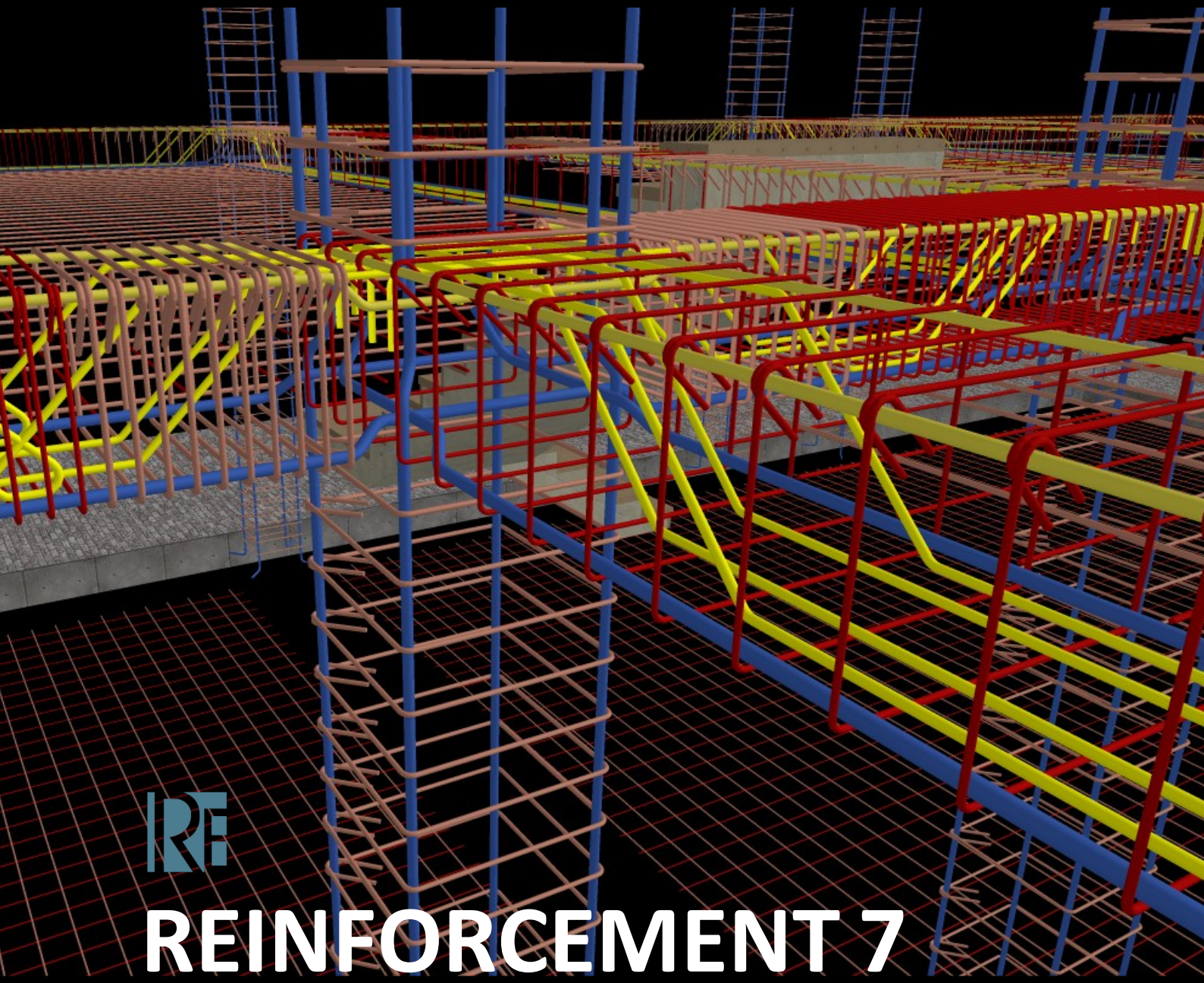


REINFORCEMENT 7

USER MANUAL

June 2025

eptar

Program and library upgrade:

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

[Eptar] Reinforcement – User Guide

The [eptar] Reinforcement 7 – User guide is under copyright and all rights are reserved.
Copyright © 2025 Éptár Kft.
To copy, translate or extract any parts of the document is only allowed, if written permission is given from the owner.
Version 1

Trademarks:

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

Contents

Chapter 1 ..... **4**
First steps 4
1.1 Installation of the solution *4*
1.2 Buying Reinforcement and online activation *6*
1.3 Activation of the solution *7*
- *Online Key* *7*
- *Borrowing the license* *7*

1.4 Loading the solution *8*

Chapter 2 ..... **8**
Using the [eptar] Reinforcement 7.0 application 8
2.1 Basic usage of the software *8*
2.2 New layers options for the reinforcement plan *9*
2.3 Placing the elements and their settings *13*
2.4 Object placement with Eptar add-on. *41*
2.5 Creating Complex structures *45*
2.6 Component tool *56*
2.7 Edit Objects *61*
2.8 The new thermal bridge breaker *69*
2.9 Deleting Elements *70*
2.10 Using ArchiCAD Label element *70*
2.11 Creating Section drawings *71*

Chapter 3 ..... **78**
3.1. Cutting list placement and ID check *78*
3.2. Using Interactive Schedule *85*
3.3. Creating custom Cutting list *87*

Chapter 4 ..... **88**
4.1. Technical support *88*

New in Reinforcement 7:

- Redesigned tool palette

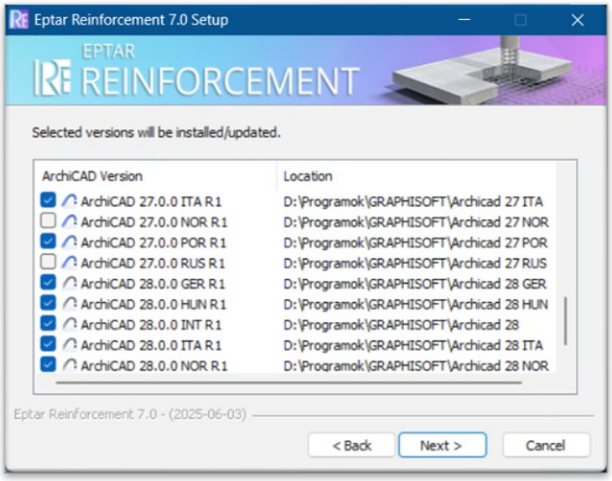
- New tool:
  * component tool
  * thermal bridge breaker element
  * multi-direction custom rebar
  * new parameters palette

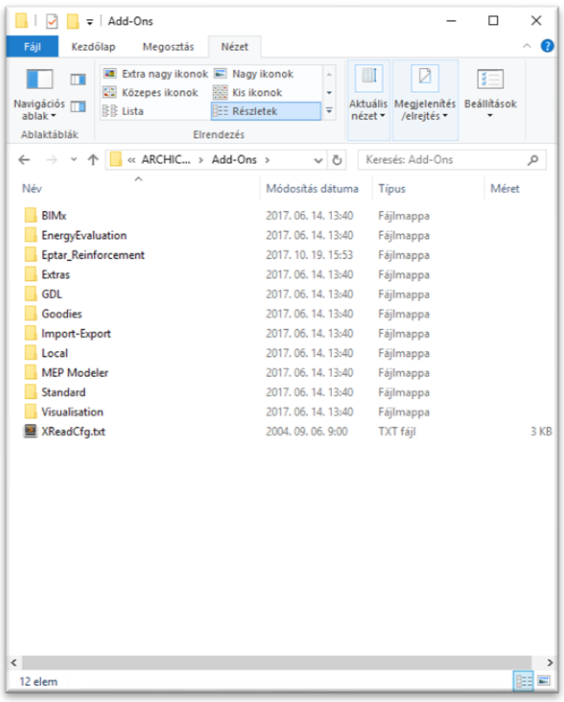
Chapter 1

First steps

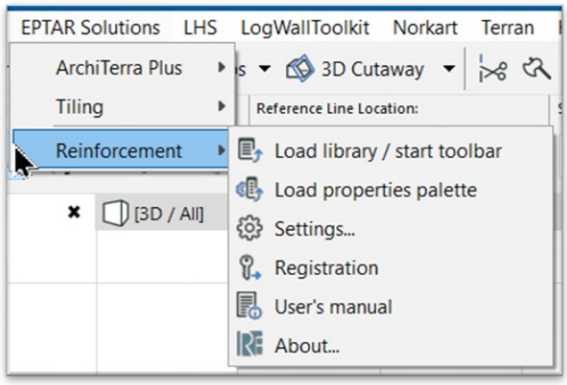
1.1 Installation of the solution

The [eptar] ---------- can be used with ArchiCAD 23 or higher versions including Archicad 28.
To install the program, launch the installation file and follow the instructions given.

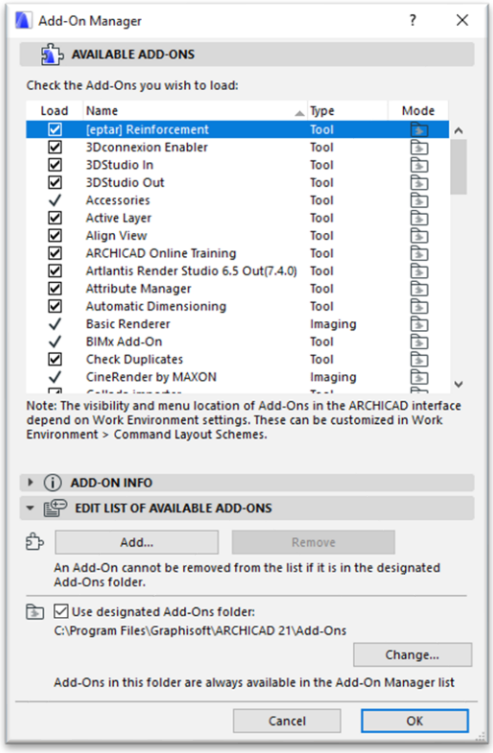


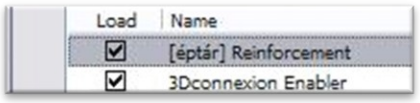


After the installation, at the first launch of ArchiCAD, „**EPTAR Solutions / Reinforcement**” option will be available in the menu bar.



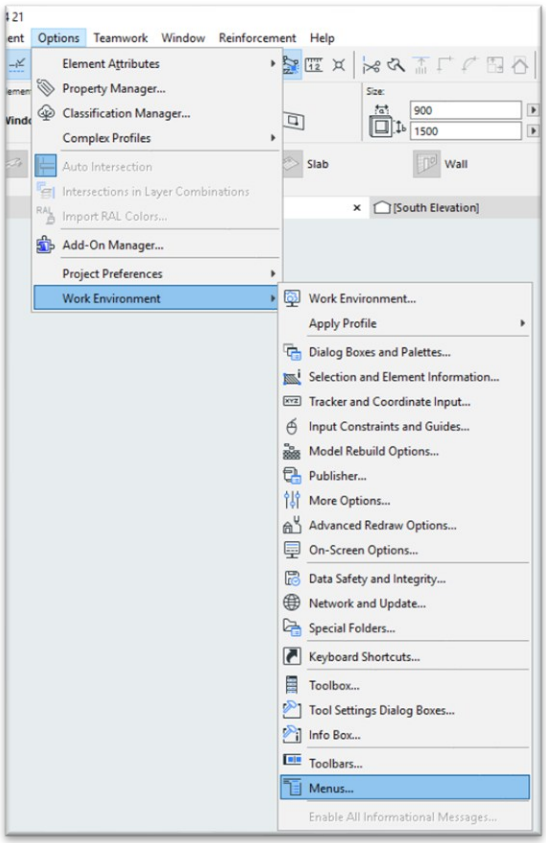
In case the Reinforcement menu does not appear, please check „**Options/ Add-On Manager…**” menu, if the „**Reinforcement**” solution is active.





If the check box is active, please read why ArchiCAD can not detect the solution, and please contact Éptár Kft. with the problem.

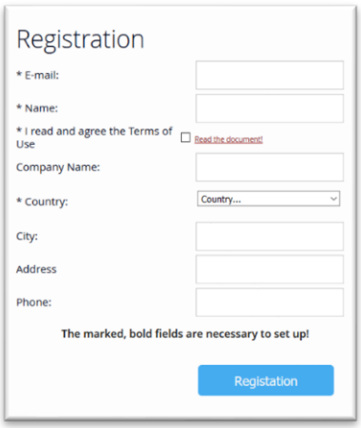
If the check box is selected but the program still can not see the „ **EPTAR solutions / Reinforcement**” menu, please check „**Options / Work Environment / Work Environment**” dialogue if Reinforcement menu is not blocked.
Make the menu visible and click„OK”.
As long as all possibilities have been checked and the program still does not display the „ **EPTAR solutions / Reinforcement**” menu, please contact the developer, Éptár Kft.



1.2 Buying Reinforcement and online activation

To start Reinforcement 7.0 the license must be purchased and activated otherwise the application will only work in demo mode. Users who own Reinforcement 1.5/2.0/2.1/3.1/3.5/4.0 and 5.0 can upgrade their solution to version 6.0.
There are two ways to buy Reinforcement: on our website after logging in: https://www.eptar.hu/cadsupport.php or through our distributor partners.

The list of Éptár Reinforcement Partners is available on this link: *https://www.eptar.hu/bim_shop_hu?lang=en*



Registration on eptar.hu

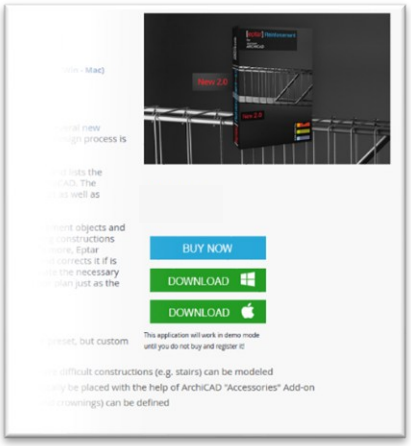
In case you want to buy Reinforcement 7.0 through our website or you already own Reinforcement, please visit our website:

https://www.eptar.hu/cadsupport.php.

If you have a registration, please log in.
If you don't have a registration, please register by pressing the **login** button and filling out the form. Then follow the instructions given in the received email and login to your account.

Warning: if you bought Reinforcement through a distributor partner, do not register, instead contact our partner.

Purchasing on eptar.hu
After logging in go to the "Store" page and choose Reinforcement 7.0.



In case you own Reinforcement 1.5/2.0/2.1/3.1/3.5/4.0/5.0 or 6.1, "Reinforcement 7.0 update" will be displayed among the products.
To purchase, click on the "Buy now" button. You can pay via PayPal, even if you don't own a PayPal account, you can still give the requested data and pay.

Activating a purchased product on our site
After purchasing Reinforcement 7.0 the product appears in your user account and on the Download page as well.

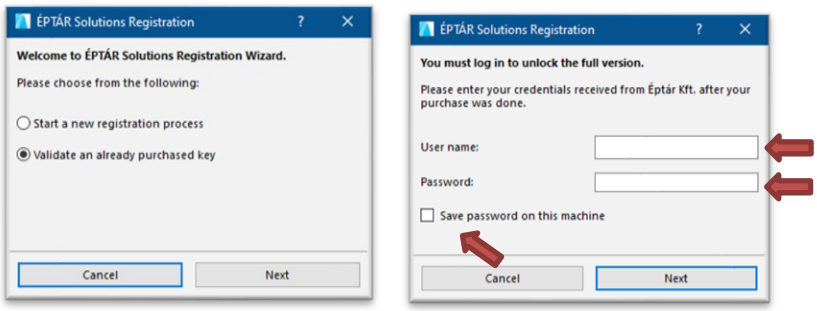
1.3 Activation of the solution

To activate the Reinforcement 7.0 plugin your computer should have an active internet connection.

IMPORTANT:
The machine key is already NOT supported. During the next upgrade the machine key will be terminated and switched to online key option.

- **Online Key**

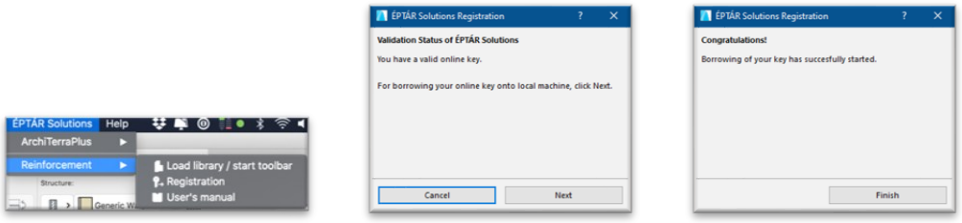
  After starting ARCHICAD, in the *EPTAR Solution* menu choose Registration in the *Reinforcement 7.0* button. Then follow the steps for Online Key Validation like in the pictures below.



- **Borrowing the license**

  We present you a new option for online key. This option is not free, it is possible to buy for each licens you have. Buying it for one license will only work with that license. How it works:

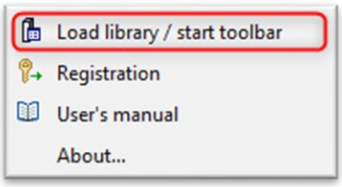
- after you registered a Reinforcement 7.0 online key open the Registration again
- click on the “Next” button and if you receive a message “Congratulation!”, your license is switched succesfully to “offline” mode for 7 calendar days, that means you can use your license without an internet connection for a week. After the time runs out, the license switches back to online mode.



If you buy this option and it does not work, please contact to the ÉPTÁR support:
info@eptar.hu

1.4 Loading the solution

The Reinforcement 7.0 solution is made up of the expansion and the attached library. To activate the library, choose „**EPTAR Solutions/Reinforcement / Load Library/start toolbar**”.

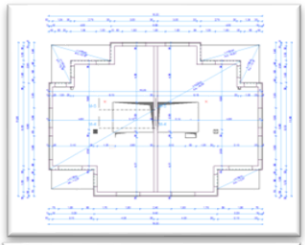


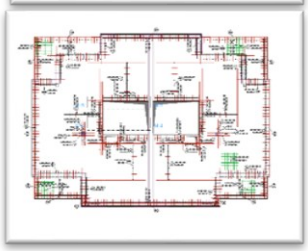
After loading the library, you can place elements like objects, but some elements (rebars, meshes) can be attached to slabs, walls or polygons by choosing the elements of „Reinforcement Floating Palette”. By choosing the command, elements of Reinforcement 7.0 library will appear, and you can connect them to slabs or walls.

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

Chapter 2

Using the [eptar] Reinforcement 7.0 application









2.1 Basic usage of the software

In reinforcement plans some levels appear differently than on the architectural plans. To get the requested design for the reinforcement plan you need to change the sheet options. Reinforcement plans can be created at the same time as architectural plans, so it is unnecessary to change the settings of the architectural one, which means for the reinforcement plan, you have to create new view options (see point 2.2).

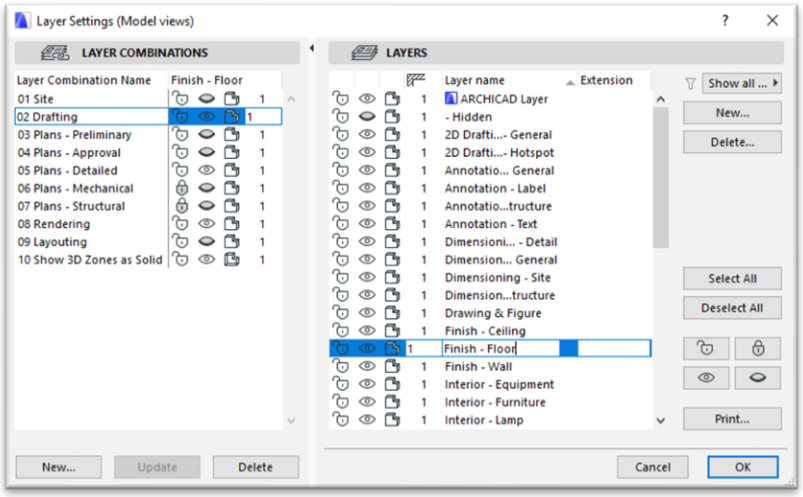
After changing the view some types of bars and meshes can be placed as objects on the layout and in the 3D model (see point 2.3). In case of some objects (rebar, mesh) Eptar add-on can help in the placement (see point 2.4). Complex elements (beams, columns) can be created from individual elements and 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 (see point 2.5) .

While placing the elements you can give them separate identifiers but this is not necessary since at the end of the planning process, during the detection of bars, the program checks the number of bars and renumbers the faulty or different serial numbered ones and optimizes the numbering (see point 2.6). You can optimize them with the help of the interactive lists during the planning (see 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 bars’ view drawing (see point 2.7).

2.2 New layers options for the reinforcement plan

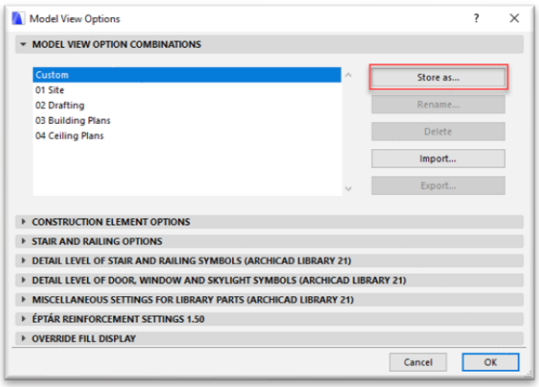
While creating architectural plans on some layouts, if the user cuts the building at a 1 meter height, and looks down from this point, the elements of the level are being displayed. While creating the reinforcement plan, this kind of display shows many useless elements. Although, in the reinforcement 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 as well. 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 the following layers:
„Reinforcement – Lower reinforcement"
„Reinforcement – Upper reinforcement"
„Reinforcement – Beams, wreaths"
„Reinforcement – Columns"
„Reinforcement – complex element container"

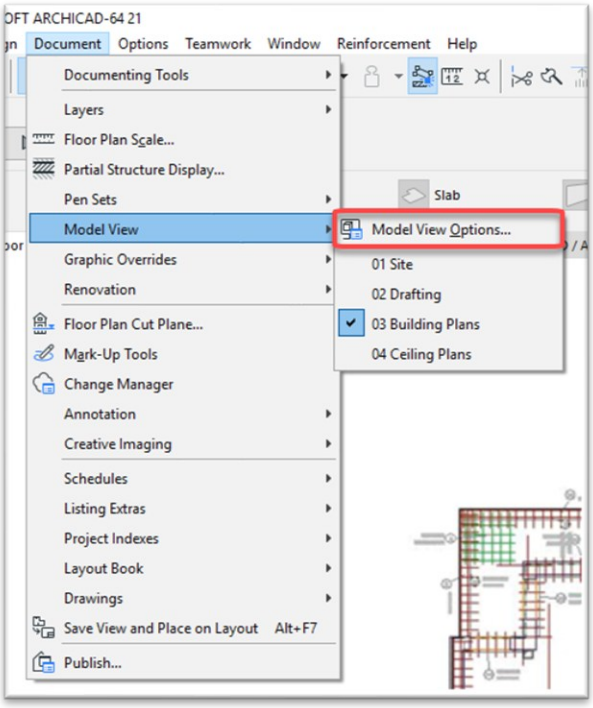
On the left side, create "**Layer Combination Settings**", with these layer visibilities:
„Reinforcement – 3D group"
„Reinforcement – Complex element definition"
„Reinforcement – Lower reinforcement"
„Reinforcement – Upper reinforcement"
„Reinforcement – Segment"

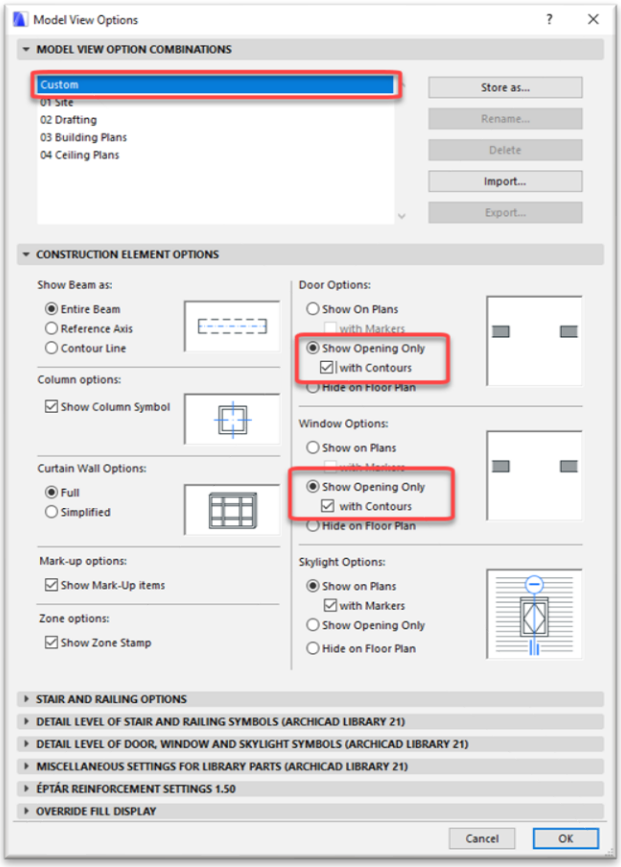
These layer views will help you to be able to change between views fast and effectively and to be able to use the right layer-options at the creation of some sheets.



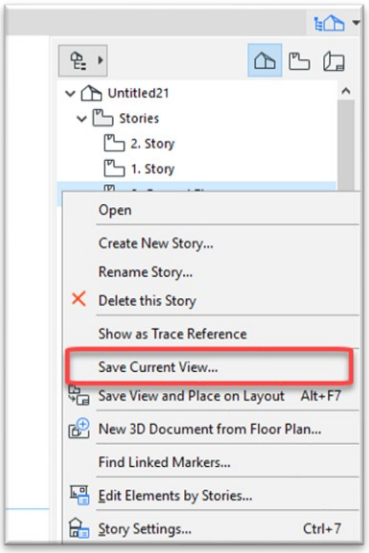
Open „**Document / Set Model View / Model View options**" window. In the appearing dialogue choose „**04 Civil Engineer View**" type. ArchiCAD at this kind of display format only shows the contour of the walls and openings, however, we can set an individual display format too.

Under the „**Override Fill Display**” tab switch „**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 name it „**05 Reinforcement plan**”. Click OK and close the window.

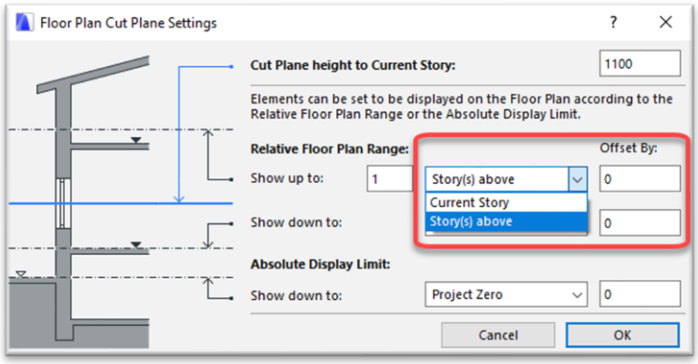




On the right side on the „**Navigator**” palette under the „**Project Map**” tab, right-click the name of the active layout. Choose „**Save Current View...**” command.
Give the view an easy, typical name (e.g. „**Downstairs – Reinforcement plan**”).

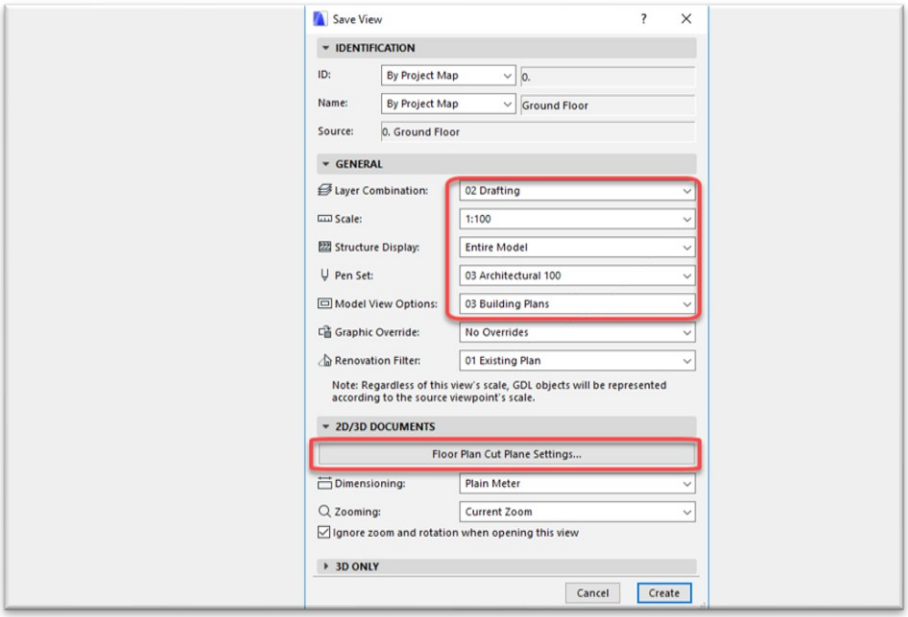


Switch „**Layer Combinations**” to „**Reinforcement – Down rebar**” type, put „**scale**” on 1:50 and switch „**Model View**” to the (recently created) „**05 Reinforcement plan**” type.

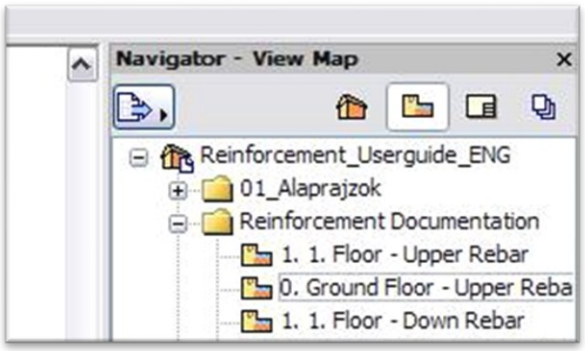


The setting of „**Floor Plan Cut Plane Settings**” is necessary because that way we can see the upper level’s walls and apertures on layout view.

Click the „ **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 switch „**Show Down to**” to „**Current Story**” 0.5.



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



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

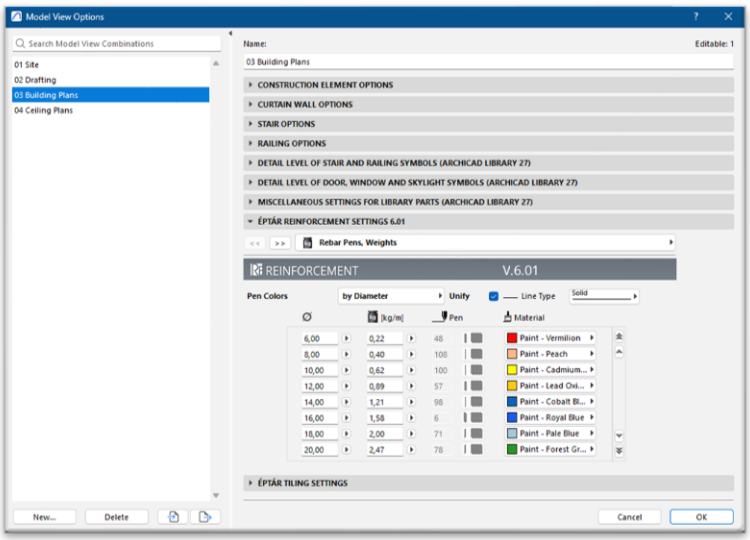
Comment
With these steps we have set the necessary options for the reinforcement plan. We advise you that in bigger offices the sample sheets (*.tpl) should be created in a way that these settings are already set so afterwards they will not have to be set again and again.

2.3 Placing the elements and their settings

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

BASIC NOTES OF THE ELEMENTS

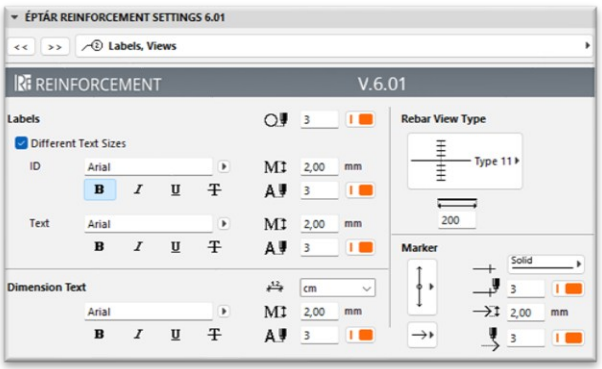
Appearance of the elements on the layout and in the 3D window can be changed globally in a separate dialogue. Open the „**Document/ Set Model View / Model View Options...**" window. Choose the „**Reinforcement Settings**" tab (lowermost).



Here, you can adjust the color of the pen for every type of element, bars or every element not in the list, and this way you can also make the plan easier to read.
Also in this dialogue, you can determine the bar's diameter or the mass associated if it is necessary. **Whenever you change a value in the dialogue, you must refresh the plan to activate the changes. In the meantime, the values of the bar list (weight values) change only, if the optimization of the bars 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 the „**2D representation**" or in the „**3D representation**" page, switch the „**Object Custom Pen**" switch ON, and set the desired color.

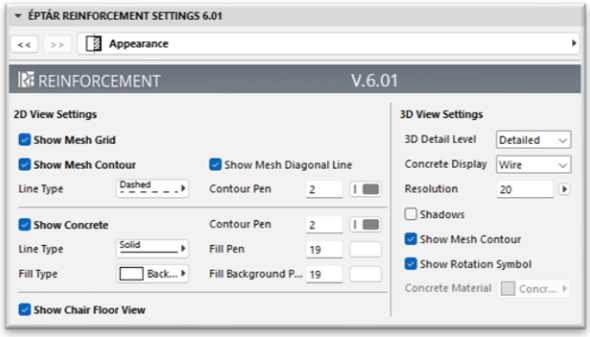
If you want the program to show the bar weight with a definite number of decimals, you have to set the display of decimals to the preferred number in the "**Angle & Font Size Decimals in Dialog Boxes**" box which can be found under "**Options /Project Preferences/ Working Units...**" .

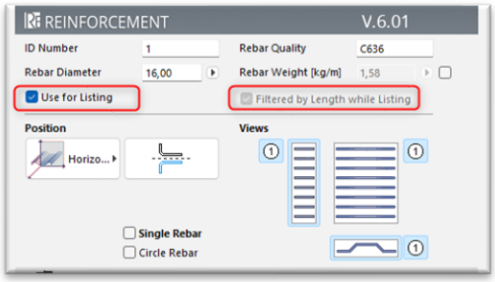


In Reinforcement 7.0 you will find further global setting options. You can define the default settings for the appearance of the views, labels and dimensions of the rebar element.
You can set the text styles of the labels, the unit of dimensions on the „**Labels, Views**" page and choose the Rebar View Type.
Settings of stirrups and meshes can be found on the" **Appearance**" page and you can even set the 3D detailing and other settings for the whole plan.

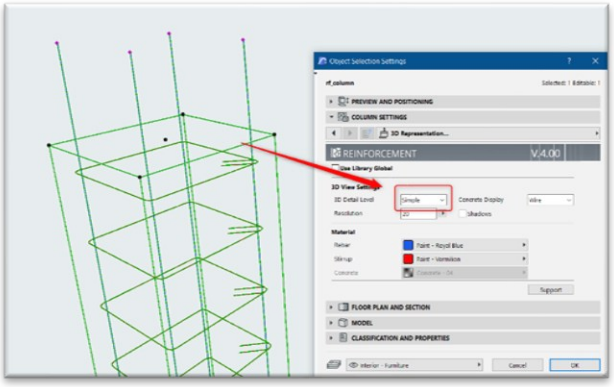
Global settings will define the default settings of each reinforcement element that you use in the plan. In every single element you will find the checkbox „**Use Library Global**”. Switch this checkbox off to be able to modify the default settings of single elements.

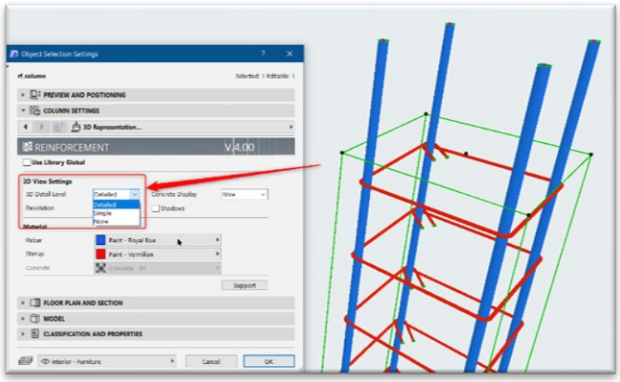
We suggest that you create sample sheets (*.tpl) so that these settings are already set. In this case those will not have to be set again in the future.



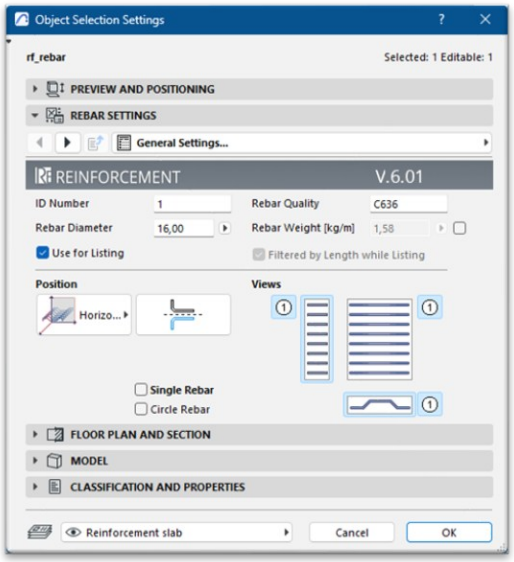


There is a „**for Listing**” switch in every element. Sometimes, some elements are only placed for a complex element’s creation in the model space, or maybe an explanation, and bars as outstretched views. In these cases, these bars can be left out from the optimization and listing. This switch is basically switched on so every element will be listed. When switching off, be careful so that you don’t miss out elements which you want to show in the list from the listing.
The “**Filtered by Length while listing**” option will separately list all rebars by length in the Cutting List.





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 this way the progress/visualization will be much faster and editing will be easier.



Every element can be given a separate serial number. If you would like to customize the name of the rebars, it is possible. In this case of creation or refreshing of detection of bars, the solution only checks if bars with the same serial numbers have the same properties. If you’ve made a mistake and you’ve given two different bars the same number, the program will renumber one of them.
However, if you do not want to spend time with this 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 the Options/ Project Preferences/ Dimensions menu. If you set the Dimensions to Plain Meter, Reinforcement add-on will be working just as usual. If you choose US Architect, you will be able to give measures and to display all measures and list content in imperial units.

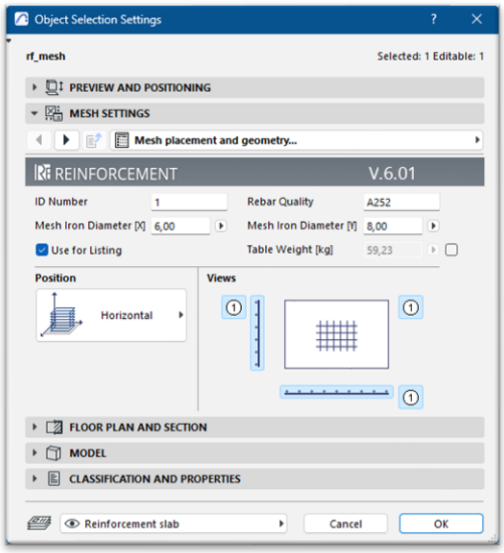
If you change the unit once you have already started working, the already placed objects will be converted.
If you find that any objects' measures were not converted (this can mainly happen to the side views of the rebar on the floorplan, and the Main View in the Cutting list.) highlight the object (or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



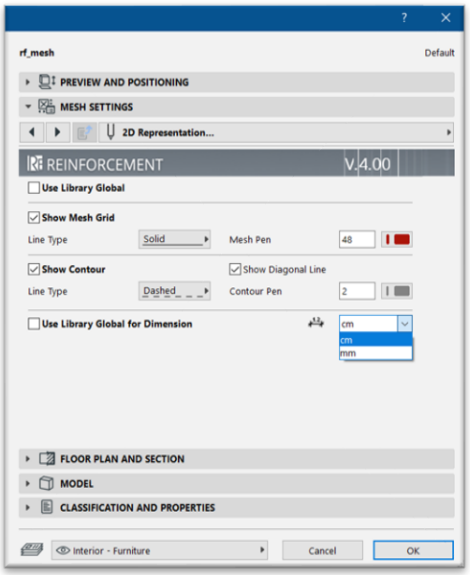
While choosing the mesh object, in the settings window you can set the element's maximum height in the plan, diameter- holes values (XX x YY), the size of overlapping and the material quality of meshes too.



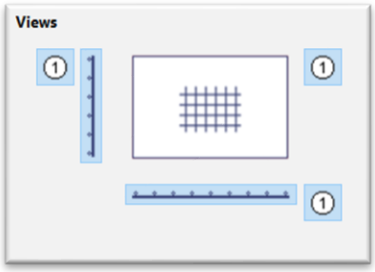
The solution automatically calculates the weight of the mesh element but if you set up a special diameter (e.g. 9 mm) then the mesh's weight dialogue becomes active and you can give the weight value to the selected diameter and the holes value.

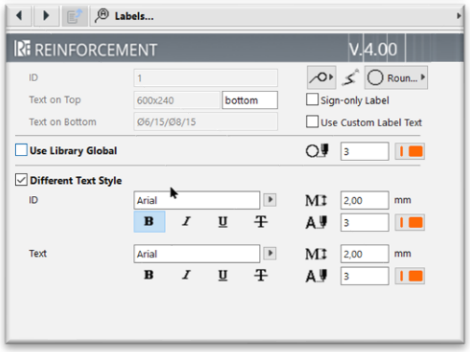
After setting the main parameters, on the second settings page (**Layout view**) you can adjust how the mesh will exist in the plan. You can make only the mesh's contour visible, make only a piece in the mesh's middle visible or make the mesh's data visible in the plan too. Since these settings can be 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.



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

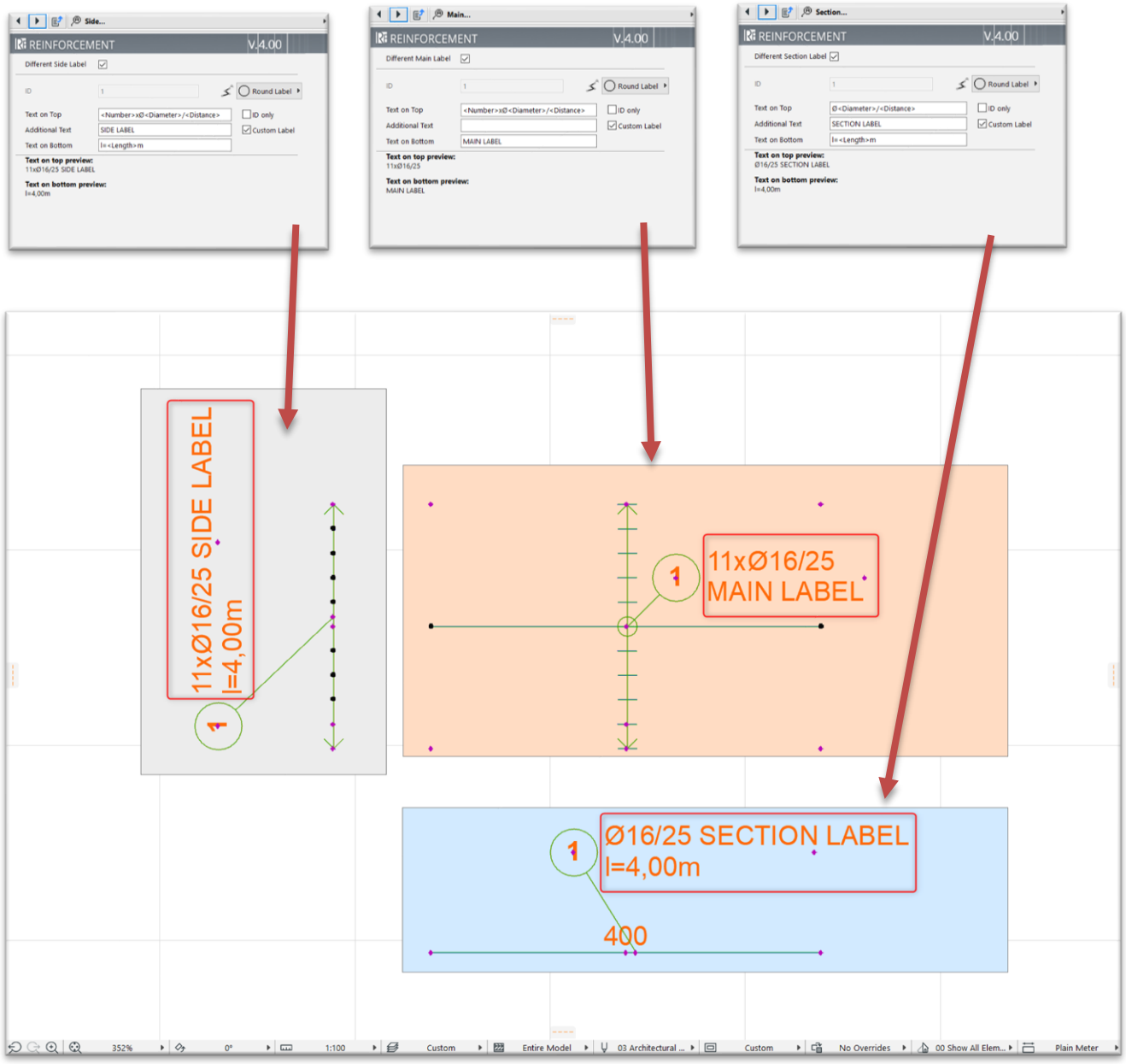




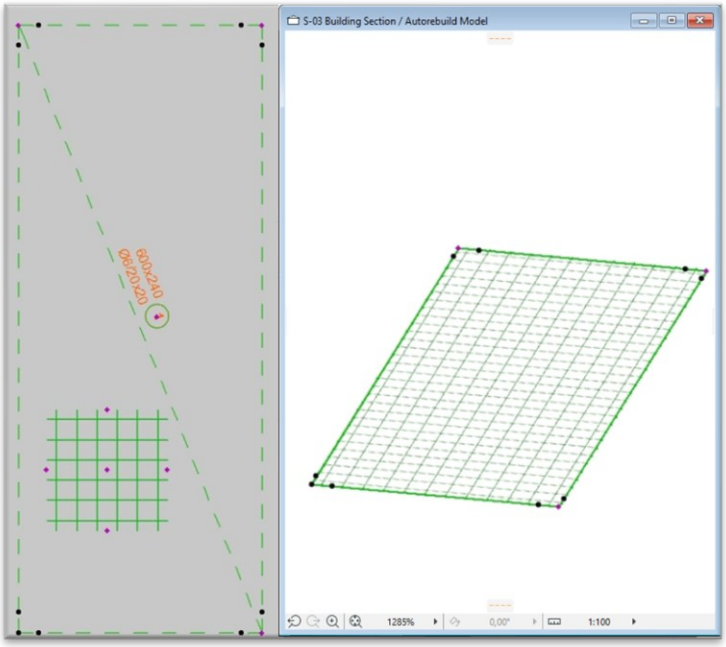
You can set the display of labels on the "**Labels**" page. If you have turned the "**Show Label**" switch on, the mesh will be labeled according to the label type you chose. Content of the label could be overwritten on this page: you can decide whether you want to see the ID, measures, diameter of the 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.

In this version there is a new way to show on the floorplan different label content on 3 different views. For this option you can turn on the checkbox on the next 3 tabs and define the text contents of each label.



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

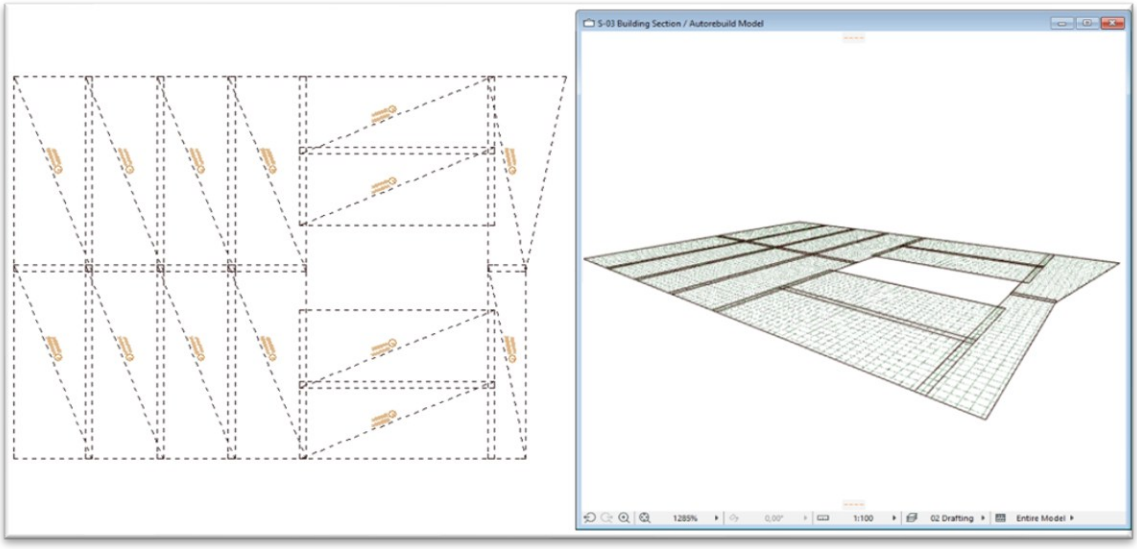


By clicking the OK button, you can place a factorial valued piece of web on the layout. This piece of web can be adjusted (make it bigger or remake it) by the holder points on its heels/corners (pink). No holes can be cut out on the manually placed mesh, but it is possible on the automatically placed mesh. You can read more about this in the 2.4 point.
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 "**Obejct 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 its 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.

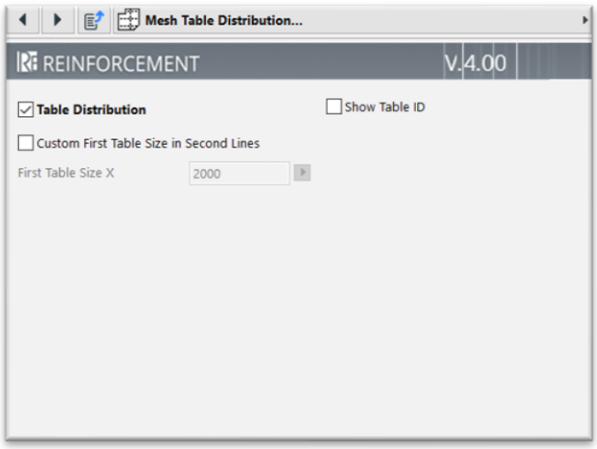
You can multiply the object with the ArchiCAD „**multiply**" command or create multiple copies of the object. There are hotspots on the corners of the mesh, these hotspots define the overlapping size of the objects. These parameters can be edited in the Object Settings dialog box with the parameters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

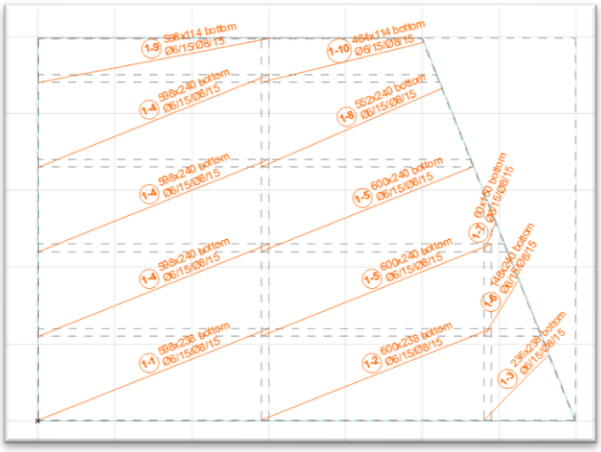


Mesh laying managment:

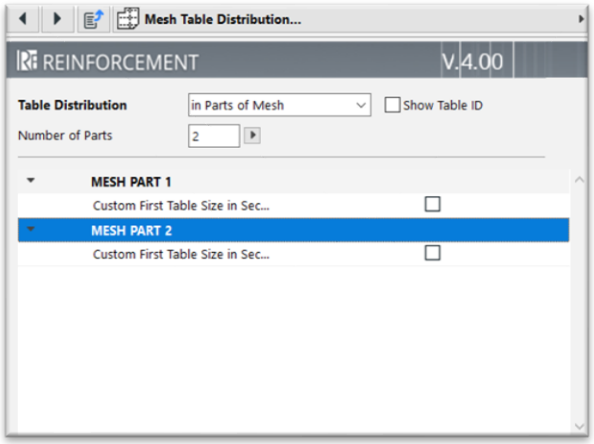
A new feature is the possibility to plan the mesh laying. This option must be enabled in the Setup window.
The table layout can be adjusted individually. It is possible to use one installation direction, but several installation directions can be planned individually.

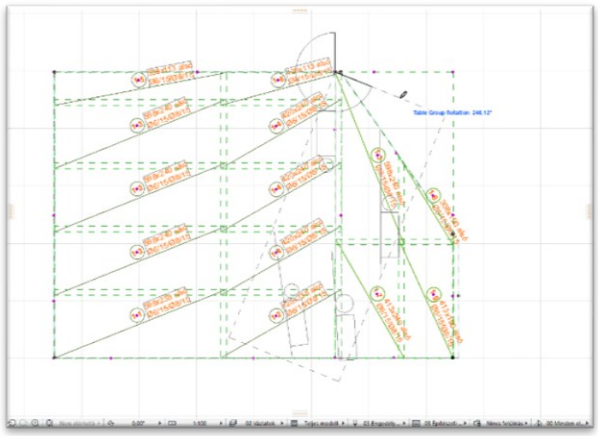
If one laying direction is used, the mesh elements are placed in one direction and matched to the shape of the structure, treated as a continuous surface. In this case, the pieces of the “protruding” mesh are automatically cut along the contour of the structure. It assigns a unique identifier to each mesh element (table). Items of the same size will be marked the same. When creating a Cutting List, it lists the mesh elements of the same sign together in a row.



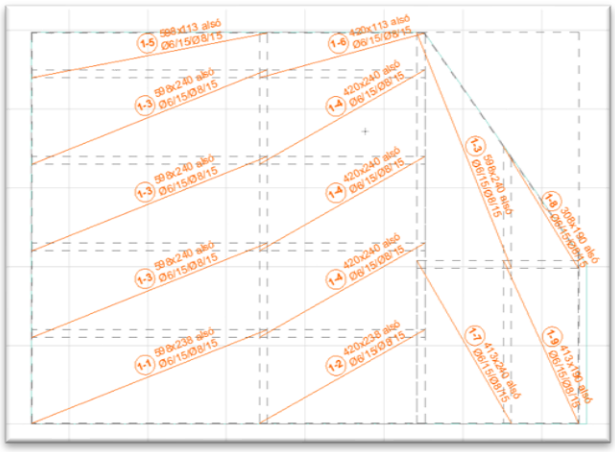


If, due to the shape of the structure, it is simpler or more economical not to distribute the mesh boards at the same time, then its surface can be planned to be laid in several directions. This is a more flexible board layout design because the boards can be rotated according to the shape of the structure, resulting in less cutting loss for cut elements.





ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1 - 1		6 - 8 mm	7	6,000 m x 2,400 m	6,000 m x 2,400 m x 7	A252	414,6 kg
1 - 2		6 - 8 mm	1	2,339 m x 2,400 m	2,339 m x 2,400 m x 1	A252	23,1 kg
1 - 3		6 - 8 mm	1	1,457 m x 2,400 m	1,457 m x 2,400 m x 1	A252	14,4 kg
1 - 4		6 - 8 mm	1	0,575 m x 1,433 m	0,575 m x 1,433 m x 1	A252	3,4 kg
1 - 5		6 - 8 mm	1	5,492 m x 2,400 m	5,492 m x 2,400 m x 1	A252	54,2 kg
1 - 6		6 - 8 mm	1	6,000 m x 1,123 m	6,000 m x 1,123 m x 1	A252	27,7 kg
1 - 7		6 - 8 mm	1	4,610 m x 1,123 m	4,610 m x 1,123 m x 1	A252	21,3 kg
				Total Weight:			558,70 kg

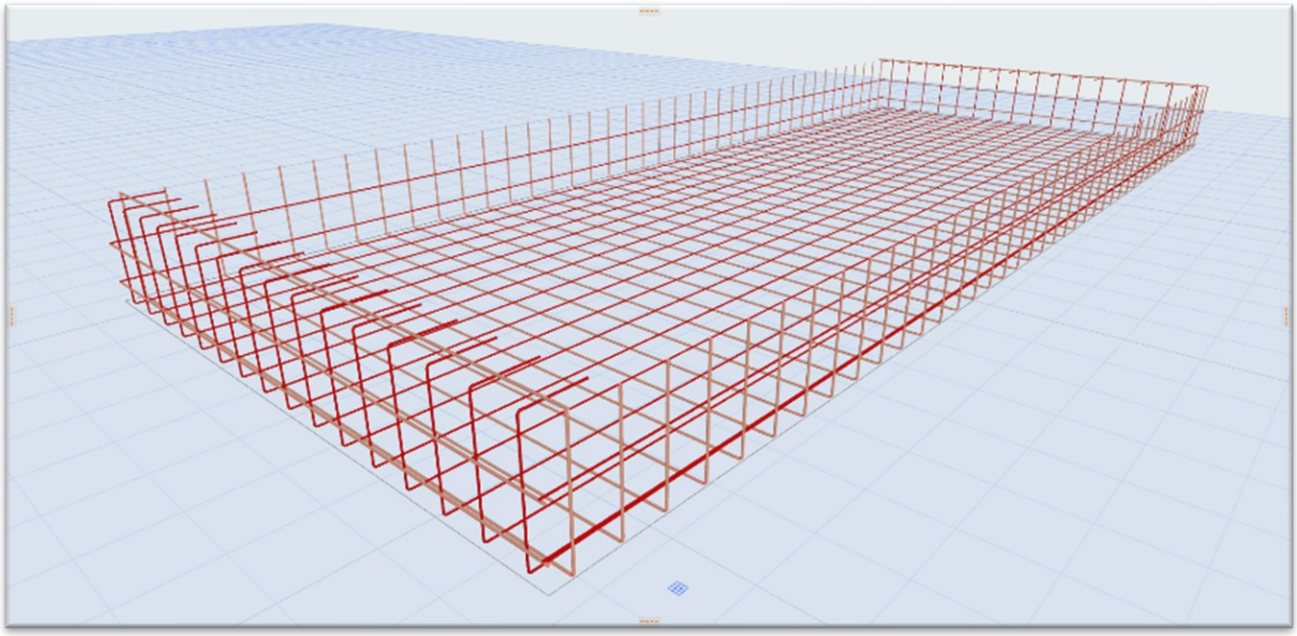


Of course, the Cutting List also collects items grouped by size accordingly.

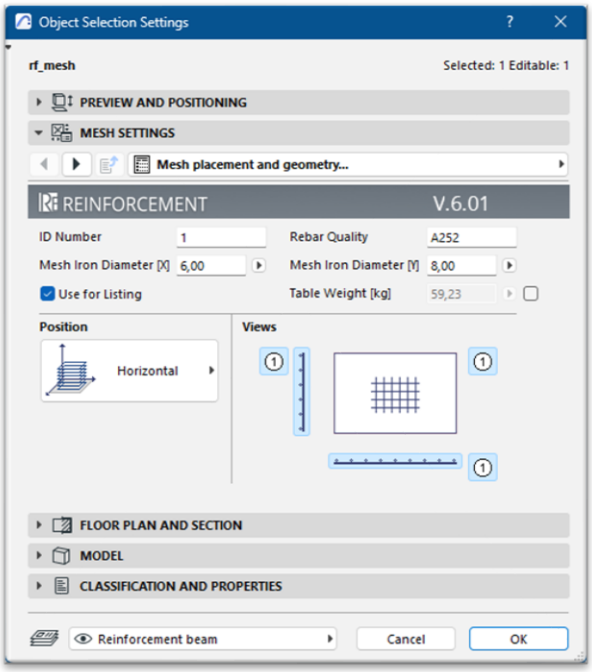
01 List

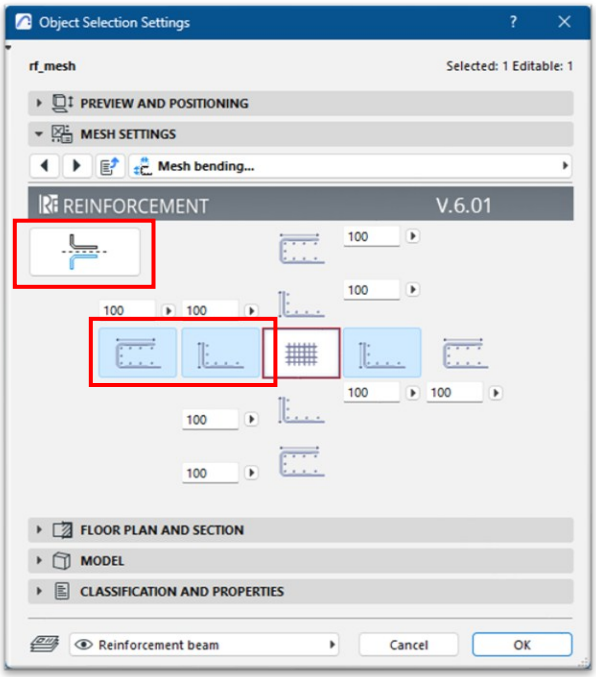
ID	Main View	Diameter (mm)	Pcs.	Length (m)	Total Length (m)	Quality	Custom 1	Custom 2	Sum Weight (kg)
1 - 1		8	164	0,640	104,960	B503	-	-	23,3
1 - 2		16	18	7,779	63,719	C636	-	-	100,5
1 - 3		12	8	1,000	8,000	C636	-	-	7,1
2 - 1		6 - 8	1	6,000 x 2,400	6,000 x 2,400 x 1	A252	-	-	59,2
2 - 2		6 - 8	1	1,282 x 2,400	1,282 x 2,400 x 1	A252	-	-	12,7
2 - 3		6 - 8	1	5,920 x 2,400	5,920 x 2,400 x 1	A252	-	-	58,4
2 - 4		6 - 8	1	4,757 x 0,229	4,757 x 0,229 x 1	A252	-	-	4,5

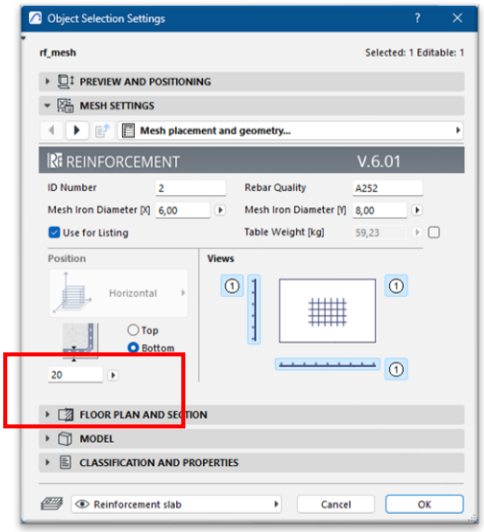
8 sum:	23,30
16 sum:	100,55
12 sum:	7,10
6 - 8 sum:	134,80
B503 sum:	23,30
C636 sum:	107,65
A252 sum:	134,80
Total Weight:	265,75



Upturned and backturned - This option is only available for manual placement!
On the mesh element it is possible to bend the edges of the net upwards or downwards. This function can be selected on the „Mesh bending" page after the Mesh element already placed on the floor plan. The upturned function can be switched on or off on the four sides of the net. If the mesh is placed in the bottom side of the slab, the bending is going automatically upwards, and if the mesh placed in the top the bending will turned downwards. You can also adjust the size of the upturn or backturn with the movable hotspots.

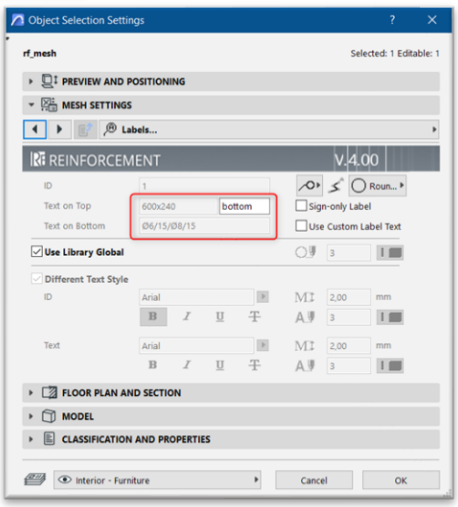


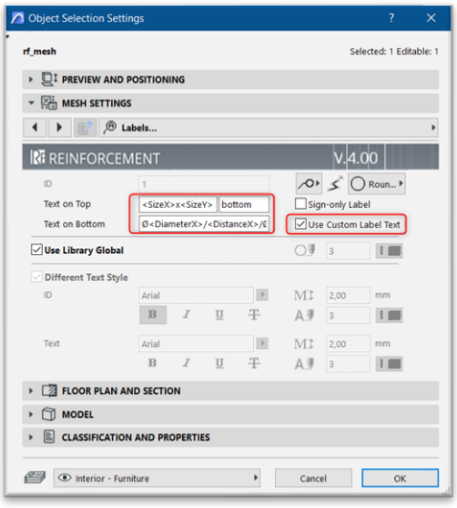




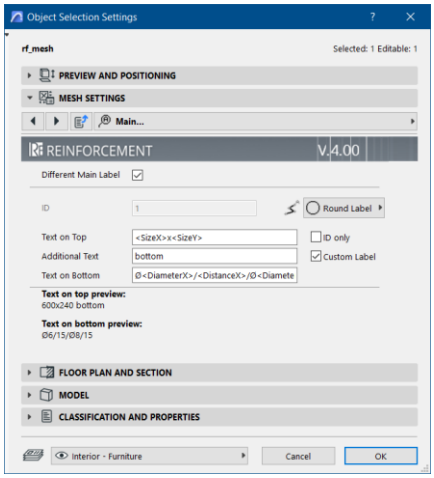
On the Mesh placement and geometry page, you can choose between bottom or top reinforcement. After you choose the bottom or top reinforcement, you can specify the value of concrete cover. This function works only for Horizontal Position. Otherwise, you can only specify concrete cover, which measures concrete cover from the selected side of the wall.

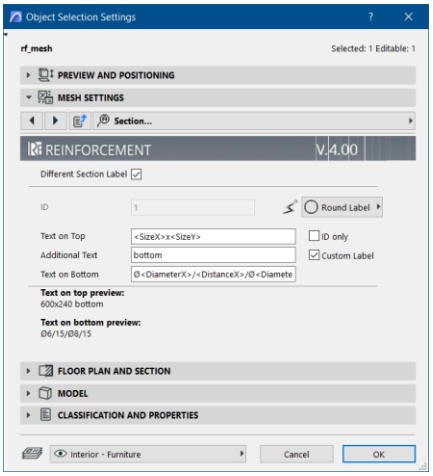
On the Labels page it is possible to give specific texts. The default setting in the Floor Plan view is to automatically show if the bottom or top reinforcement has been selected. If you would like to add custom text, turn on the „Use cutom label text" checkbox, and you can modify the original standard label text.

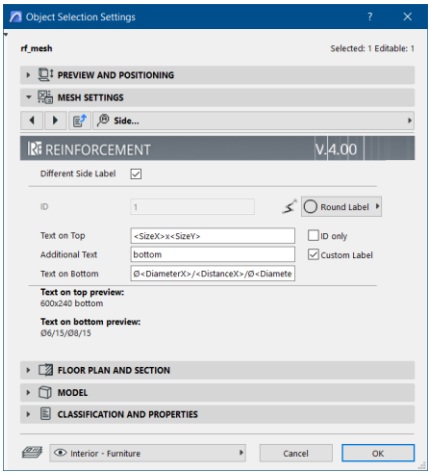




If you would like to use different text on a different views of the placed reinforcement element, go to next 3 pages. Here you you can turn on the "Different Main Label", Different Section Label" or the "Different Side Label" checkboxes, a apply different text contents in the labels.

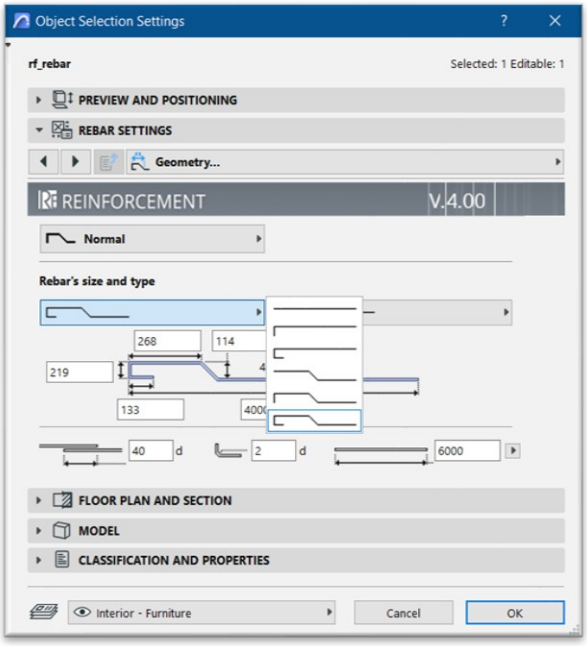






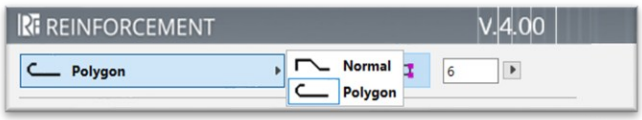
REBAR ELEMENT

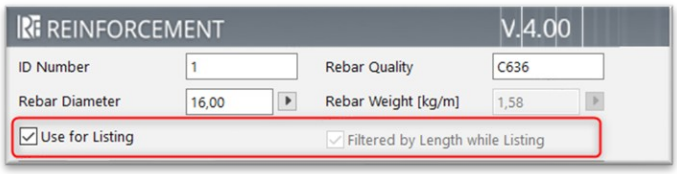
To place the rebar, choose from the dialogue, the „**rf_rebar**” object.



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.

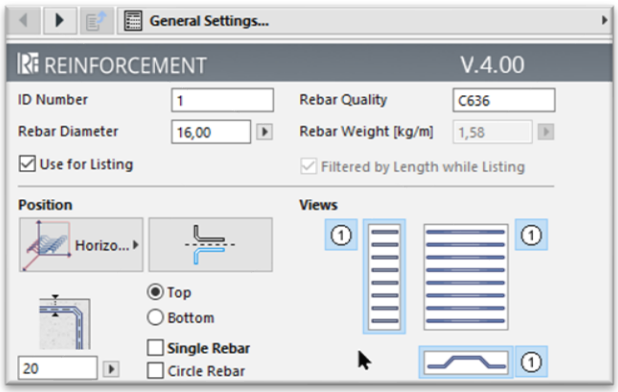
If you cannot find the requested end style of the rebar you can choose the „**Polygonal**” option which helps you define individual rebar shapes (find more information in the next chapter).



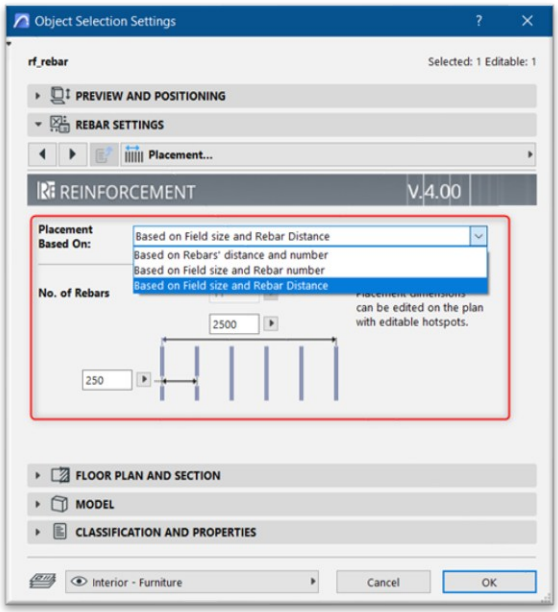


You can choose two options during the rebar's geometrical definition. The solution can calculate with the full length of rebars but you can ignore this filter and filtering the rebars regarding their end's structure. If you switch the „**Calculate with length**” check box during the listing process rebars will be filtered regarding the full lengths, even if 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



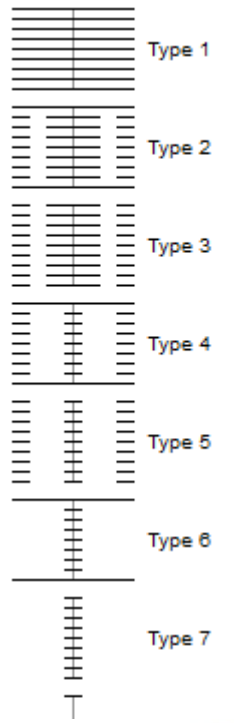
the rebars to define columns or other vertical style complex elements.



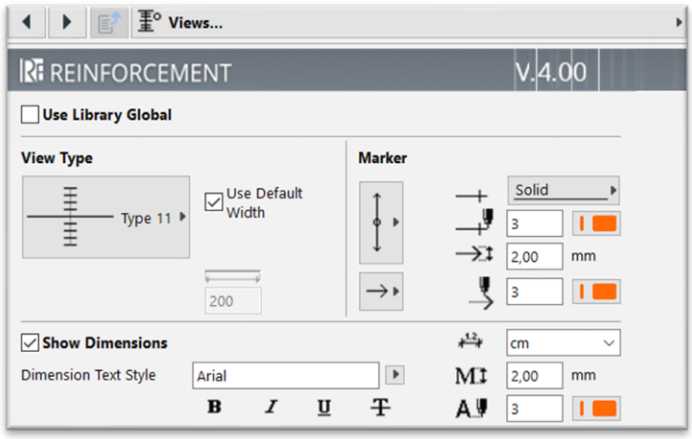
In the „**Placement**” tab you can choose between different types of placement methods. You have the possibility to place the rebars based on the:

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

To place the object on the floorplan, click the „OK” button.



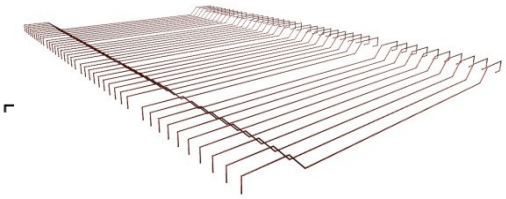
The 2D representation mode and the placement method can be defined with editable hotspots of horizontally placed rebars. Based on the hotspots' position the 2D representation can look different:

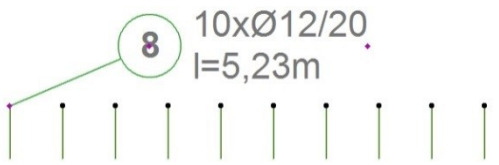


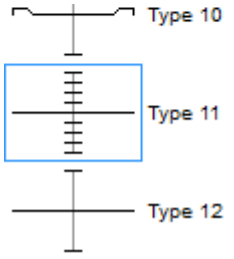
Rebards have 12 basic display options:

- 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 Eptar add-on.







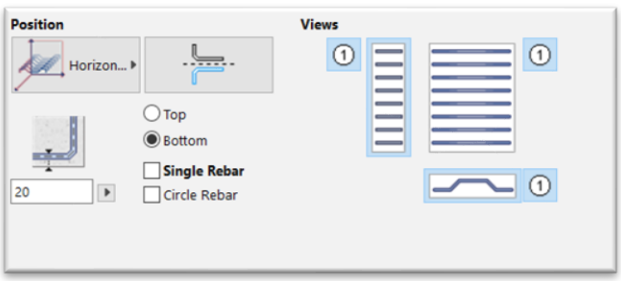
The vertically placed rebar objects' 2D representation is the top view of the rebar. This 2D view is not able to be modified, but you can activate the label for this version as well.

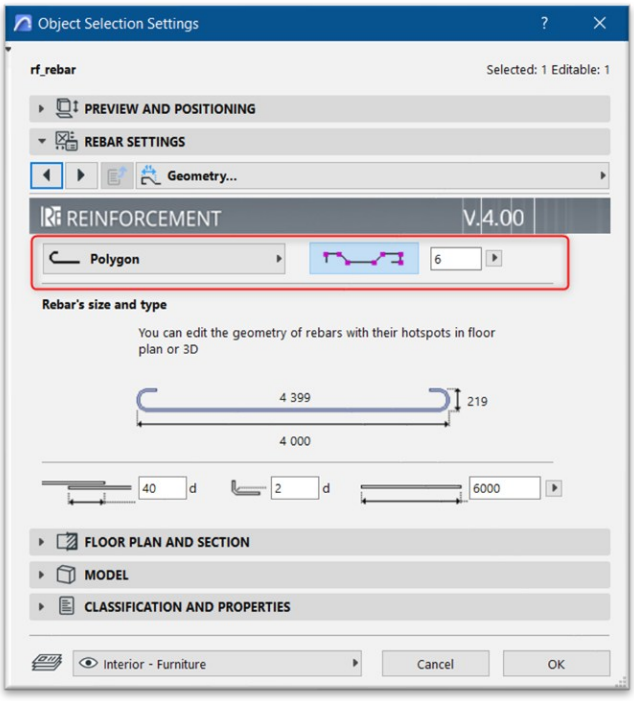
Attention!
Labels will only appear on the floorplan if you have switched on the "**Show Label**" button on the "**General Settings**" page

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.

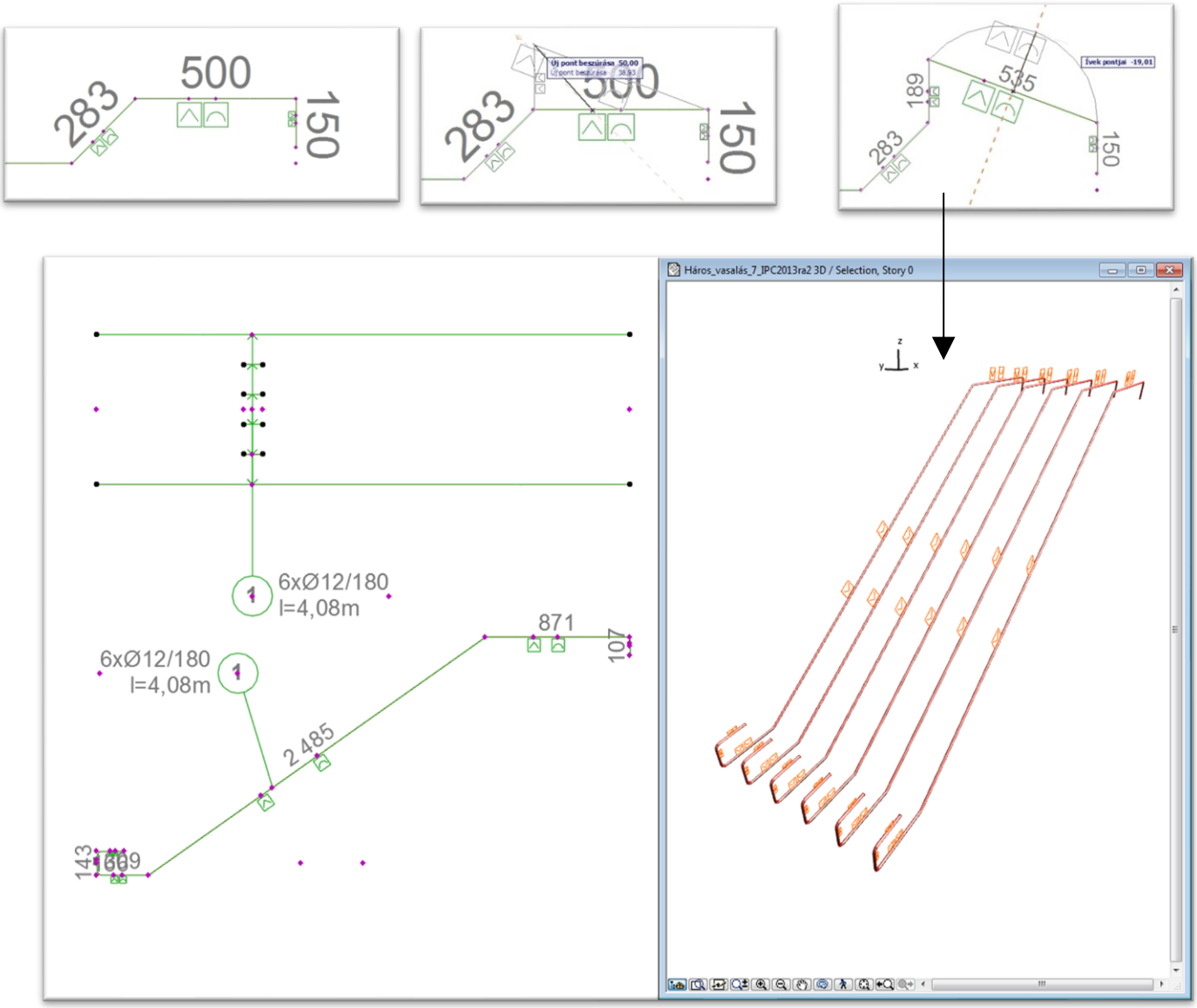
Choose the „**rf_rebar**" object and select the „**Polygon**" style under the „**Geometry**" tab page and click the „**Enable Polygon Editing**" check box.



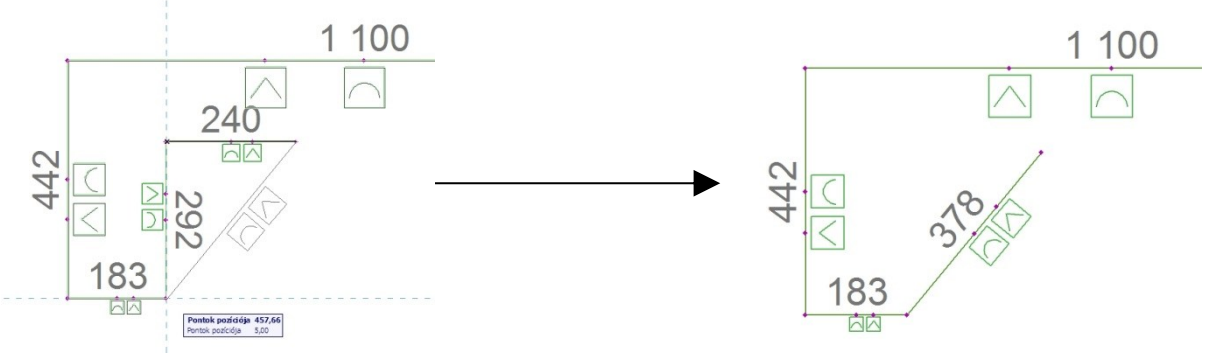


On the Floor Plan you can see the rebar object and the side view of it.

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



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

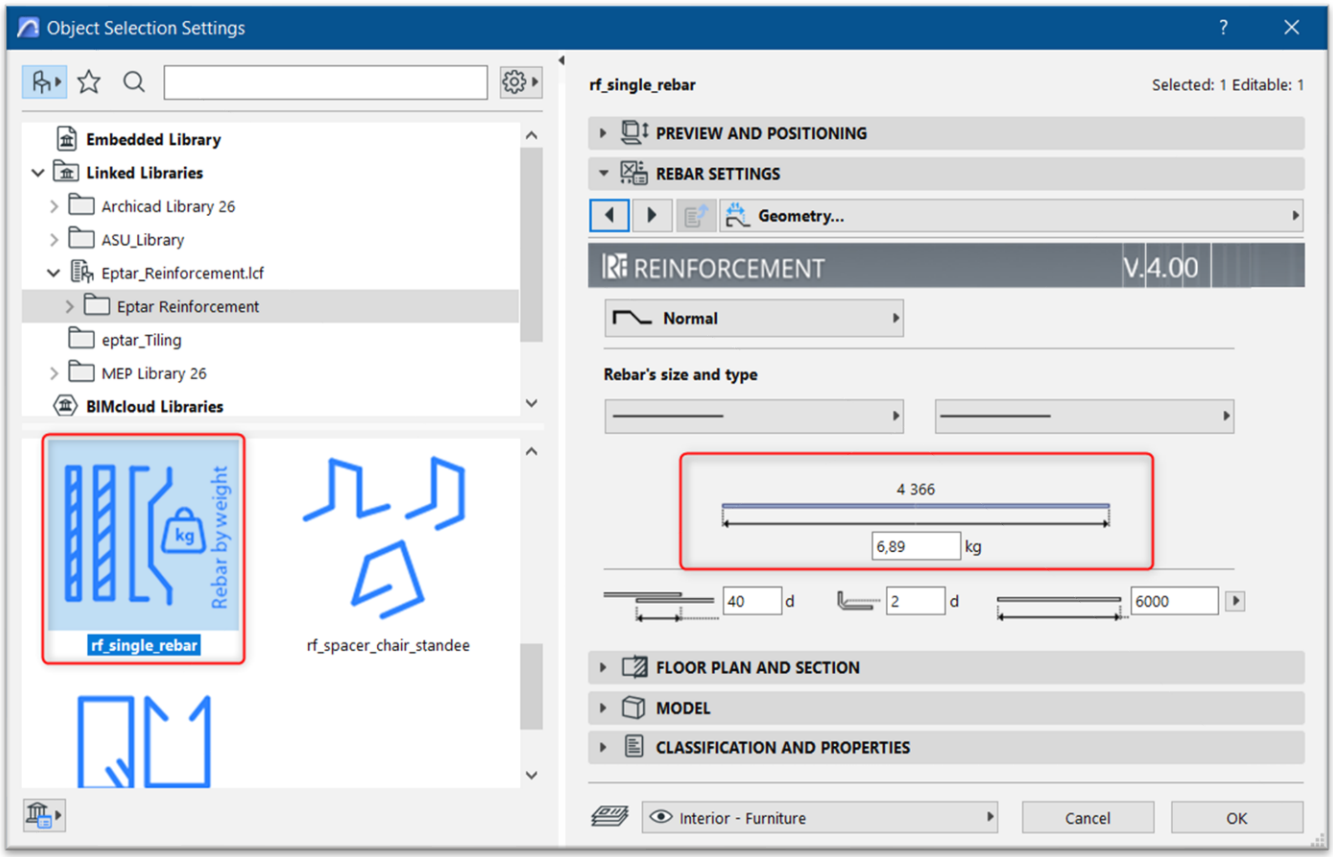


The polygon style rebar can be placed as a single rebar or multiplied for a length.

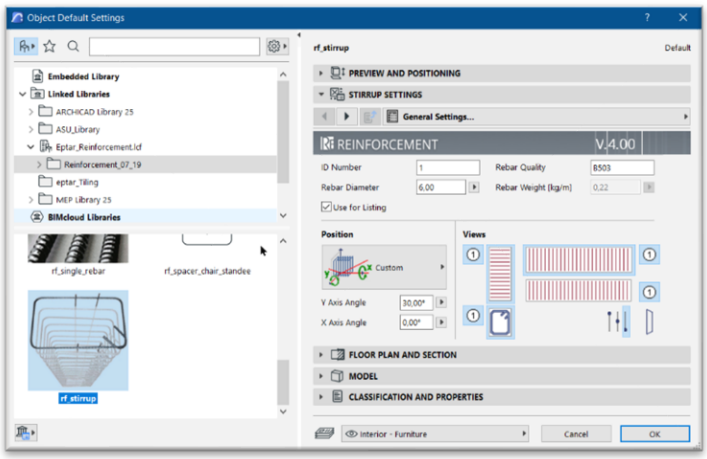
Note
When you create a new corner point on the Polygon path, one of the two new line sections will contain no editable hotspots. If you want to edit this line section (add new corner point), please move one of the polyline's existing corner points and in a second step move back to original position.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

SINGLE REBAR ELEMENT

It is the same as the rebar element but in kilograms. You can ask for meter and only one piece can be placed.



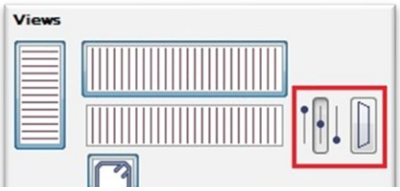
STIRRUP OBJECT

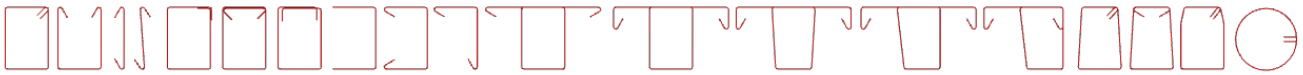


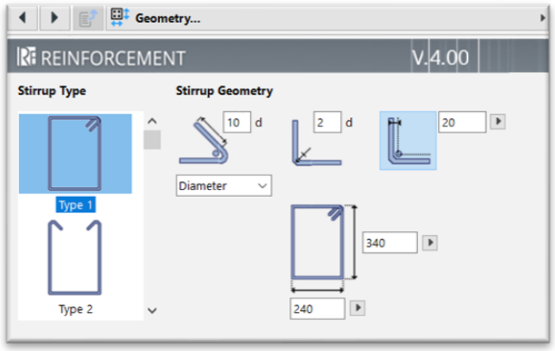
The stirrup object is a basic accessory for more complex elements. With the help of stirrups it is possible to define columns (pillars), beams and crowning objects.
Select the „**rf_stirrup**” object in the object settings dialog.

In “**Position**” menu you can choose vertical (for columns), horizontal (for beams) and custom position for the stirrups. In 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 and the individual stirrups get a slanted position in space.
If you want to make a slanted stirrup group with vertical stirrups, it is recommended to place an individual stirrup and use the built in “**Multipy**” order of ArchiCAD.

In case of custom position stirrups you have many view options to choose from. Besides the usual vertically projected views you can also 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 in the Floor Plan view.





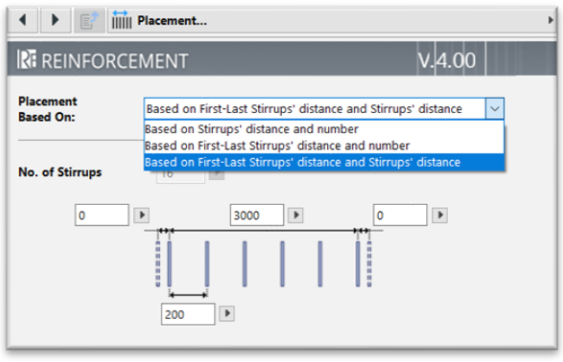


Select the stirrup section form and set up the parameters on the „**Geometry**” tab page.

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 click the “**Show Rebar Distance**”.
If you do not find the necessary predefined shape in the dialog, select the „**Polygon**” style.
The polygon style helps to define free stirrup shapes 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 bar final turnbacks during the polygon definition.

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

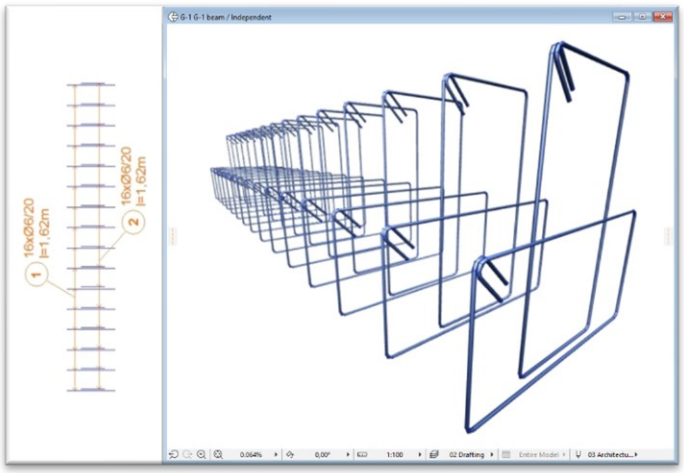
- Length and number
- Number and distance
- Length and distance

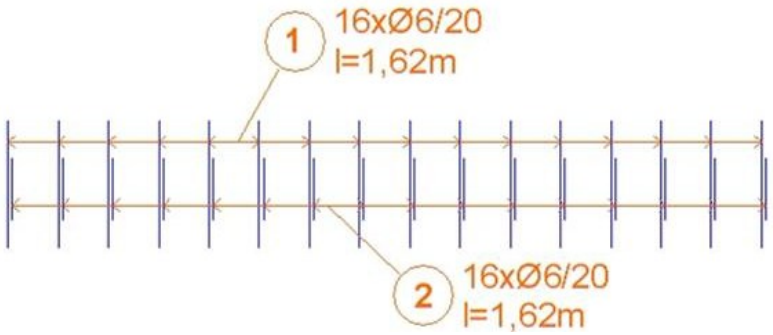


You can edit the distance between the stirrups with editable hotspots on the Floor Plan or in 3D view.

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

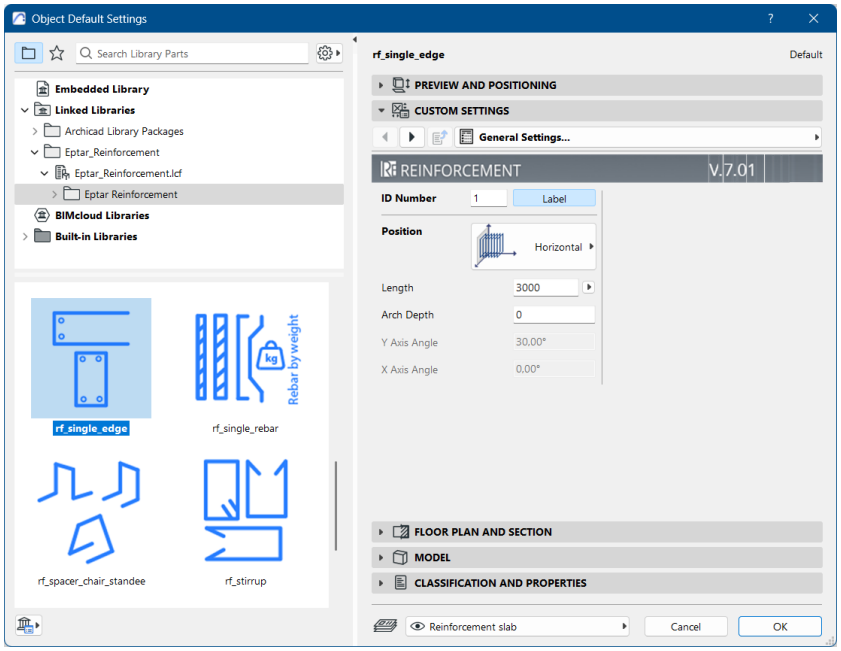
On the Floor Plan you can switch on the Label for the object on the „**2D Representation**" tab page.
You can also use the original ArchiCAD label elements, find more details about this functionality in point 2.8.





- **RF_SINGLE_EDGE ELEMENT**

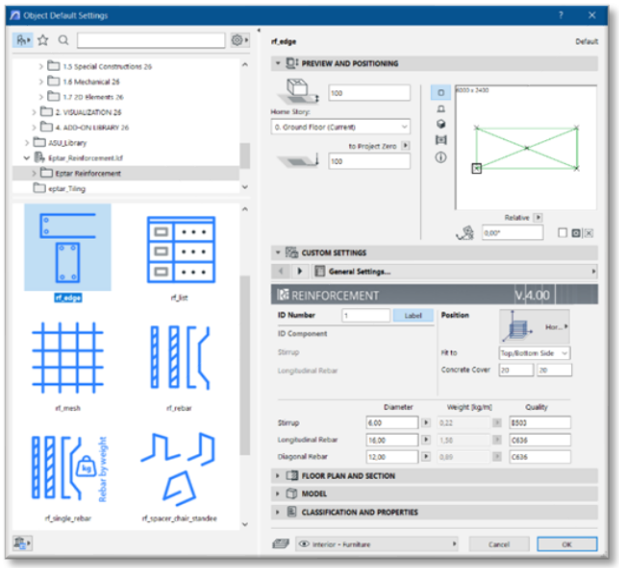
The **"rf_single_edge"** element can be placed in slab and straigth wall element. Even the curved wall is supported yet. The rf_simgle_edge element allows the user to define slab and wall edge reinforcement at its perimeter and at the edge of the placed holes. The components of these reinforcements are possible to be set side by side separately.
To place the rebar, choose the „**rf_single_edge**" object from the dialogue, and click on OK.



The new dialog appears, where you can choose an option, on which edges you would like to place it: place to all edges, or place only to the selected edges.

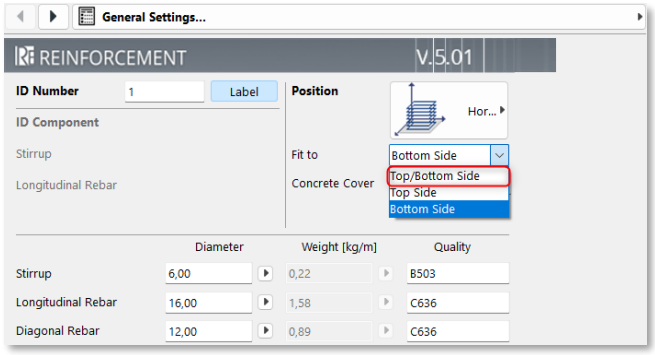
- **The complex RF_EDGE ELEMENT**

The **“rf_edge”** element can be placed in slab and straigth wall element. Curved wall is not supported yet. The rf_edge element allows the user to define slab and wall edge reinforcement at its perimeter and at the edge of the placed holes. The components of these reinforcements are possible to be set side by side separately.
To place the rebar, choose the „**rf_edge**” object from the dialogue.

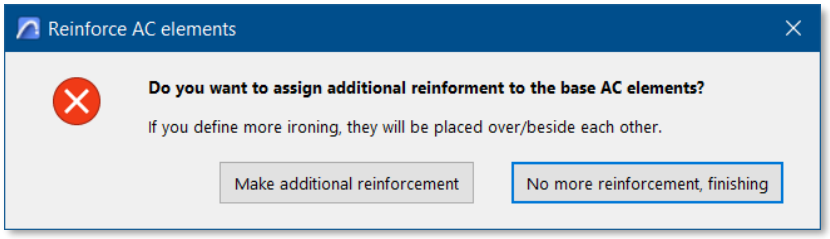


Rf_edge element in slab

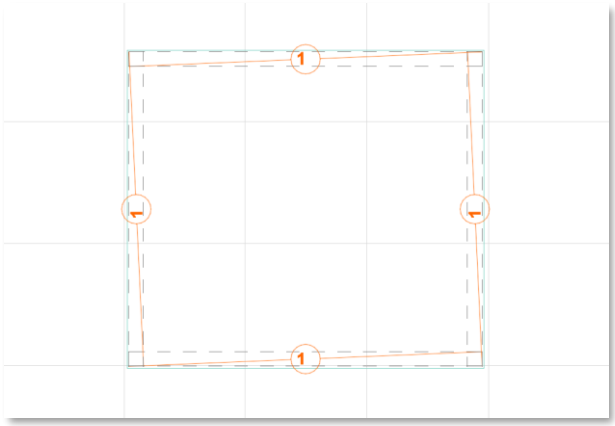
To place the element, as usual, the slab must be selected. On the Reinforcement palette, the "Slab" icon becomes active, then by clicking on it, the Slab reinforcement settings window will appear. Select the rf_edge element from the library elements.
Among the settings of the element, you can specify the ID of the element, its position (horizontal in the slab), the location of the reinforcement in relation to the thickness of the slab, the concrete cover, the diameter of rebars and stirrups, and their material quality. If in option „Fit to” the „Top/Bottom side” is set, the add-on places the reinforcement only in core ot the composite slab.

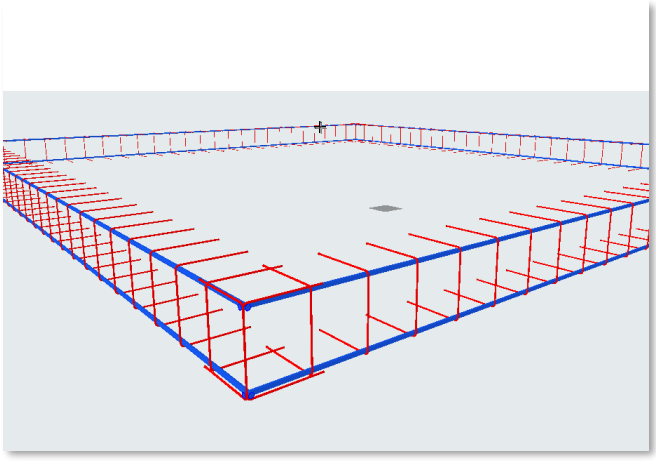


When you have set these parameters, click OK. A window asking "Do you want to assign additional reinforcement to the base AC elements?" will appear. Here it is possible to place additional reinfocements in this element without interrupting the reinforcement process.
If you don't want to place more reinforcements, click the "No more reinforcement, finishing" option!

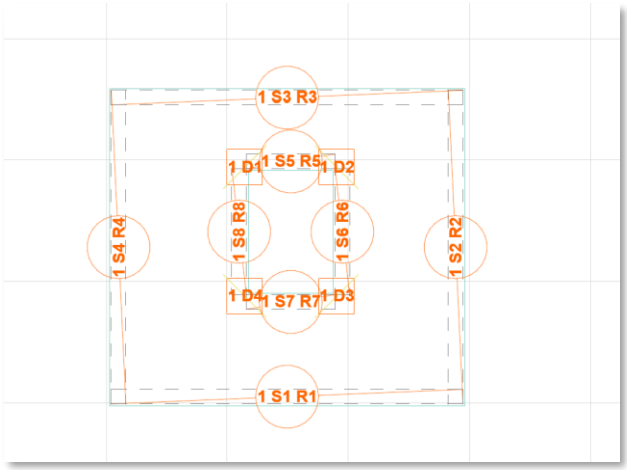


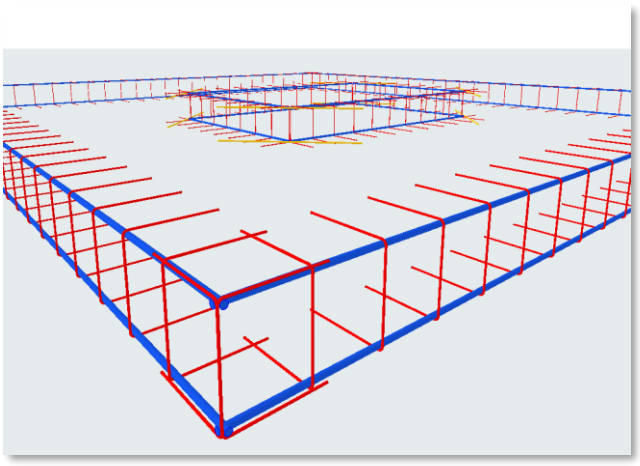
The program places the defined rf_edge element along the edges of the slab.



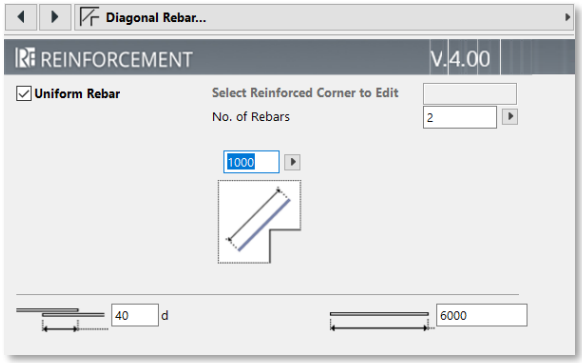


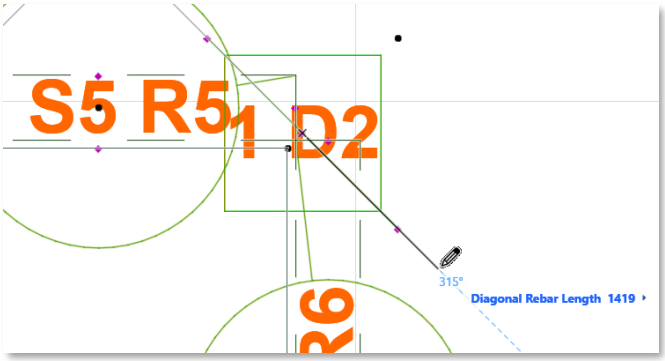
The program also places the reinforcements defined for the outer edges around openings in the slab.

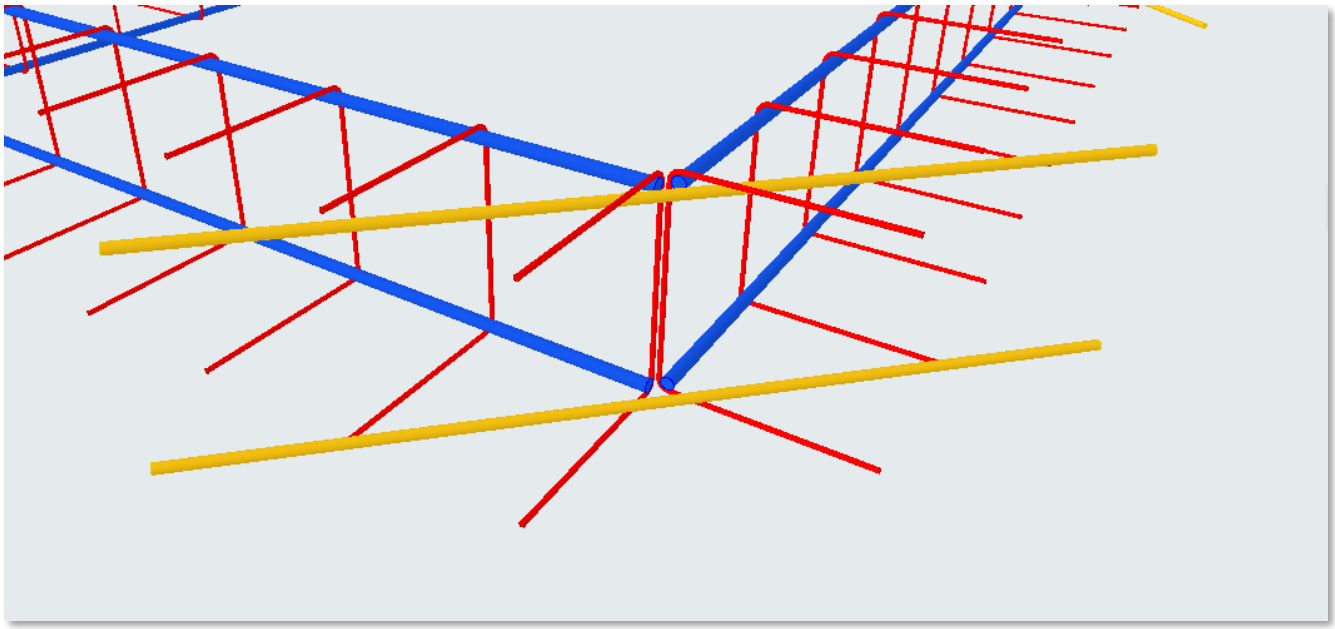




A diagonal iron can also be defined at the corners of the openings, the length of which can be adjusted by adding a value or graphically, it can also be turned off completely.

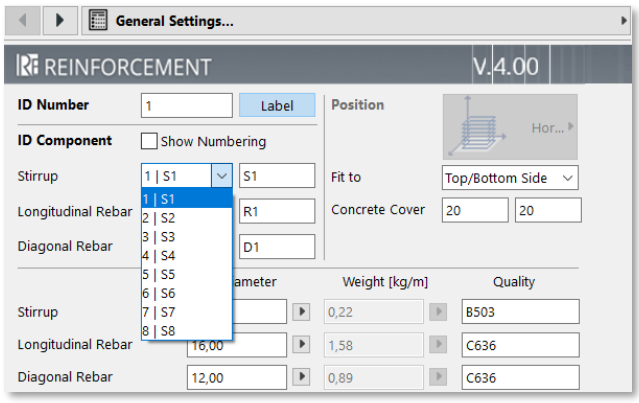


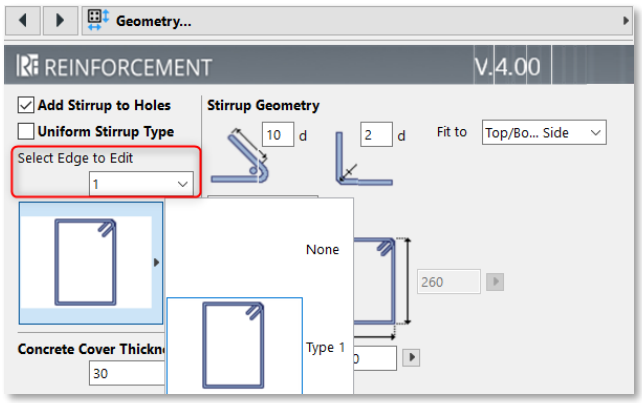




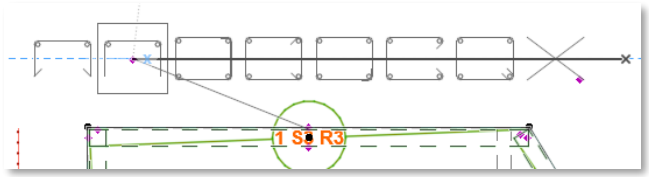
You can modify the parameters of the placed element by selecting it. Select it and open the Object settings window.

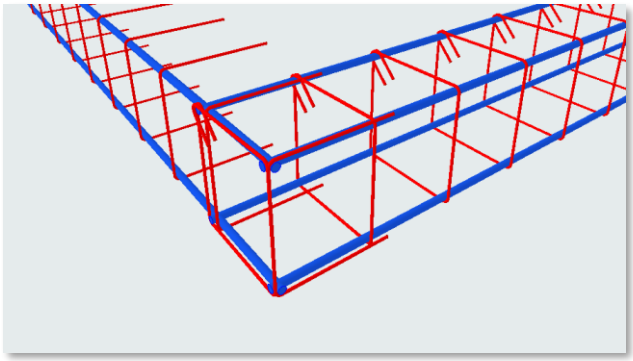
You can immediately see that there are many more options compared to the settings that appear when you place them. The numbering of the rebars and stirrups can be adjusted per edge.





In the next window, you can choose a stirrup type for each edge individually. You can also set the type of stirrups on the floor plan from a graphic menu. If you choose the "no stirrup" option, the reinforcement will not be defined on that edge.



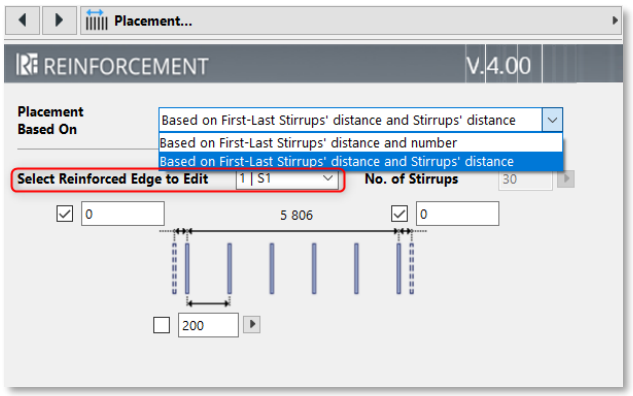


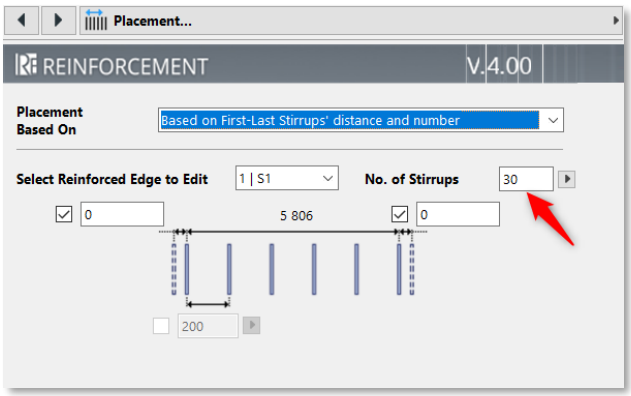
In the case of a closed stirrup, the program automatically defines 4 rebars.

In the next window, you can set stirrup allocation per edge, where you can choose between two options:

- Based on the number of pieces, or
- Based on the distance of stirrups

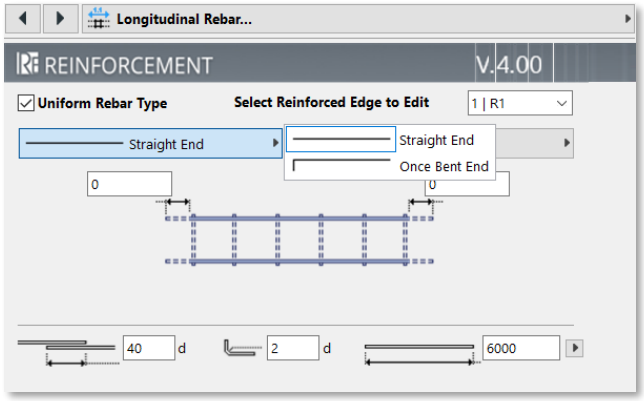
The start and end of the stirrup allocation (the location of the first and last stirrup from one or the other end) can also be set separately.

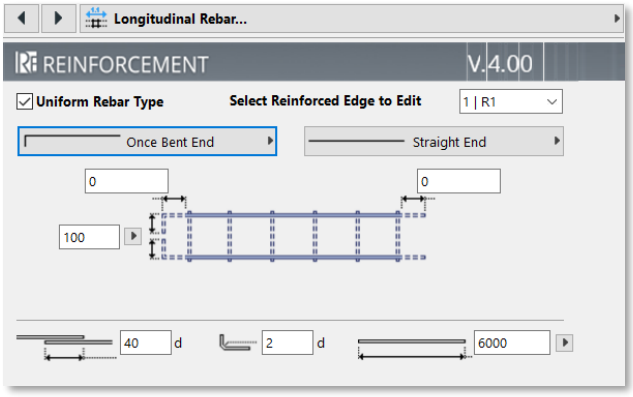




In the next window, you can define the parameters of the rebars per edge:

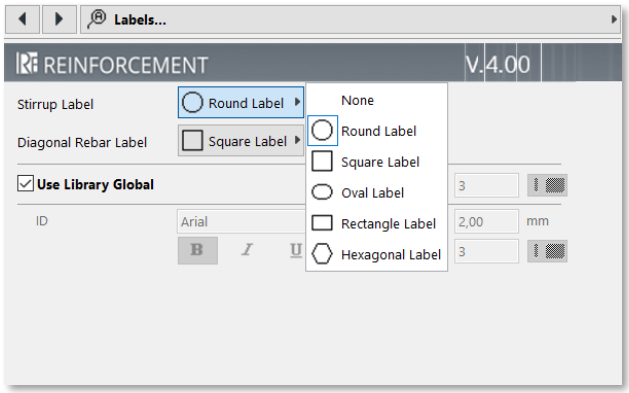
- the shape of the rebars: bends at both ends of the rebar
- rebar overhangs at both ends
- the degree of overlap of the rebars
- the extent of the radius of the bends
- rebar length

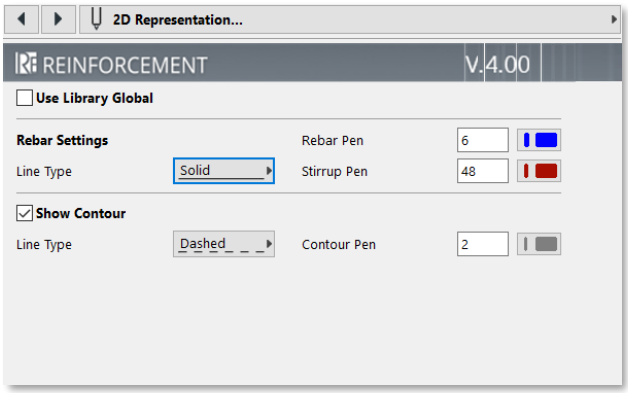


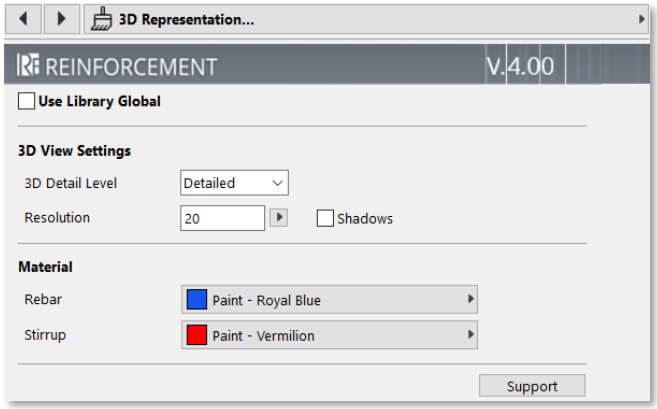


In the next window, you can set the parameters of the diagonal irons placed at the corners of the openings, which was already described previously.

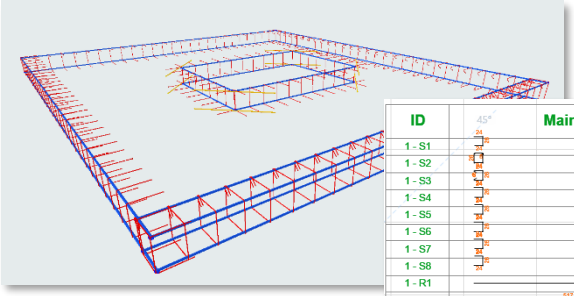
In the last two windows, you can set 2D representations, labels, fonts text sizes, as well as display colors.







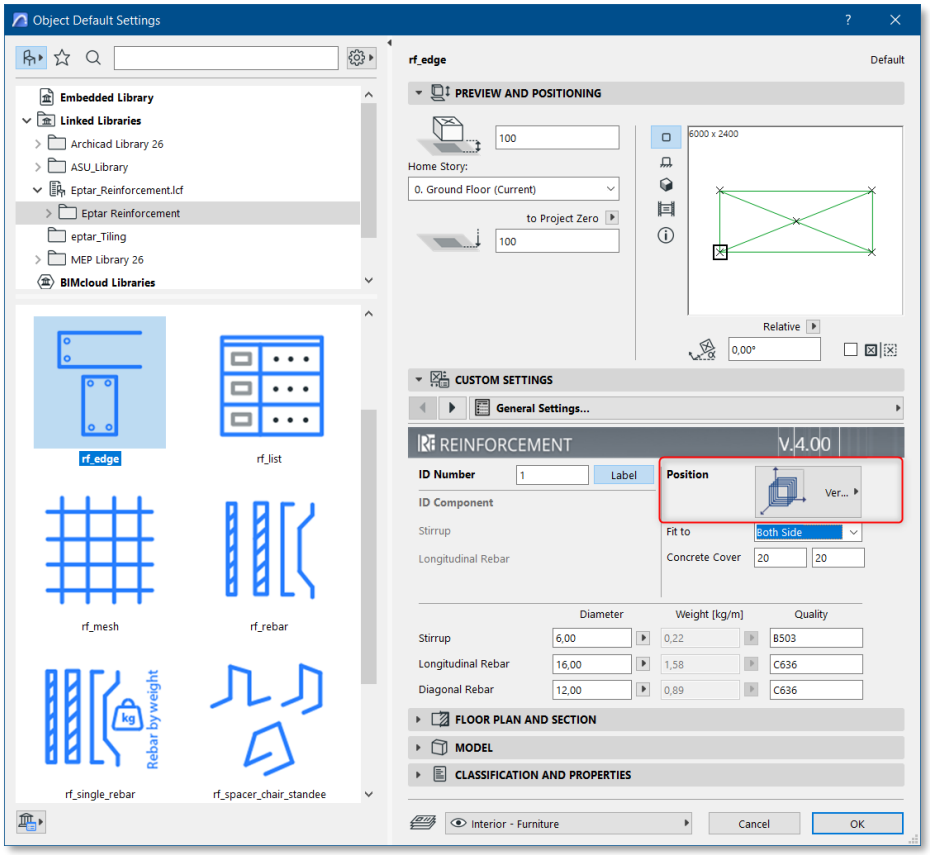
Rf_edge element listing:
select the placed element and click on the "Listing" icon on the Reinforcement palette. A small list icon will appear next to the cursor, click on the floor plan to place the list. The program collects all the components of the selected element in a list and summarizes them



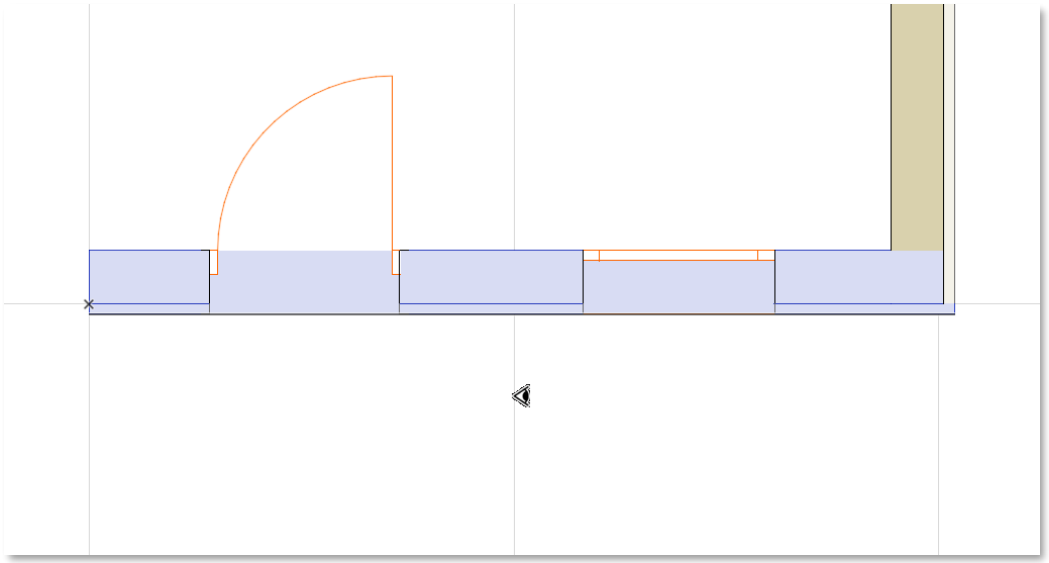
ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1 - S1		6 mm	30	0,740 m	22,200 m	B503	4,9 kg
1 - S2		6 mm	27	1,120 m	30,240 m	B503	6,7 kg
1 - S3		6 mm	30	0,860 m	25,800 m	B503	5,7 kg
1 - S4		6 mm	27	0,740 m	19,980 m	B503	4,4 kg
1 - S5		6 mm	8	0,740 m	5,920 m	B503	1,3 kg
1 - S6		6 mm	11	0,740 m	8,140 m	B503	1,8 kg
1 - S7		6 mm	8	0,740 m	5,920 m	B503	1,3 kg
1 - S8		6 mm	11	0,740 m	8,140 m	B503	1,8 kg
1 - R1		16 mm	2	5,806 m	11,611 m	C636	18,3 kg
1 - R2		16 mm	4	5,167 m	20,670 m	C636	32,6 kg
1 - R3		16 mm	2	5,806 m	11,611 m	C636	18,3 kg
1 - R4		16 mm	2	5,167 m	10,335 m	C636	16,3 kg
1 - R5		16 mm	2	1,465 m	2,929 m	C636	4,6 kg
1 - R6		16 mm	2	2,075 m	4,151 m	C636	6,5 kg
1 - R7		16 mm	2	1,465 m	2,929 m	C636	4,6 kg
1 - R8		16 mm	2	2,075 m	4,151 m	C636	6,5 kg
1 - D1		12 mm	2	1,000 m	2,000 m	C636	1,8 kg
1 - D2		12 mm	2	1,000 m	2,000 m	C636	1,8 kg
1 - D3		12 mm	2	1,000 m	2,000 m	C636	1,8 kg
1 - D4		12 mm	2	1,000 m	2,000 m	C636	1,8 kg
		6 mm sum:					28,05 kg
		16 mm sum:					107,91 kg
		12 mm sum:					7,10 kg
		B503 sum:					28,05 kg
		C636 sum:					115,02 kg
		Total Weight:					143,07 kg

Use of rf_edge element in wall elements

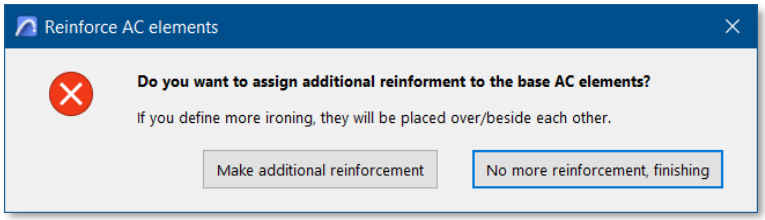
Of course, this element can not only be used in slabs, but also in straight walls. To place the element in the usual way, you must select the wall, then click on the "Wall" tool in the Reinforcement palette to open the wall reinforcement settings window. Select the rf_edge element from the list. As with the slab, only certain parameters can be set. What is important, is to set the position of the reinforcement to vertical, then click the OK button after setting the other parameters.



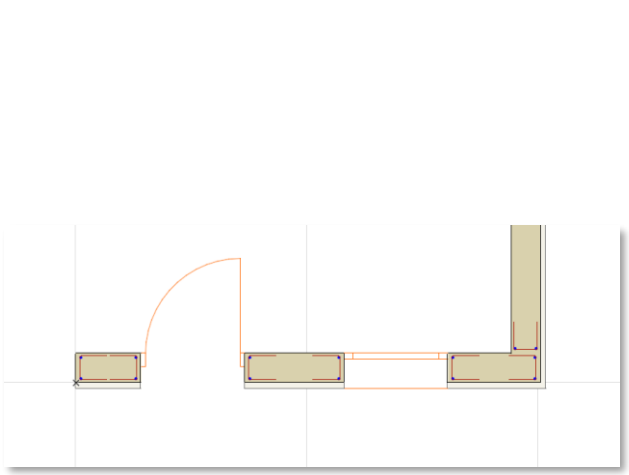
The cursor changes to an "eye" shape, and the rf_edge element is placed by clicking along the walls.

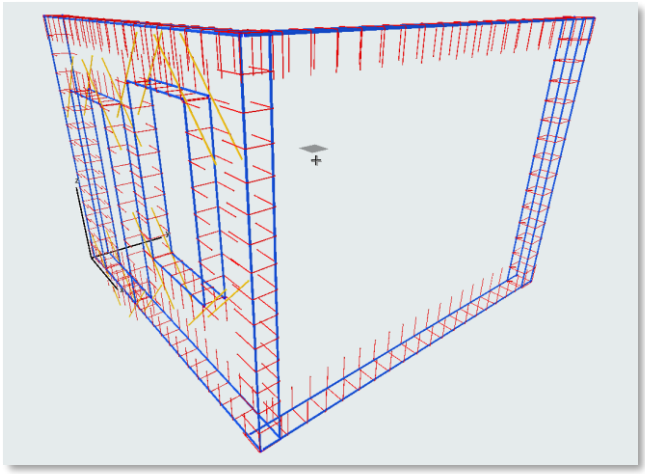


The program asks if you want to place additional irons. If yes, click the "Make additional reinforcement" button, but if not, the program returns to the floor plan with the "No more reinforcement, finishing" button.

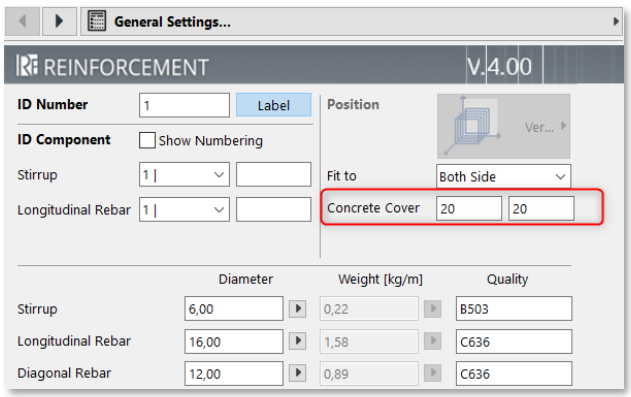


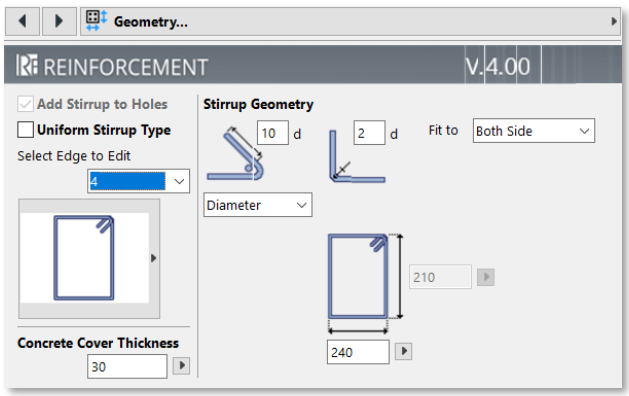
The reinforcements are placed in the selected walls.



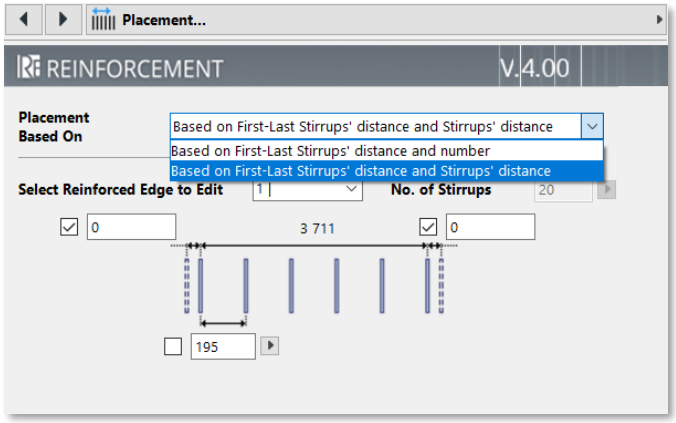


You can change the parameters of the placed reinforcements by selecting them. After selection, open the Object settings window. You can see that significantly more parameters can be set now than before placement.
You can set the labeling, diameter, and quality of the rabars and stirrups individually. It is important that the concrete cover can also be adjusted here.
In the next window, you can specify the type of stirrups and other parameters.

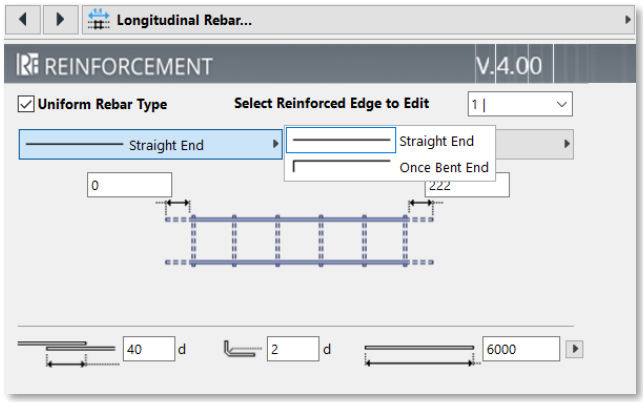


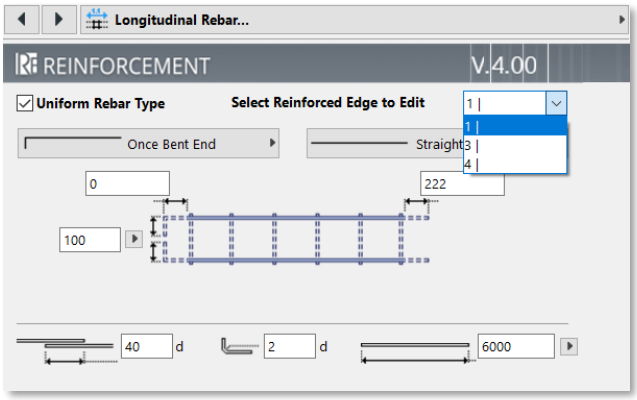


In the next window, stirrup allocation can be set, which can also be done separately for every edge.

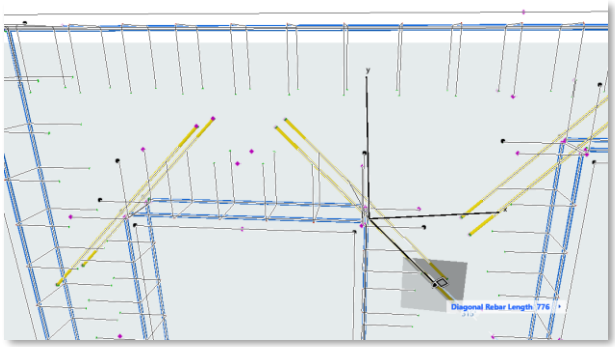


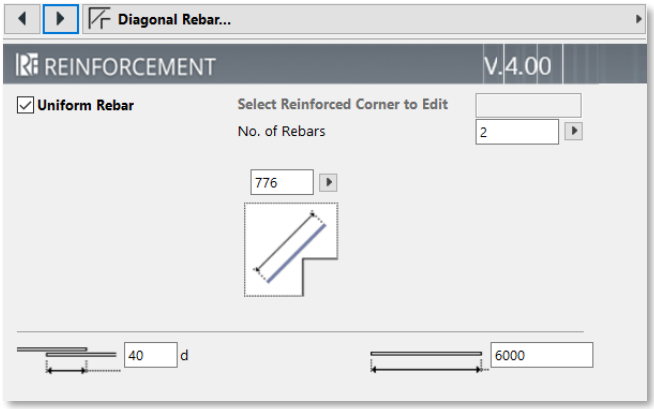
Moving on, we can set the shape, length, overlap, and shape of the reinforcement. If the ends of the reinforcement are bend, we can specify the extent of the bend.
By turning on the "Uniform rebar type" switch, the settings will be applied to all rebars in the element.





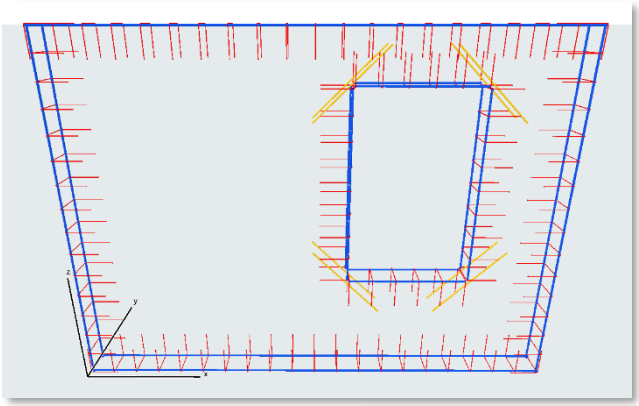
A slanted iron can also be defined at the corners of the openings, the length of which can be adjusted by adding a value or graphically, it can also be turned off completely.





Rf_edge element listing:

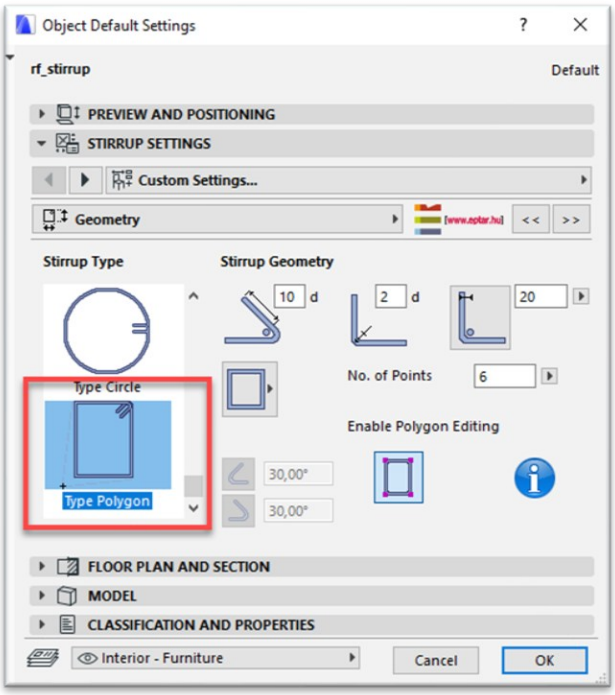
- select the placed element and click on the "Listing" icon on the Reinforcement palette. A small list icon will appear next to the cursor, click on the floor plan to place the list. The program collects all the components of the selected element in a list and summarizes them



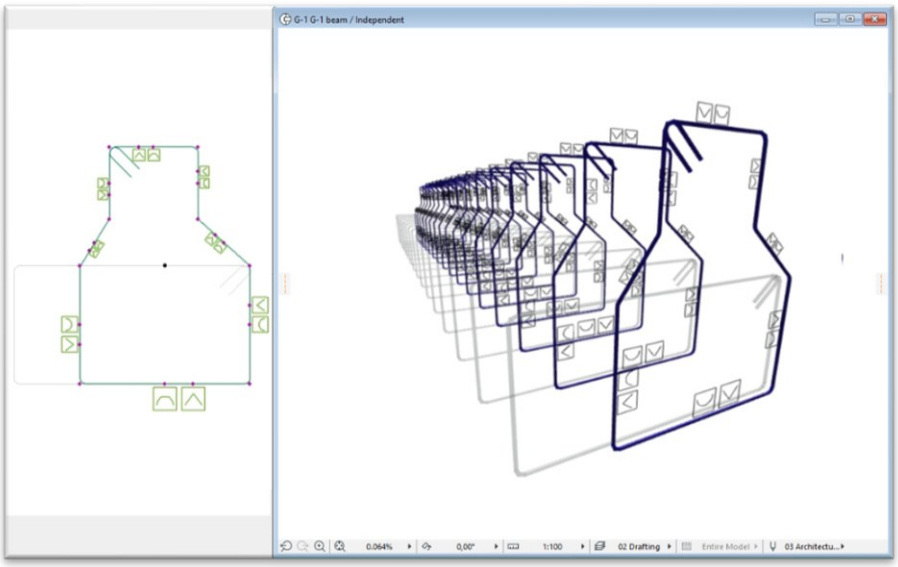
ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1 - S1		6 mm	16	0,690 m	11,040 m	B503	2,5 kg
1 - S2		6 mm	21	0,690 m	14,490 m	B503	3,2 kg
1 - S3		6 mm	16	0,690 m	11,040 m	B503	2,5 kg
1 - S4		6 mm	21	0,690 m	14,490 m	B503	3,2 kg
1 - S5		6 mm	6	0,690 m	4,140 m	B503	0,9 kg
1 - S6		6 mm	9	0,690 m	6,210 m	B503	1,4 kg
1 - S7		6 mm	6	0,690 m	4,140 m	B503	0,9 kg
1 - S8		6 mm	9	0,690 m	6,210 m	B503	1,4 kg
1 - R1	294	16 mm	2	2,940 m	5,880 m	C636	9,3 kg
1 - R2	396	16 mm	2	3,961 m	7,923 m	C636	12,5 kg
1 - R3	294	16 mm	2	2,940 m	5,880 m	C636	9,3 kg
1 - R4	396	16 mm	2	3,961 m	7,923 m	C636	12,5 kg
1 - R5	96	16 mm	2	0,960 m	1,920 m	C636	3,0 kg
1 - R6	156	16 mm	2	1,560 m	3,120 m	C636	4,9 kg
1 - R7	96	16 mm	2	0,960 m	1,920 m	C636	3,0 kg
1 - R8	156	16 mm	2	1,560 m	3,120 m	C636	4,9 kg
1 - D1	78	12 mm	2	0,776 m	1,551 m	C636	1,4 kg
1 - D2	78	12 mm	2	0,776 m	1,551 m	C636	1,4 kg
1 - D3	78	12 mm	2	0,776 m	1,551 m	C636	1,4 kg
1 - D4	78	12 mm	2	0,776 m	1,551 m	C636	1,4 kg
		6 mm sum:					15,93 kg
		16 mm sum:					59,47 kg
		12 mm sum:					5,51 kg
		B503 sum:					15,93 kg
		C636 sum:					64,98 kg
		Total Weight:					80,91 kg

STIRRUP OBJECT- POLYGON

If you choose the „**Polygon**” shape placement into the Stirrup settings dialog, you can define free, polygon path defined stirrup objects.
Select the „**rf_stirrup**” object. On the settings dialog on the „**Geometry**” tab page select the „**Polygon**” shape.
Click „OK”, place the object on the Floor Plan. The solution places a closed or opened stirrup polygon into the scene.

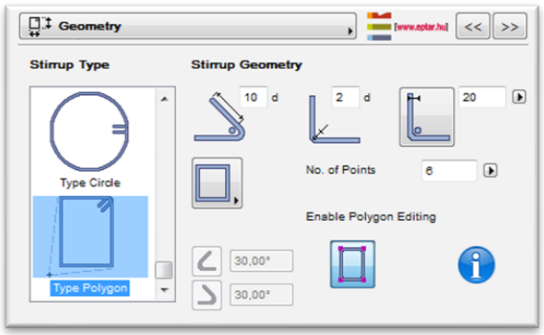


You can find two editable hotspots on every edge of the stirrup. One of these hotspots helps to insert a new point into the polyline, the other defines arc instead of straight section-line.
Move the editable hotspots and define the required section shape.



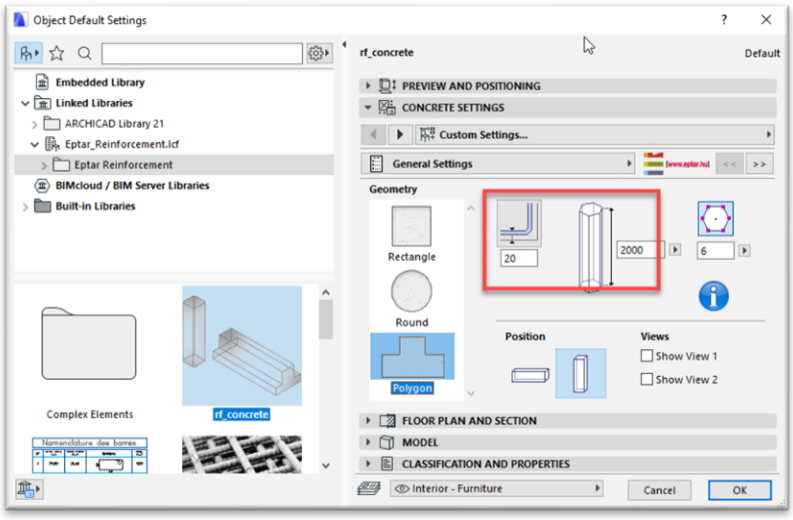
When you finish the shape definition, switch off the „**Editable hotspots**” check box. The small icons will disappear on the 2D view and 3D model.

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

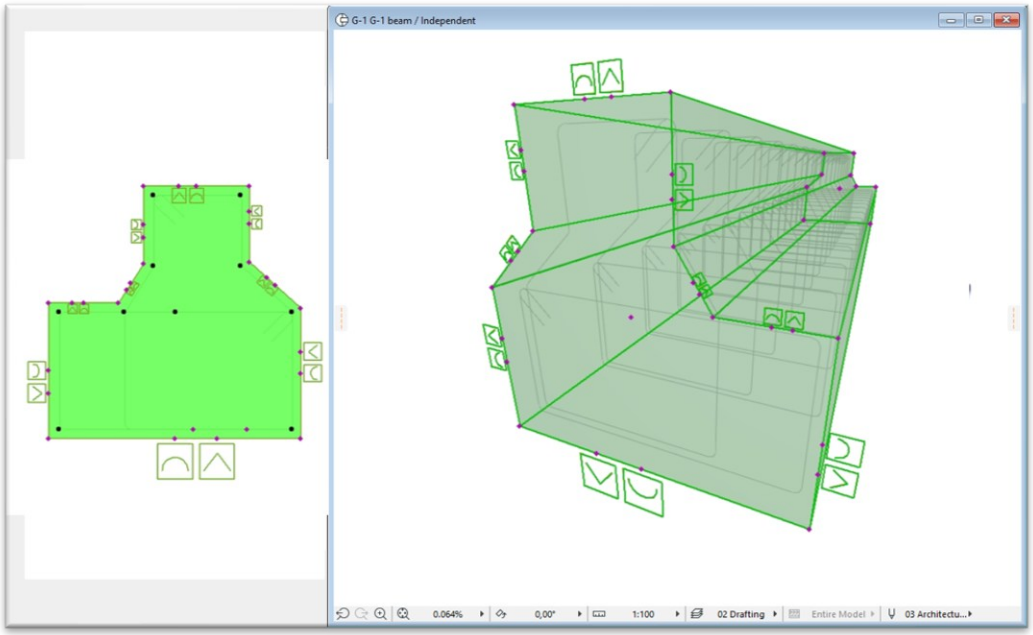
Concrete object is an auxiliary object to define complex elements. The object helps to place rebars and stirrups to the correct position regarding the defined concrete cover.



The 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 is 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.

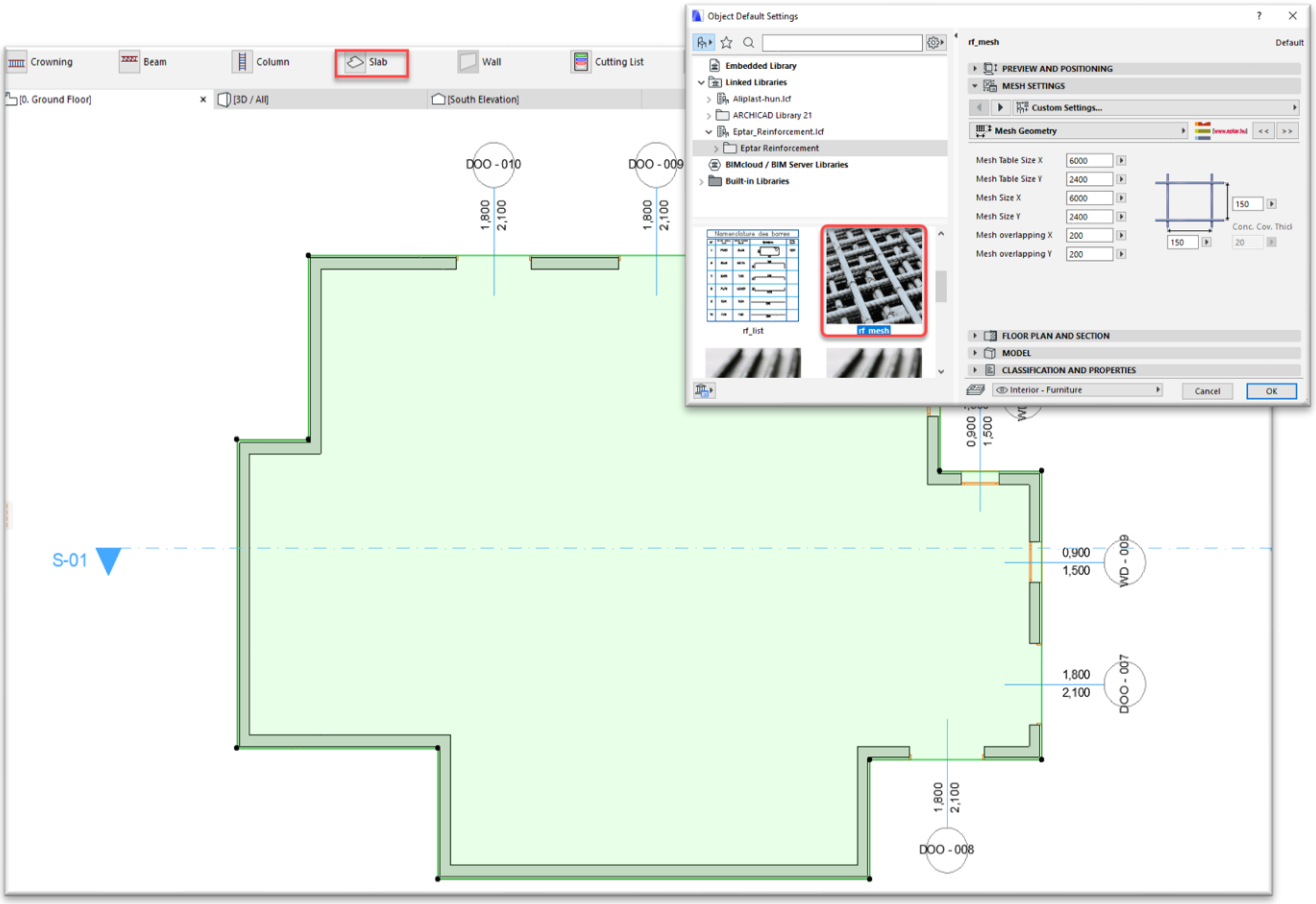
If you set up the „**Bar Distance**" parameter in the „**Objects settings**" dialog, there will be hotspots and hotlines to define the exact placement points for rebars and stirrups. If you click "Show Bar Distance" you will be able to see the placement lines.
The object is easy to modify with editable hotspots on the Floor Plan, section- or in 3D-view.



These objects are not available in the cutting list and do not count in the Rebar optimalization method.

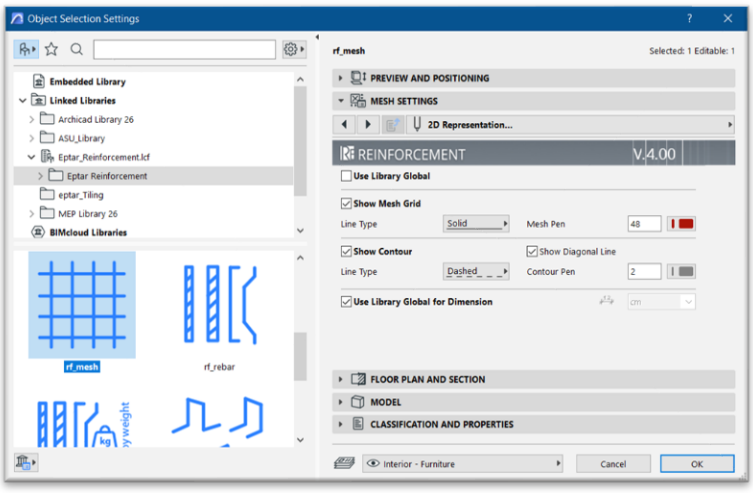
2.4 Object placement with Eptar add-on.

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



Select the ArchiCAD slab or wall objects to fill out with rebar or mesh elements. Use „**Floating palette / Slab or Wall**” 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 the „OK” button. ArchiCAD automatically places the rebar or mesh elements into the polygon shape slab or wall with the parameters already defined.

To change the placement direction of the rebar / mesh objects, select the object and click „**Enable polygon editing**” or „**Show Sections**” check box on the Object settings dialog. Click „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 off the „**Enable Polygon Editing**” check box.



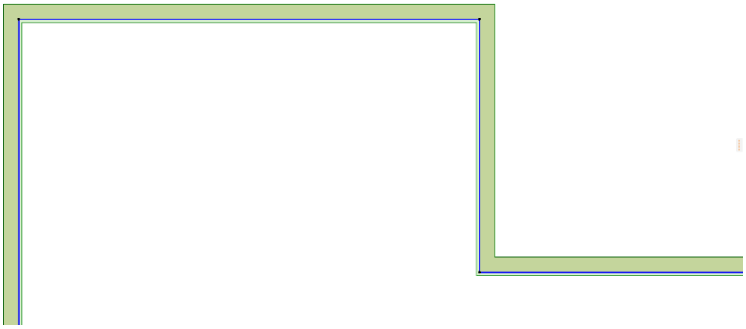
After installing and registration, the **"Reinforcement"** menu item appears in the menu toolbar. Click the **„Load library / start toolbar”** button to display the freely movable floating palette. After selecting the item you want to be reinforced, the program will automatically recognize that you want to reinforce a wall or a slab. On the floating palette, the current wall / slab icon flashes. After clicking, the item selection window pops up and you can choose to insert a mesh or a rebar element. Once you have set everything up, the program will ask you if you want to place more reinforcements. If so, then the item selector will reappear, if not, the dialog closes and the program places the selected types of objects. In case of slabs, the program only allows the horizontal bars to be placed. In case of walls, the program only allows vertical or lateral bars to be placed. 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.

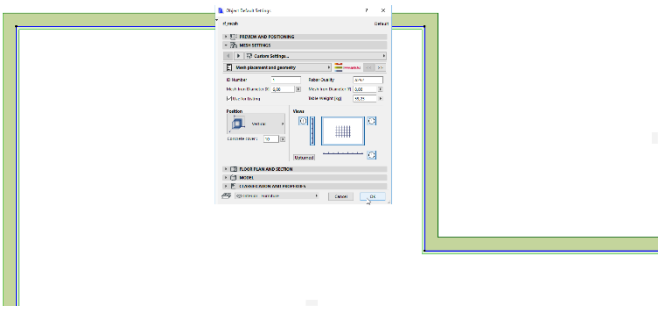
It is possible to reinforce several slabs or walls simultaneously. It is important that only slabs or walls can be simultaneously reinforced. If a slab and a wall are selected at the same time, an error message of the solution will appear.

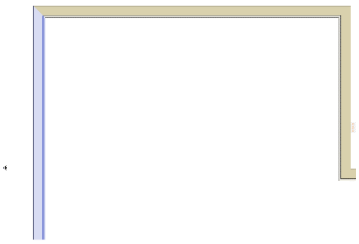
When fixing a wall, the solution does not make the reinforcement of the corners.

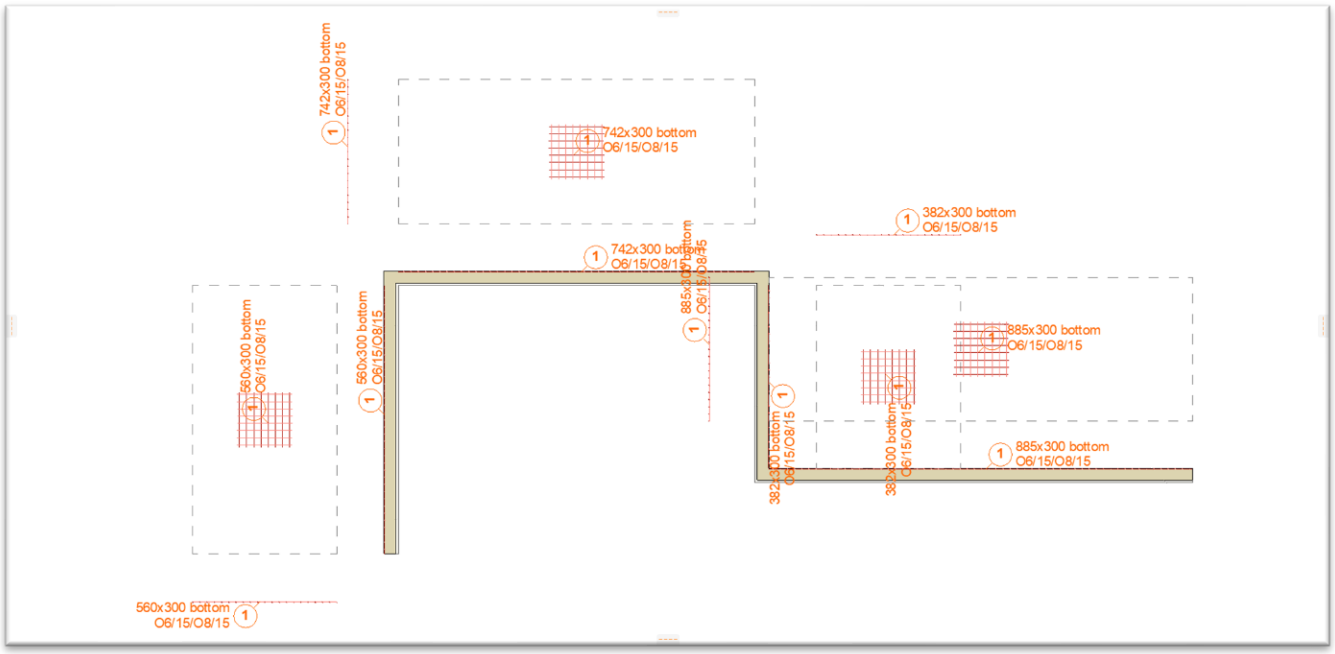
Select the wall elements on the Floor Plan which you want to have rebars or meshes. Click the wall icon on the floating palette. Select the rebar or mesh object in the dialog box and set up the necessary parameters as we defined in the earlier chapter. Click the "OK" button. The solution returns to the Floor Plan where you need to click to specify which side you want to place the mesh / rebar elements. The solution asks if you want to place more reinforcements, if not, the solution automatically places the mesh / rebar elements on the Floor Plan.

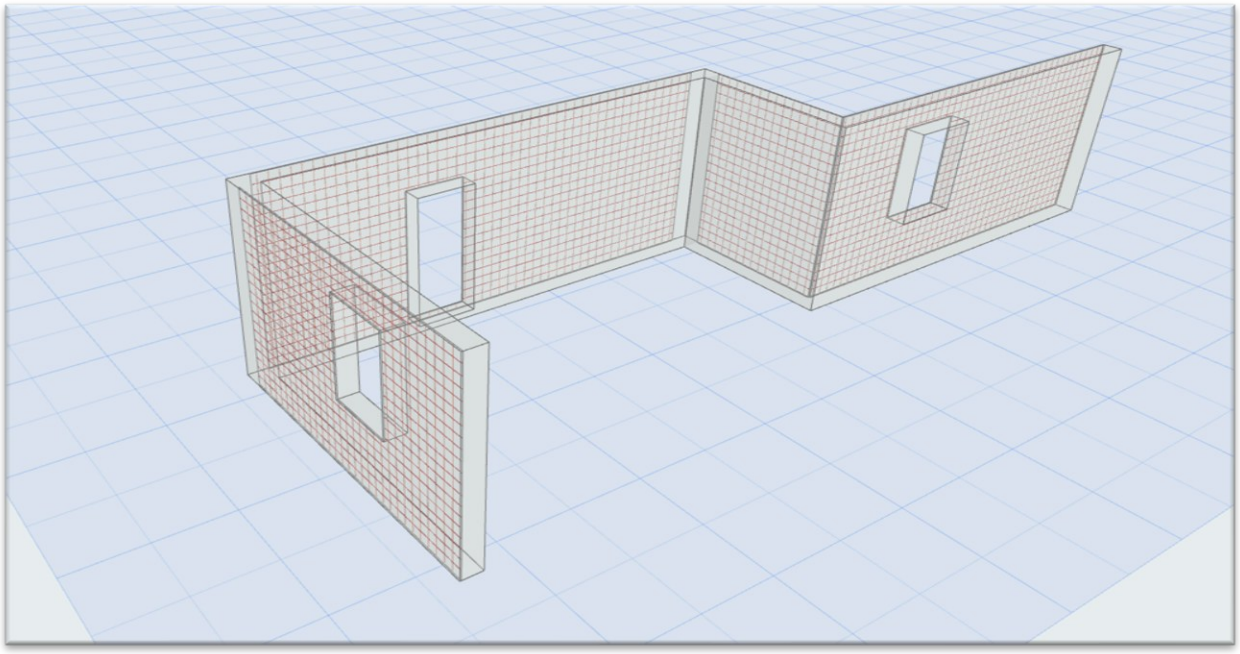




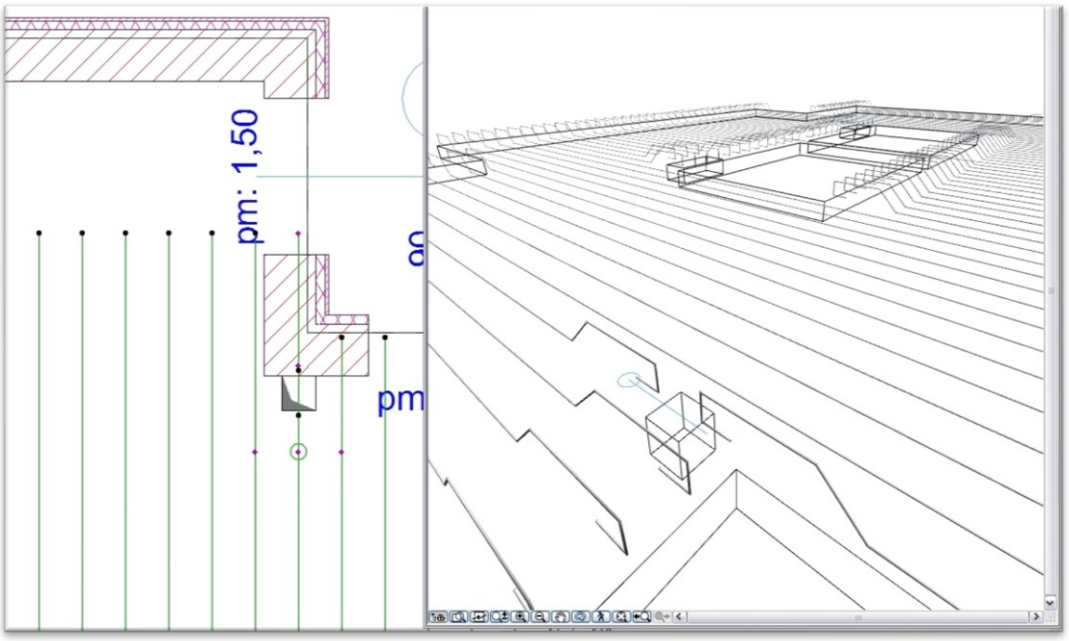





742x300 bottom
O6/15/O8/15
742x300 bottom
O6/15/O8/15
382x300 bottom
O6/15/O8/15
742x300 bottom
O6/15/O8/15
885x300 bottom
O6/15/O8/15
560x300 bottom
O6/15/O8/15
560x300 bottom
O6/15/O8/15
885x300 bottom
O6/15/O8/15
382x300 bottom
O6/15/O8/15
382x300 bottom
O6/15/O8/15
885x300 bottom
O6/15/O8/15
560x300 bottom
O6/15/O8/15



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 does not hesitate to finish a rebar if it crosses the hole and place a new one after the hole. It can cause problems, if you are not careful enough.



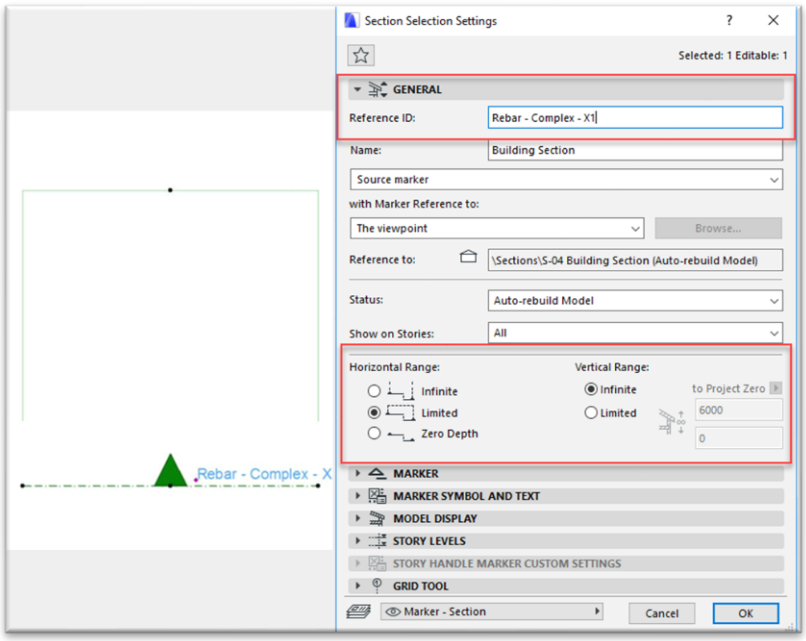
The rebars and meshes - which have been placed with Éptar's add-on – will be grouped together, so the styles of the rebar or meshes will be in one placed group.
In this 3.5 version the mesh does not support the calculation of overlapping and does not calculate the optimal placement direction. We are working on creating a new update which supports this functionality.

2.5 Creating Complex structures

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

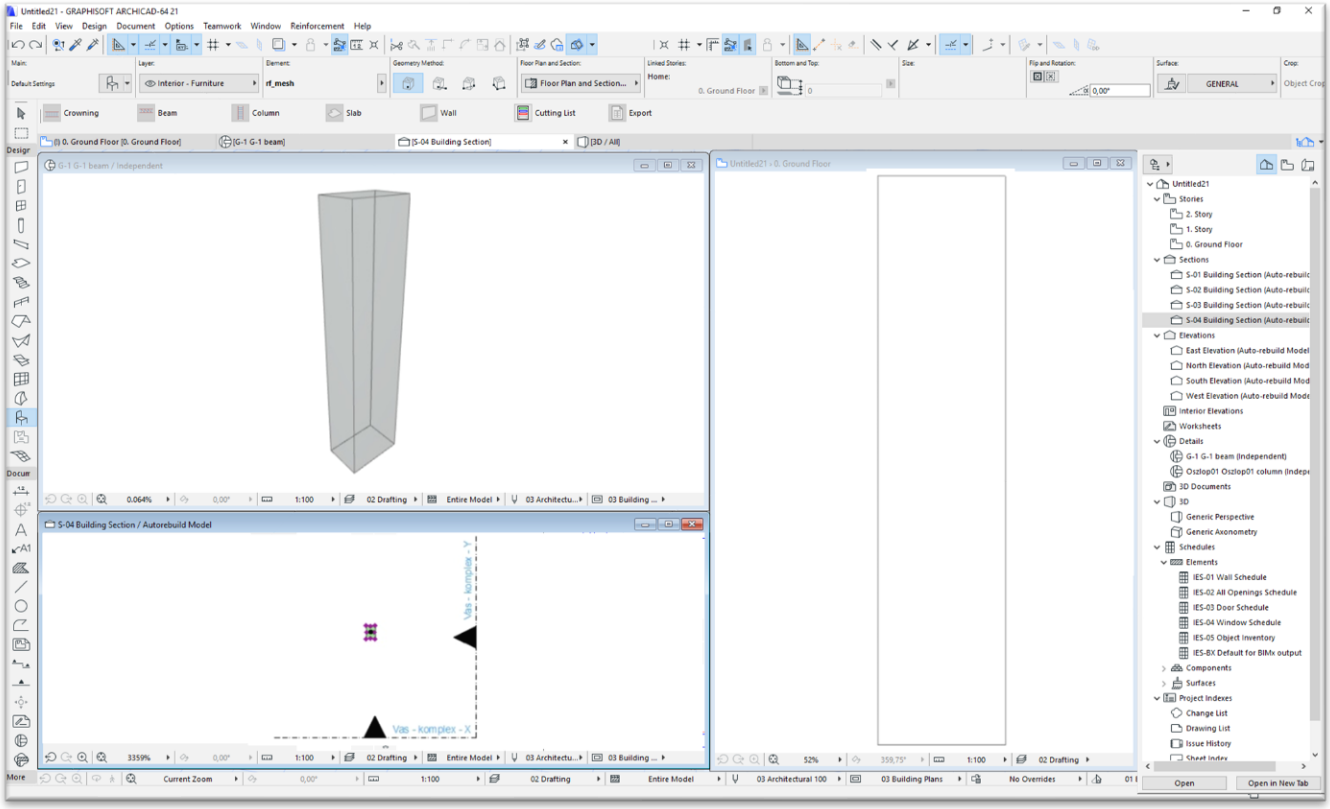
2.5.1 Working Area

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.

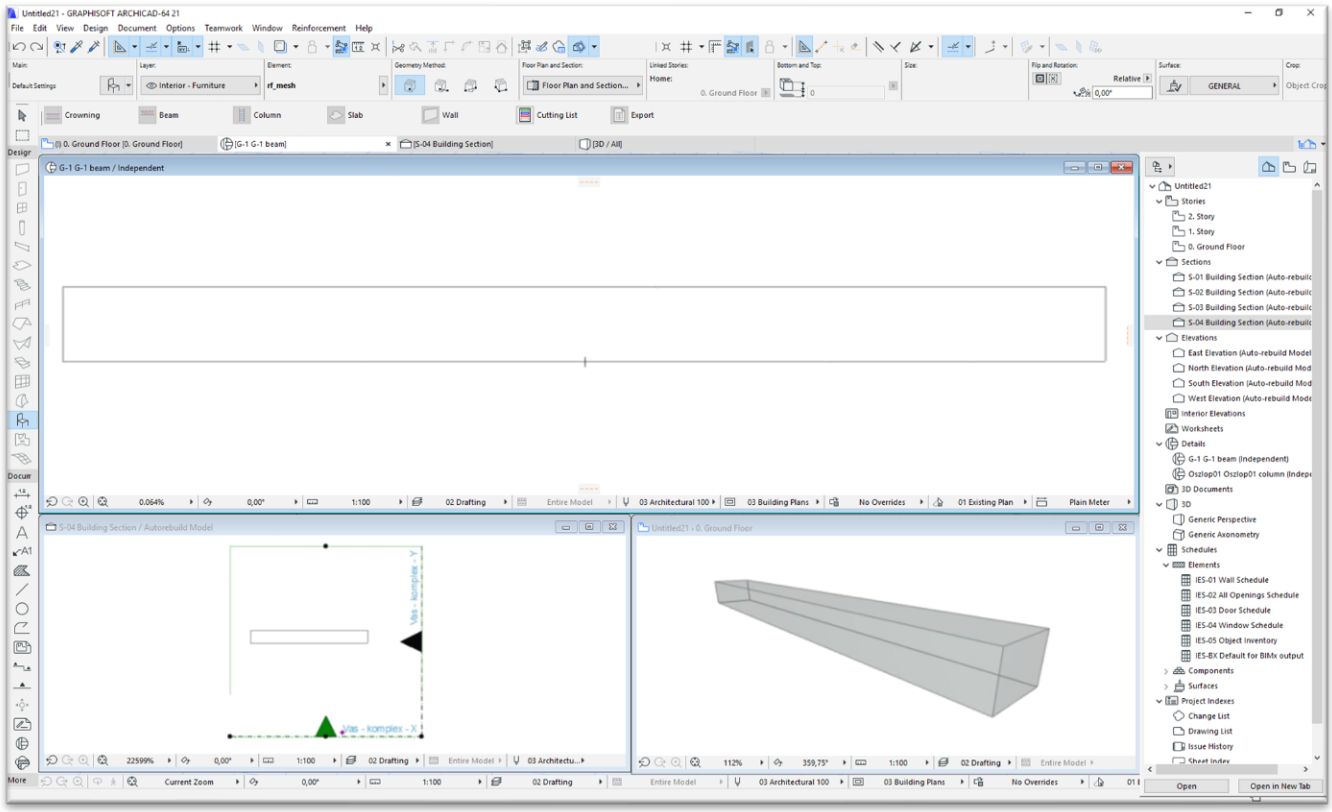


Move the copy of the section on the Floor Plan and rotate it with 90 degrees, these two sections will represent the front and side view. Change the name of the copied section to „**Vas – complex – Y**”.

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



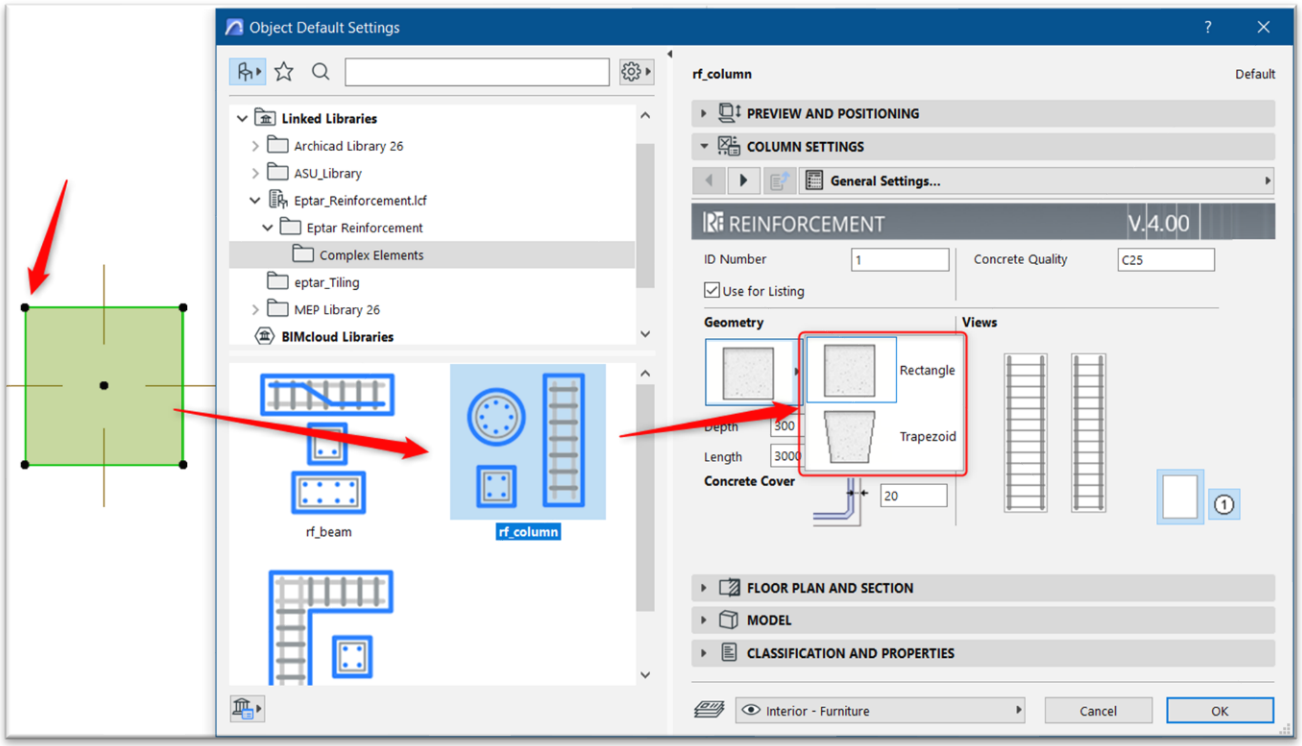
For beam reinforcement definition we suggest a similar window setting like this:



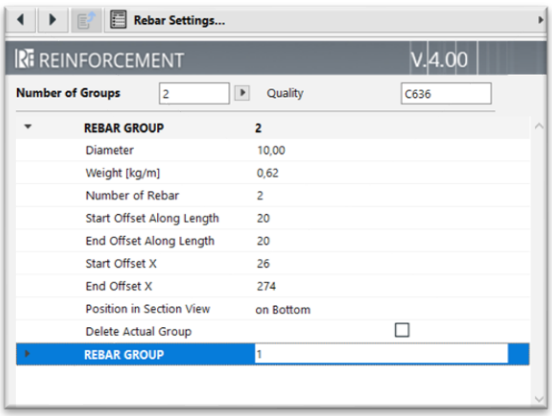
COLUMN DEFINITION

In the 4.0 version, the placement of reinforcements into columns is possible **automatically** (NEW) and **manually** (old).

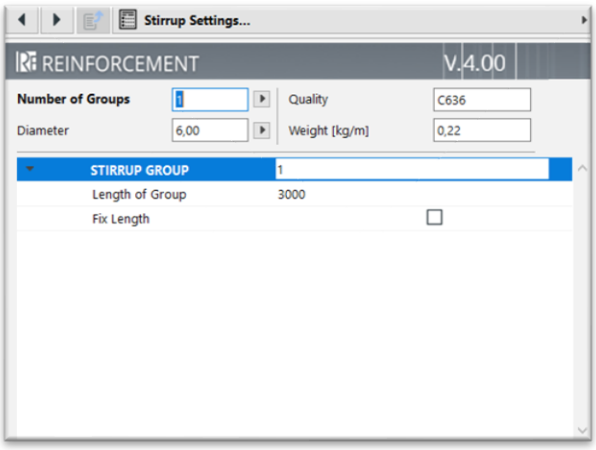
<u>New automatic reinforcement method:</u> the automatic method only supports the **simple rectangular** and **trapezoid** section profiles of columns.

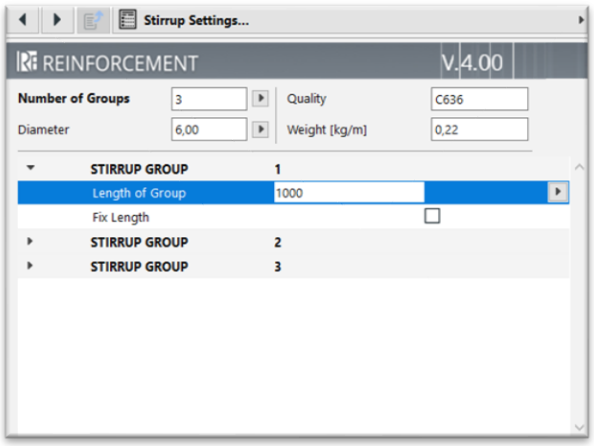


As is the case for all other elements, first the column should be selected. When the column object is selected, click on the "*column*" icon in the Reinforcement palette. In the previous version the first "column" icon was used to create complex elements, but now the settings window appears. The **rf_column** element will be preselected, and you can set all parameters for the current column. These setting are already well known, because the **rf_column** element is the same as was in previous versions. Set all the necessary parmeters, and click on OK, the addon will place the correct reinforcement in the column construction.
As in the earlier version of the **rf_column,** in this element 4 rebars are defined by default, but you can define more.



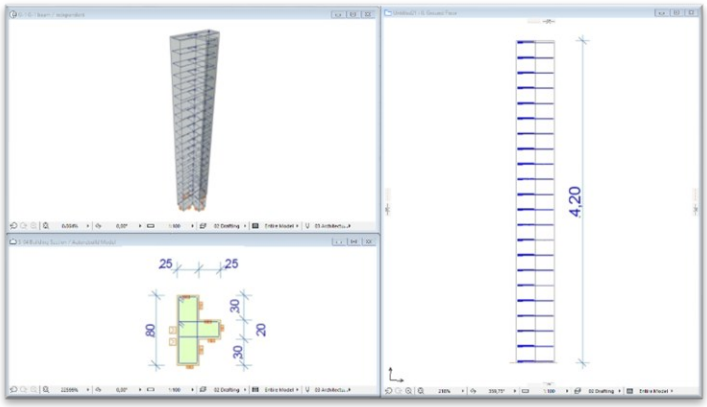
In case of stirrups only one group is defined, but of course many other groups are possible to create.



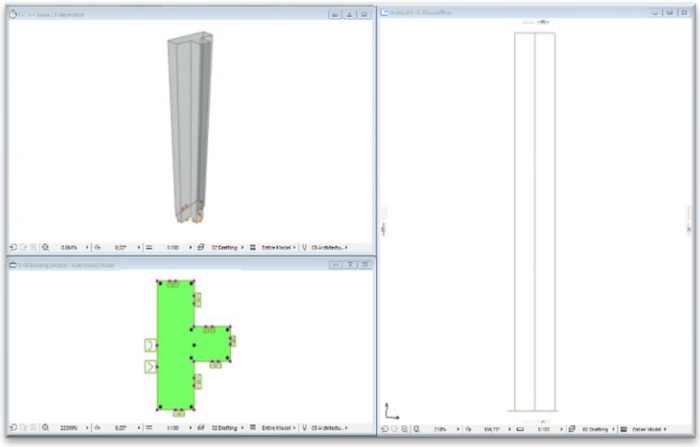


<u>Manual method:</u>
In our system one column contains at least one stirrup and two rebars. Fewer objects are not enough to define a column structure.

Place one or more „**rf_concrete**" object(s) on the Floor Plan somewhere in the horizontal range of the sections „**Vas-complex –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 („**Bar Distance**") parameter value.

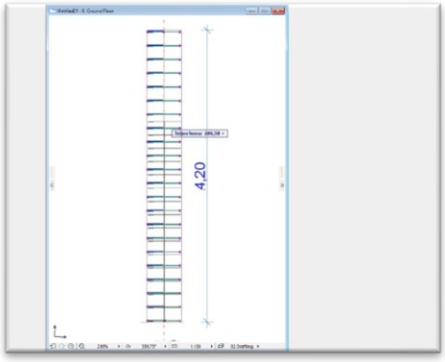


Select the „**rf_stirrup**” objects from the library. Open the „**Object Settings**” dialog and choose the „**Vertical**” placement option on the „**General**” tab page. Choose a shape for the stirrup and set up the parameters. Click the „OK” button and place the stirrup object inside of the „**rf_concrete**”.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



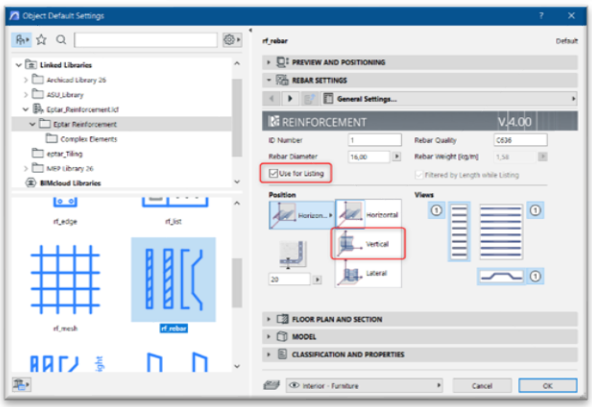
If you choose polygon style stirrup, please define the correct shape with the editable hotspots in the Floor Plan view, connect the polygon corners to the hotspots and hotlines of the „**rf_concrete**” object.
Click on the opened Section window, the stirrups will be visible immediately on the bottom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

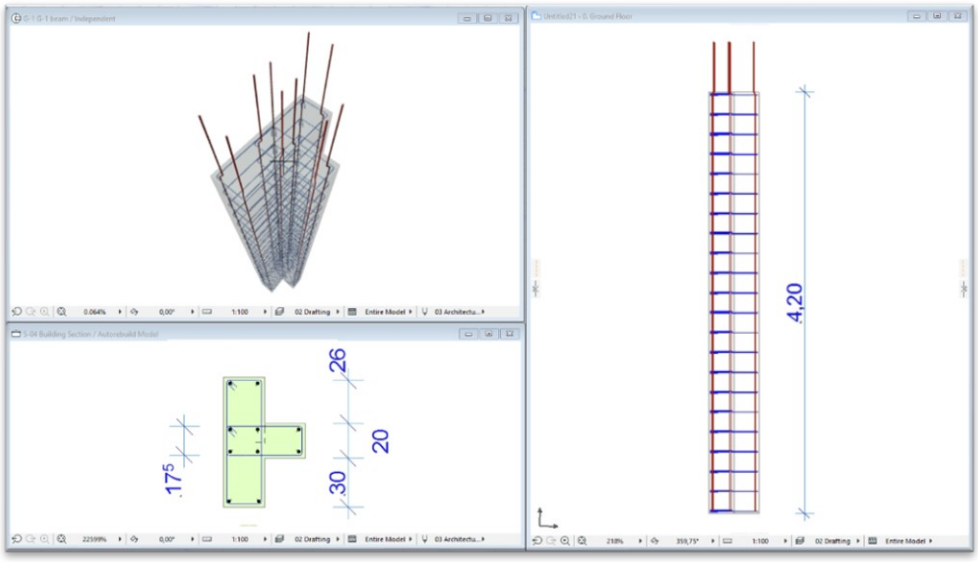
Use the editable hotspots to define the correct height of the stirrup group and change the stirrup's distance if it is necessary (for more information about stirrup placement see point 2.3).



Unfortunately, there is no possibility to copy or place 3D elements in the section window, if you need more stirrup groups in the column, go back to the Floor Plan, copy the existing stirrup object and click on the 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.

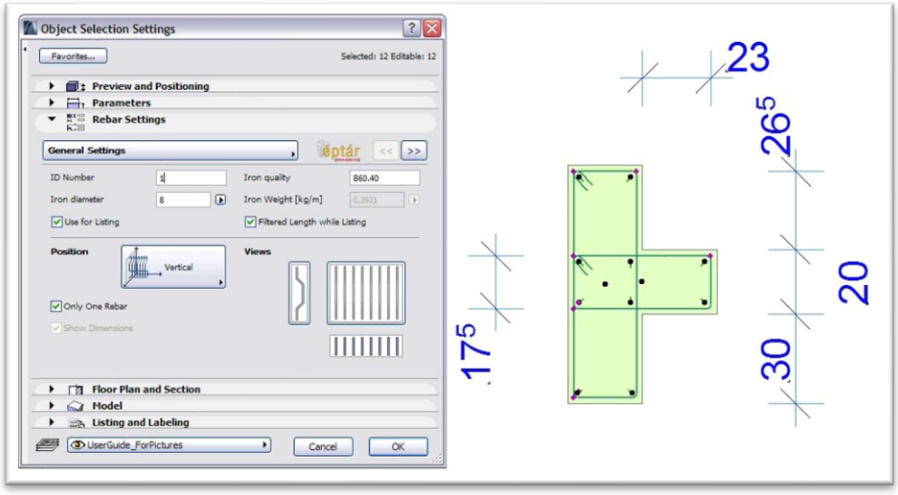
Select the „**rf_rebar**” object from the library. Select the „**Vertical**” placement option on the user interface and click the „**Only one rebar**” check box.
Set up the right style of the endings in the „**Geometry**” tab page and change the parameters. Click „**OK**”. Place the rebar object inside 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 as it is



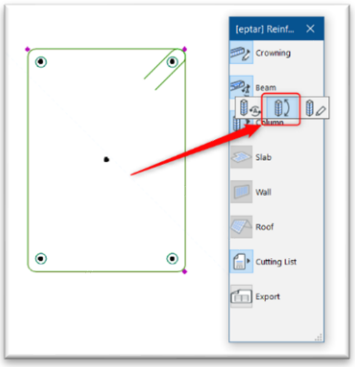


necessary.

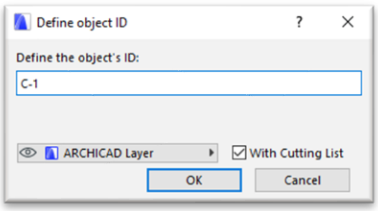
When you are ready with the column objects rebar structure, select all the rebar, stirrup, and concrete elements in the Column and make sure, the „**Use for Listing**” parameter is selected.



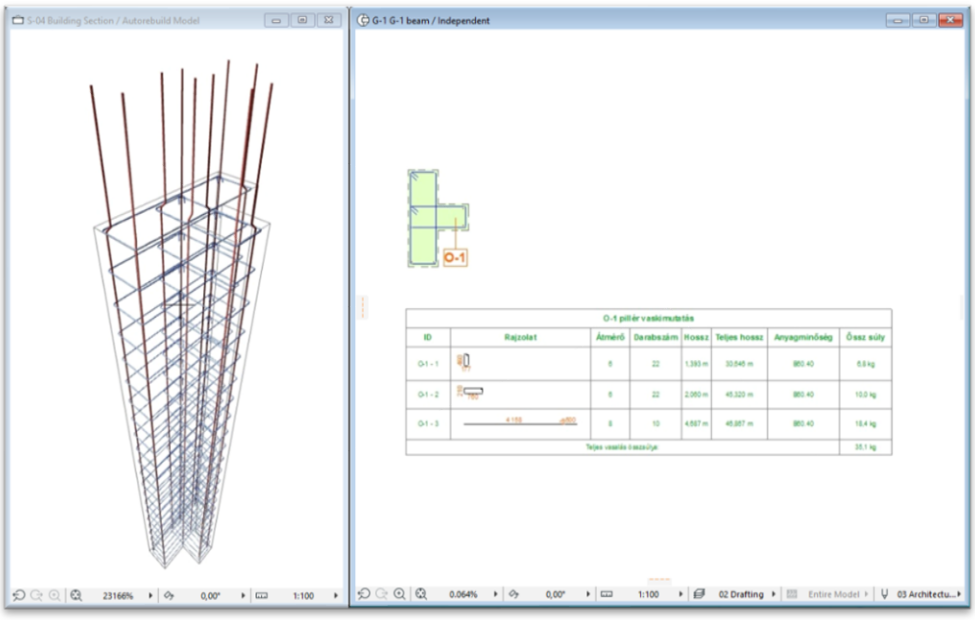
Select all the rebars, stirrups and concrete elements in your construction on the Floor Plan (section view is not supported for this command) and choose the „**Floating palette / Column / Create Column**” command (second icon).



In the opening dialog fill out the name (ID) of the new column (if you choose a name, which already exists, the application will ask you to choose another name). After the definition, you can place this column with this name to the model. The rebar calculation table will contain this name for the components (rebars, stirrups) of this column as well.
In the dialog, you can see a check box with a name „**With List**”. If you click this check box (default), you can place an individual rebar table about this construction on the Floor Plan (useful for detail drawings).
Click „OK”.



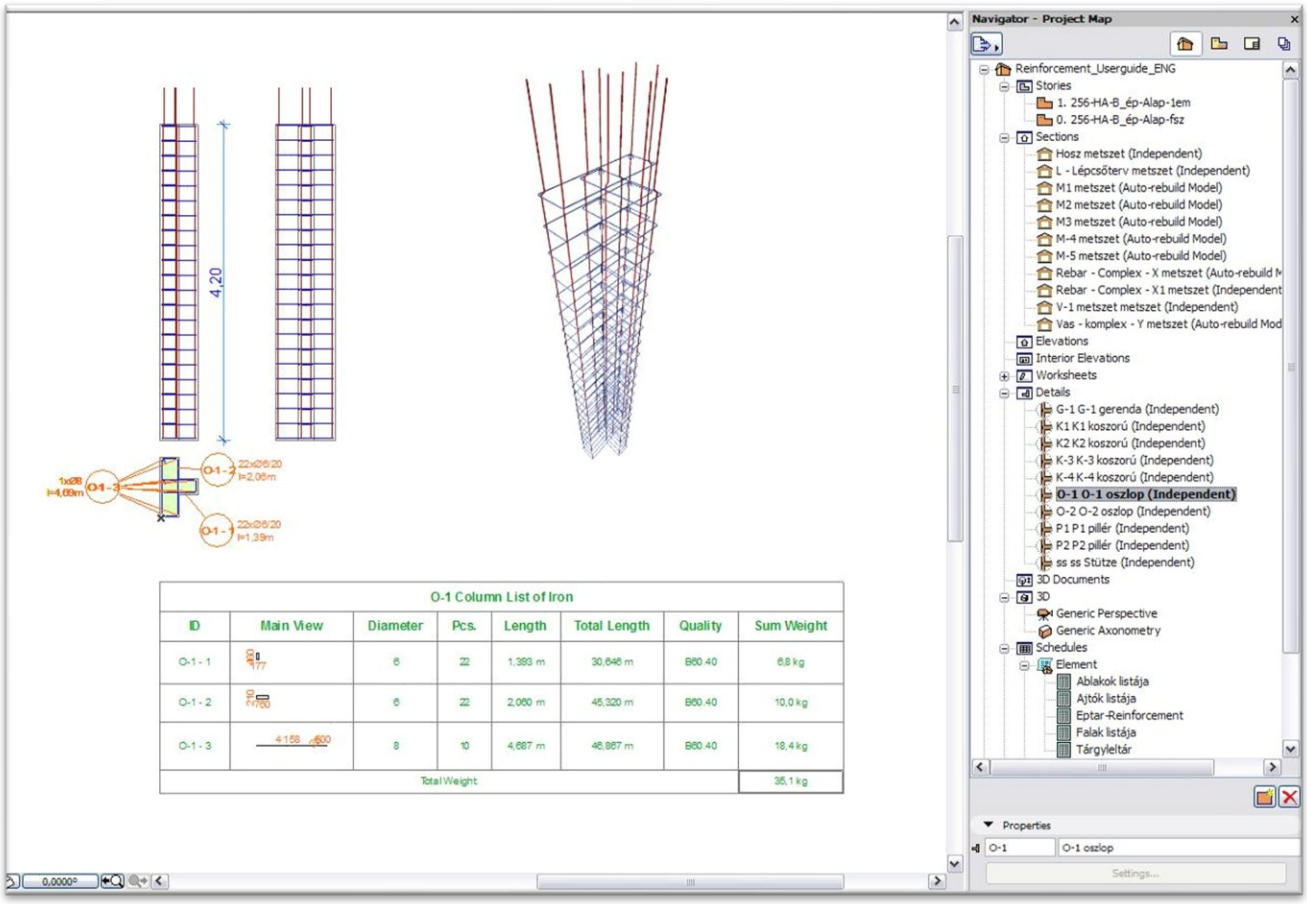
Place the column object on the Floor Plan, at least one copy of it (you can copy this column to multiply it on your plan. It is also suggested to save this object to the „**Favorites**”). If you requested a cutting list you also can place it into the Floor Plan.
In the column you created, all components’ (rebars, stirrups) ID will contain the column’s name and an individual, internal ID behind it. This time, the user can **not** change these names (e.g. if you used **O-1** as a name of the column, the rebars and stirrups number will look 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 the components of the columns, in addition 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 point 2.6).

We suggest 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 click 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.



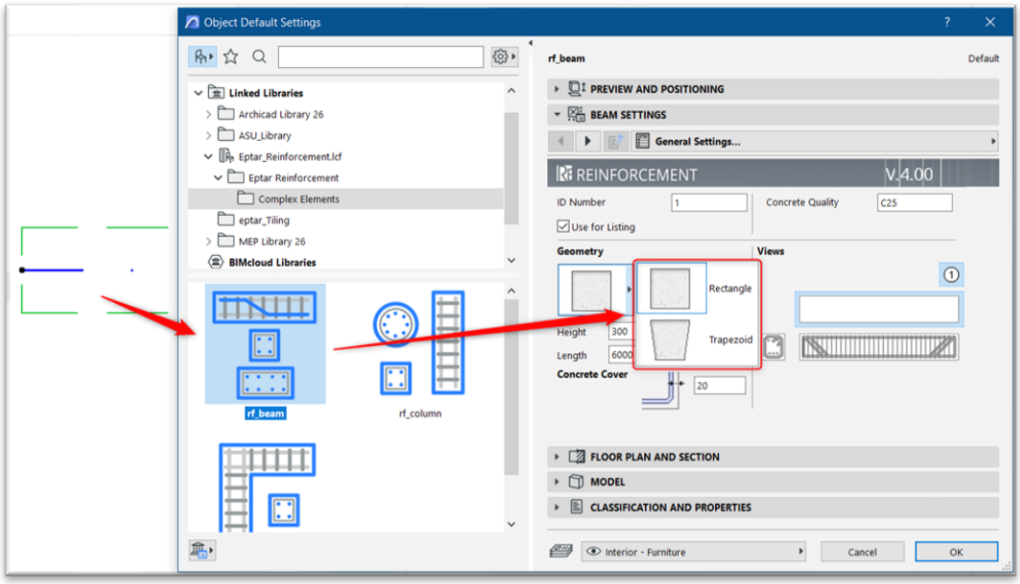
One of the possible detail drawings of columns:

We suggest 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**).

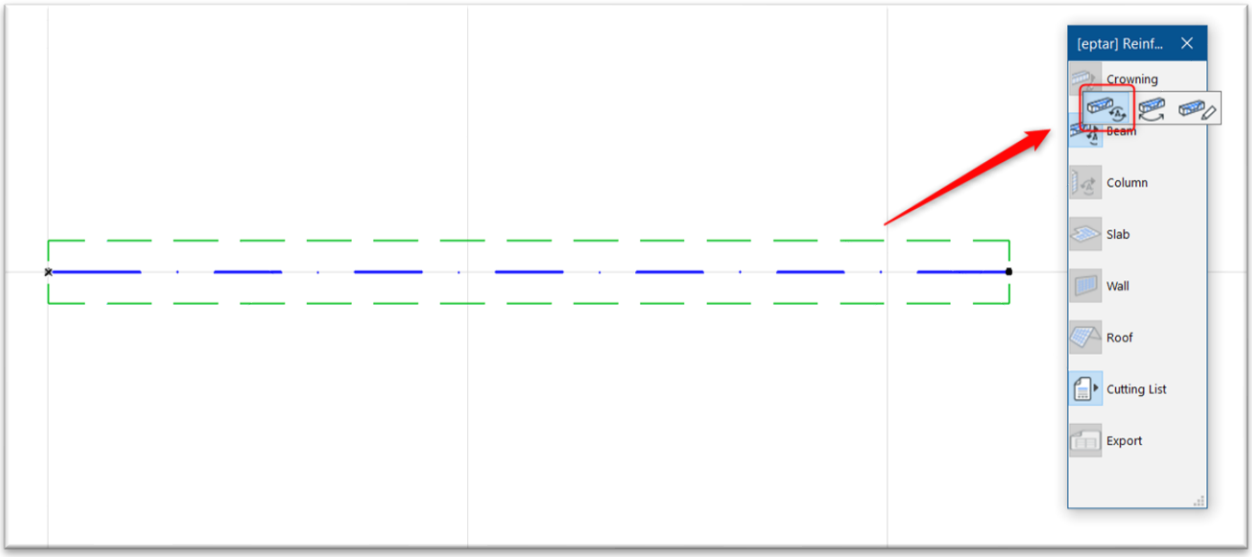
BEAM DEFINITION

In the 4.0 version, the placement of reinforcements into beams is possible **automatically** (NEW) and **manually** (old).

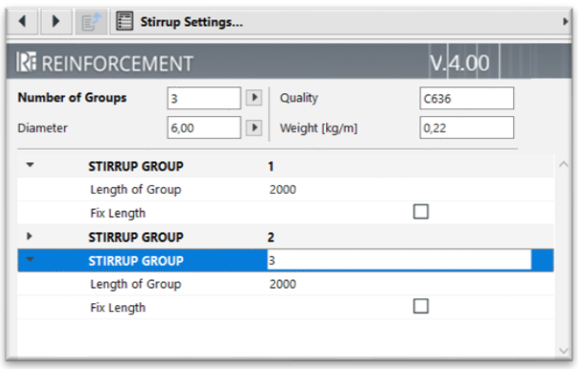
New automatic reinforcement method: the automatic method only supports the **simple rectangular** and **trapezoid** section profiles of beams.

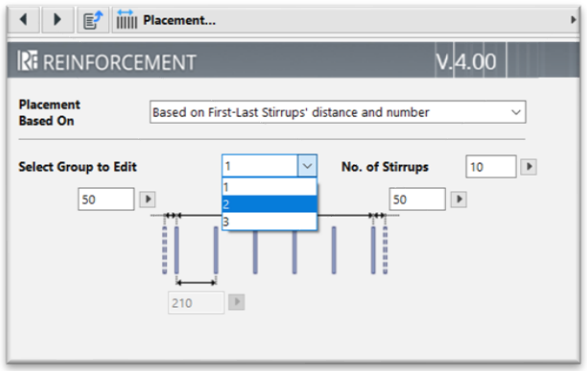


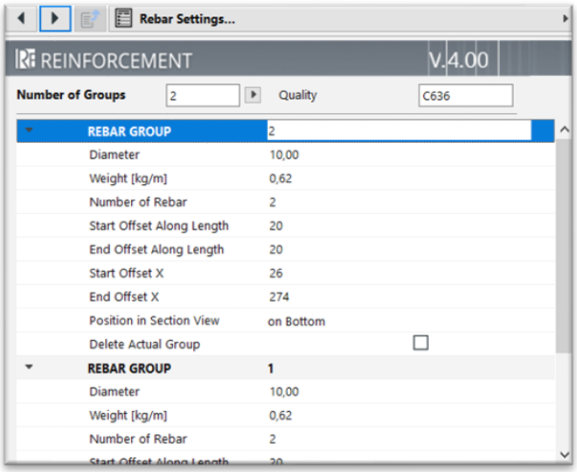
First the beam should be selected. When the beam object is selected, click on the "*beam*" icon in the Reinforcement palette. In previous versions the first "beam" icon was used to create complex elements, but now the settings window appears.



The **rf_beam** element will be preselected, and you can set all parameters for the current column. These setting are already well known, because the **rf_beam** element is the same as it was in previous versions. Set all necessary parameters, and click on OK, the addon will place the correct reinforcement in the column construction.
As in the earlier version of the **rf_beam,** in this element 4 rebars are defined by default, but you can define more.



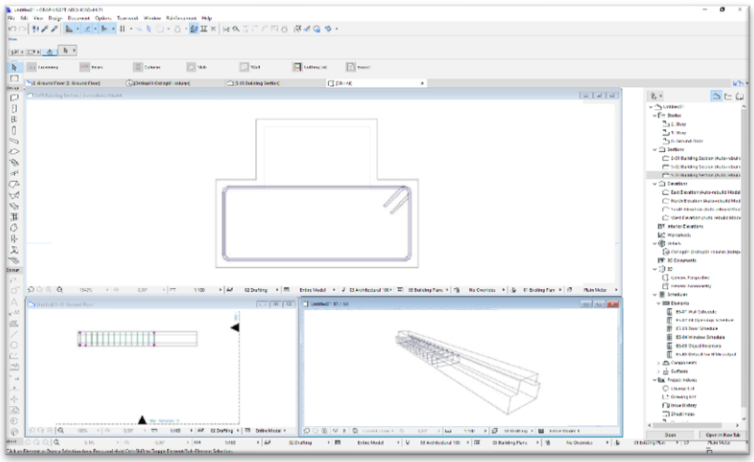




<u>Manual method:</u>
In our system, one column contains at least one stirrup and two rebars. Fewer objects are not enough to define a column structure.

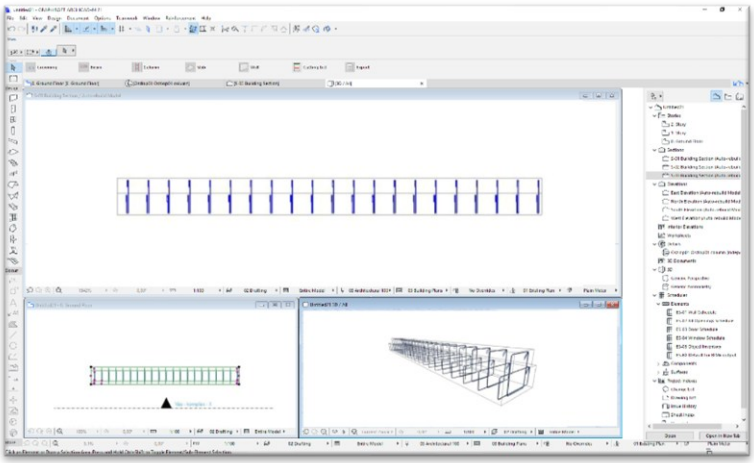
Place one or more „**rf_concrete**" objects on the Floor Plan somewhere in the horizontal range of the sections „**Vas-complex –X**" and „**Y**". Define the concrete external surface of the beam with these objects.
Select all the „**rf_concrete**" objects and set up the concrete cover („**Bar Distance**") 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 the „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, define the correct shape with the editable hotspots on the section view, then connect the polygon corners to the hotspots and hotlines of the „**rf_concrete**" object.
Click on the Floor Plan window.

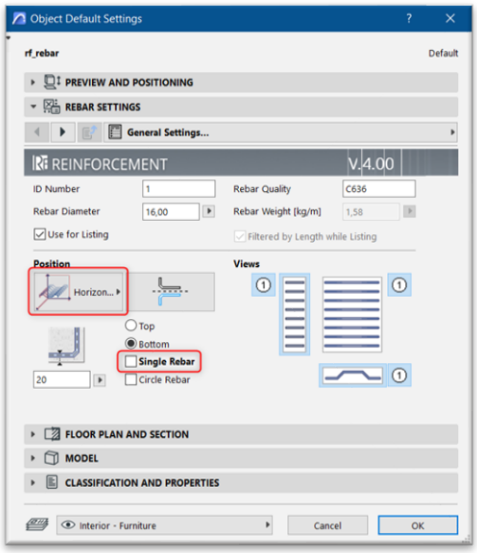


Use the editable hotspots to define the correct length of the stirrup group and change the stirrup's distance, if it is necessary (for more information about stirrup placement see point 2.3).
Place stirrup objects inside of the beam contour which can define the stirrup structure needed. Define the positions, shapes and stirrup distances with editable hotspots.

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

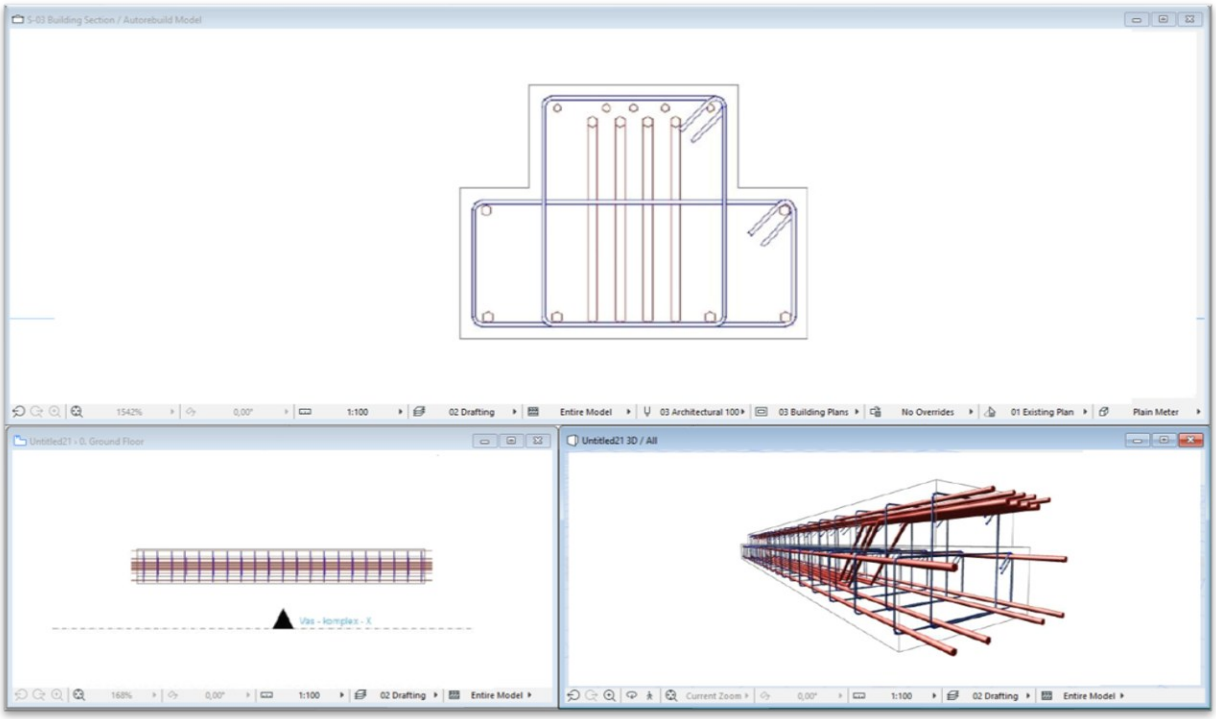


Select the „**rf_rebar**" object from the library. On the user interface select the „**Horizontal**" placement option and click the „**Only one rebar**" check box.
Set up the right style of the endings on the „**Geometry**" tab page and change the parameters. Click „**OK**". Place the rebar object inside the stirrup contour. Click on the Section window and move the rebar to the right position.



Unfortunately, it is not possible 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.

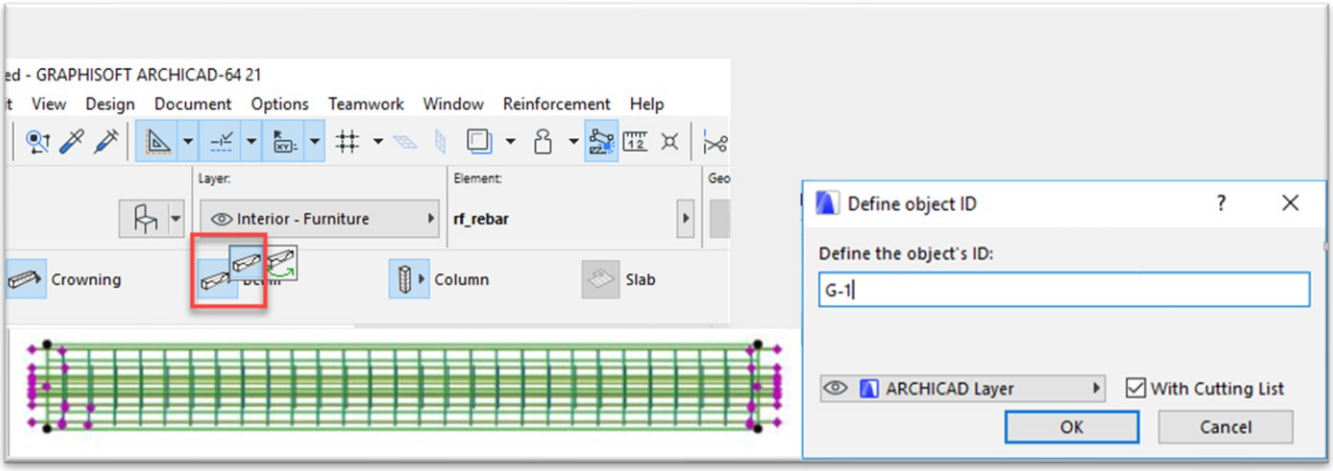
If you need rebars with different parameter sets for the beam, repeat the process above as many times as it is necessary.

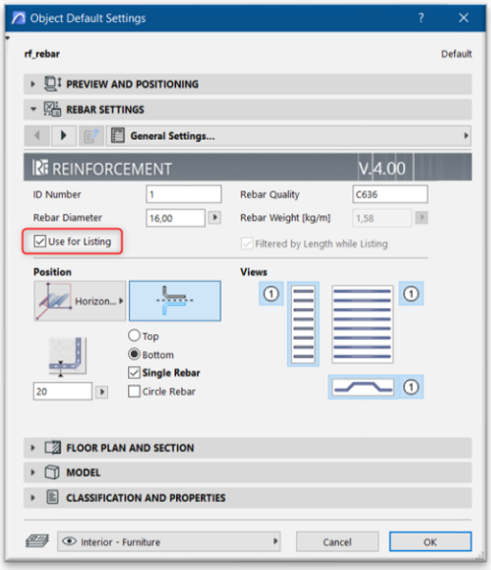


When you are finished with the beam objects rebar structure, select all the rebar, stirrup and concrete elements in the beam and make sure that the „**Use for Listing**" parameter is selected.

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 the „**Floating palette / Beam / Create Beam**" command.

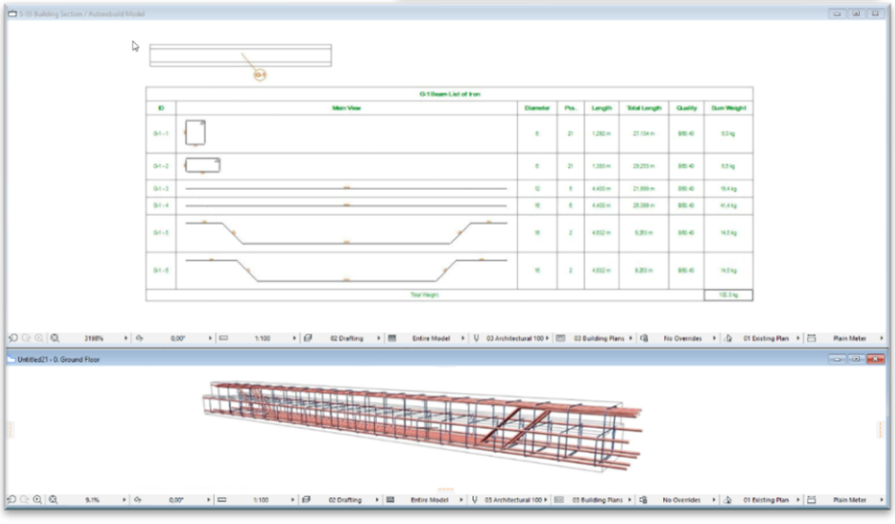
In the opening dialog fill out the name (ID) of the new beam (if you choose a name, which already exists, the solution will ask you to choose another name). After the definition you can place this beam with this name in the model. The rebar calculation table will contain this name as well for the components (rebars and stirrups) of this column.
In the dialog, you can see a check box with a name „**With List**". If you click 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 beam object on 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 can also place it on the Floor Plan.

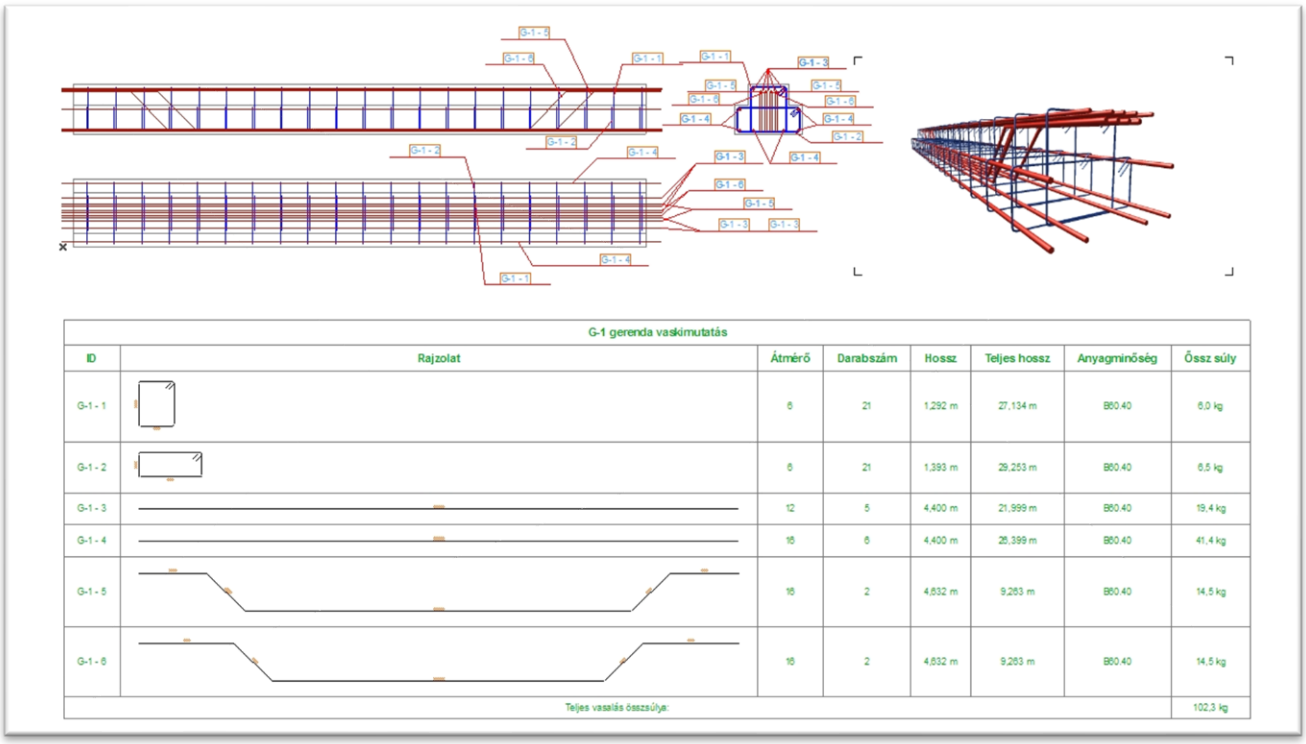
In the beam you created, all components' (rebars and stirrups) ID will contain the beam's name and an individual, internal ID behind it. This time the user can **not** change these names (e.g. if you used **G-1** as a name of the beam, the rebars and stirrups number will looks like: **G-1**-1**, G-1**-2**, G-1**-3**…**).
When you place the beam on 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. During the rest of the working process, you can find here the components of the beam, in addition, 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 (find more information about the beam changes in point 2.6).



We suggest 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 front 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 can also be set up in the „**Object Settings**” dialog.

One of the possible detail drawings of a beam:

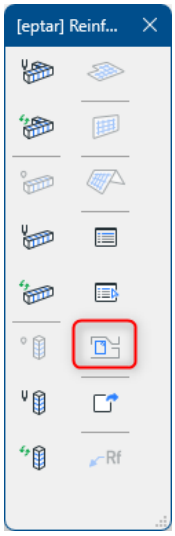


G-1 gerenda vaskimutatás

ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
G-1 - 1		8	21	1,292 m	27,134 m	B60.40	6,0 kg
G-1 - 2		8	21	1,393 m	29,253 m	B60.40	6,5 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 összsúlya:							102,3 kg

2.6 Component tool

Until now, it wasn't possible to draw the components of reinforcement in complex elements—like beams or columns—automatically. They could only be drawn manually. While these reinforcements appeared in the cutting list, they couldn't be extracted as individual drawings for documentation purposes.



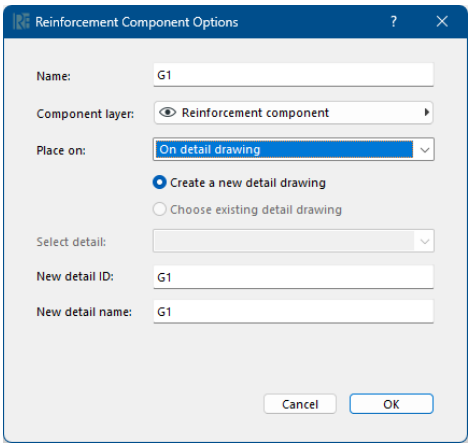
The newly released tool now allows the drawing and proper documentation of such complex reinforcement components.

The example structure consists of 3 columns connected by a beam. Let's take a look at it in 3D. The reinforcement is visible in all structural elements, which were placed as complex elements. Let's go back to the floor plan.

Previously, in the settings for placed reinforcement, you could enable views that showed, besides the cross-section, two side views as well. These two views could be repositioned, but documenting them was still cumbersome. The cross-section drawing couldn't be moved elsewhere because it was embedded in the structure.

Now, all of this can be addressed with the new tool.

The process is unchanged: select the complex element, then click the **Component Tool** icon on the Reinforcement palette. The program summarizes the reinforcement in the complex element and a dialog box appears.

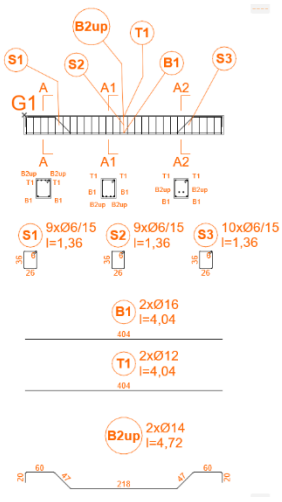


In this window, you can give a name to the drawing for later identification.

The drawing will get its own dedicated layer, which the program creates. Of course, you can choose a different layer if desired.

Next, you can choose whether to place the drawing on the current floor plan or on a detail drawing sheet. I'll choose the floor plan option now.

After clicking OK, a small list-like icon appears next to the cursor, similar to when creating a cutting list. Click on the drawing, and the reinforcement elements will be drawn individually. The name you specified will also be shown.



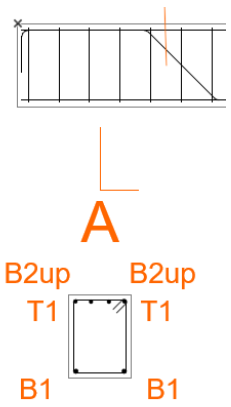
When the drawing is selected, you can see many hot spots on it. The drawing can, of course, be moved anywhere at any time.

Let's look at its components. In the view drawing, the labels appear. These labels identify all reinforcements in the column, such as main bars and stirrups. The reinforcement IDs appear with a prefix from the drawing name.

The labels can be repositioned, and for a cleaner visual appearance, the leader lines can be broken using the hot spots, so that the labels don't overlap or obscure each other.

The position of the cross-section is also shown in this view, and the nearby text and the drawing name label can also be moved as needed.

Moving on. The reinforcement drawings can be individually shifted along with their labels, and the labels themselves can be moved separately. If the drawing needs to be relocated, this can be done at any time.



Next feature: creating a cross-section. The cross-section's position is marked by a symbol that has an ID and can be moved in the view. Wherever it is placed, the cross-section drawing will show the reinforcement at that location. In the cross-section view, the main bars receive their IDs. These are paired and move together concentrically when the labels are moved. The labels include the drawing name as well.

The drawing name can be turned off in all labels. When selecting the drawing and opening its settings window, under the **Group Naming** tab, you can rename it at any time. Below that, label properties can be set—like font, size, and color.

Disabling the **Show Group Naming** toggle will hide the name from the drawing—but not from the labels. To remove the drawing name from labels too, go to the **Label** tab and uncheck **Show Whole Sign**. This removes the drawing name from all labels, which also reduces their size.

Opening the settings window again and switching to the next tab: here you can turn off the side view or the cross-section view.

Further down, you can set the number of cross-sections. This means that this number of section symbols will appear in the view, and that many cross-sections will be shown in the drawing. Entering for example 3, the list below expands. Each cross-section can have a unique ID. The program generates these automatically, but they can be customized.

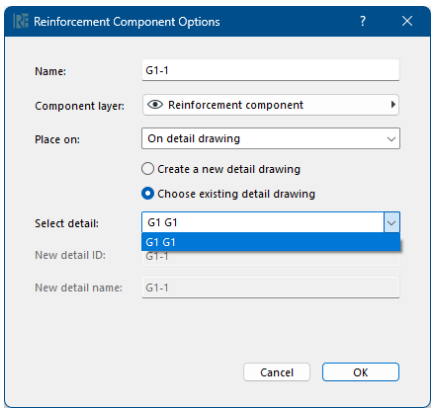
Clicking OK, three section markers and three cross-sections now appear in the view.

The section markers can be moved anywhere on the structure. Wherever the marker is placed, the cross-section view shows the reinforcement at that exact location. As the marker is moved, the cross-section view updates in real-time to reflect the changes.

By repositioning the labels, the full cross-section becomes visible, and when moving the section marker continuously, the section view updates almost instantly to show the current position. This lets you graphically verify that you're displaying the desired cross-section.

Now you can select the beam reinforcement and click the Component Tool. The settings window appears again. Enter the name for the beam drawing and choose to place it not on the floor plan, but on a new detail drawing. You can also assign an ID and name to the detail drawing.

The new detail drawing sheet opens, and click to place the drawing there. Returning to the Navigator and the project map, you can see that a new independent detail drawing has been created.



Back to the floor plan and selecting the other complex element (for example column), and click on the Component Tool again, the dialog come up again, you can provide a drawing name and keep the option to place it on a new detail drawing. Another independent detail drawing is created where I can place this drawing.

Back on the floor plan, you can select the next beam. Clicking the Component Tool opens the drawing dialog. This time, in addition to naming it, you can choose the **"Select existing detail drawing"** option. The list below shows the previously created detail drawings. Select the beam detail drawing and click OK. The selected detail drawing opens, where a reinforcement drawing already exists. Now place the new drawing next to the previous one.

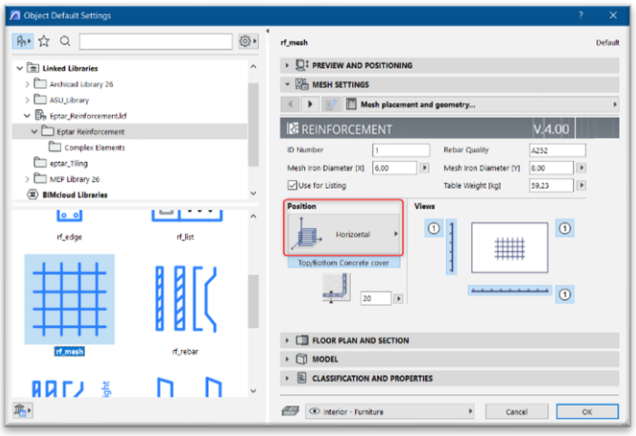
Returning again to the floor plan, and now select the beam reinforcement. Open the settings window and increase the stirrup spacing density, then click OK. The change is automatically reflected in the reinforcement drawing.

Select the other beam and adjust the stirrup group lengths in the settings. The change is immediately visible on the floor plan and automatically updates the associated reinforcement drawing.

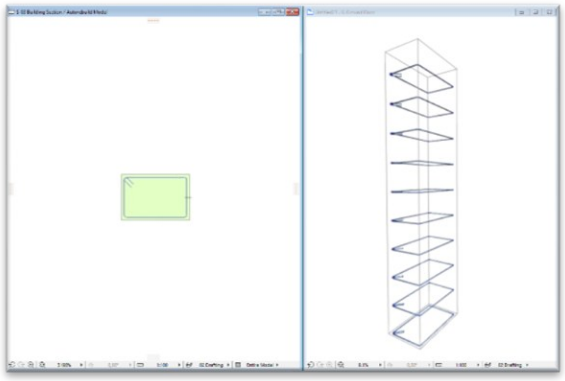
CROWNING DEFINITION

In Reinforcement one crowning contains stirrup systems and rebars. The crowning object is able to stretch on the Floor Plan instead of the beam element which is a fix size object.
The crowning system needs to contain at least one stirrup and two rebars.
Place one or more „**rf_concrete**" objects on the Floor Plan somewhere in the horizontal range of the sections „**Vas-complex –X**" and „**Y**". Define the external concrete surface of the crowning with these objects.
The crowning system should be built in **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 („**Bar Distance**") parameter value.

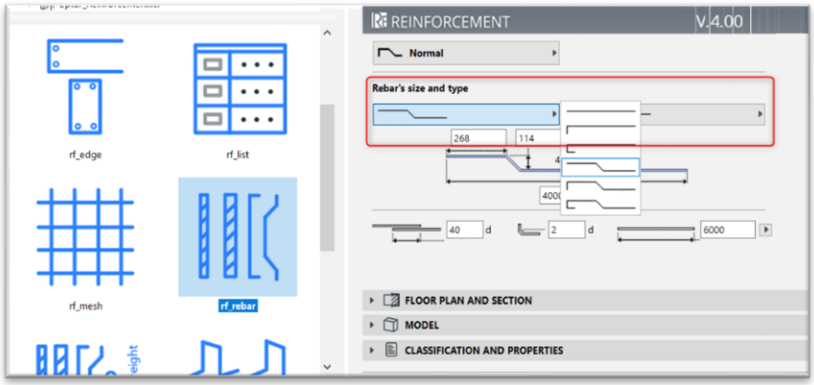
Select the „**rf_stirrup**" objects from the library. Open the „**Object Settings**" dialog and choose the „**Vertical**" placement option on the „**General**" tab page. Choose a shape for the stirrup and set up the parameters. Click the „OK" button and place the stirrup object inside of the „**rf_concrete**".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

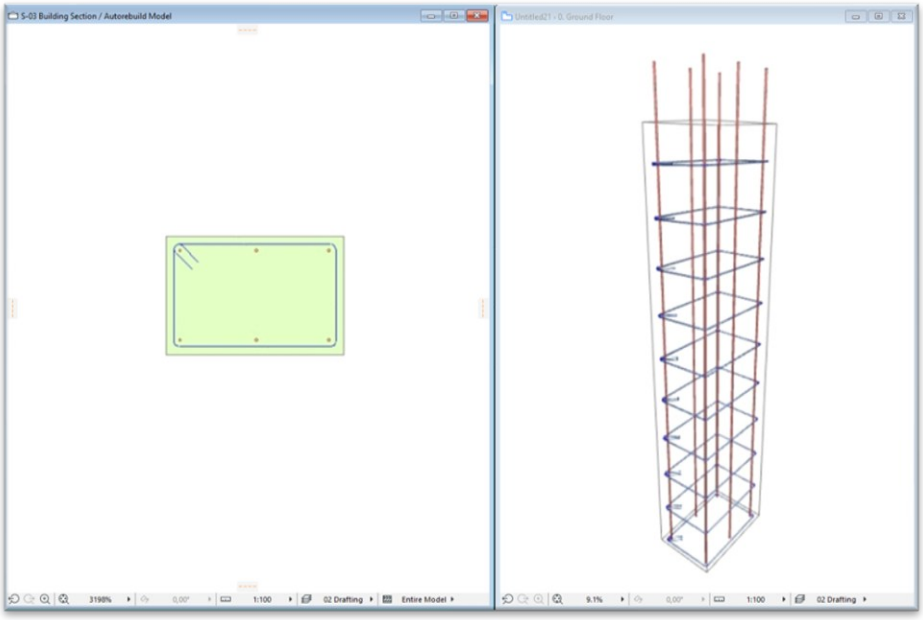


Select the „**rf_rebar**" object from the library. On the User Interface select the „**Vertical**" placement option and switch on the „**Only one rebar**" check box.
Set up the „**Straight**" style of the endings in the „**Geometry**" tab page. Click the „**OK**" button. Place the rebar object inside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

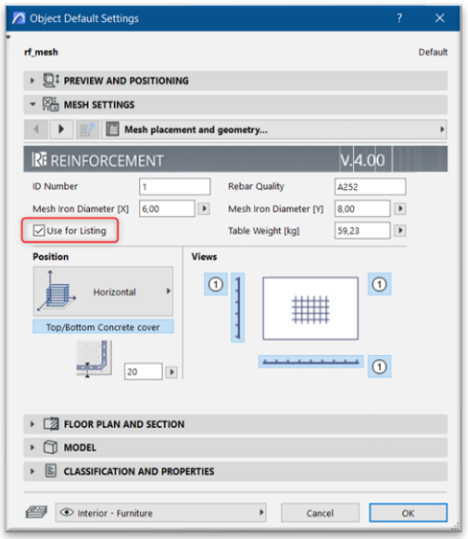


If you need more rebars in the crowning, go back to the Floor Plan view and multiply your rebar object on the Floor Plan to the right positions.

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

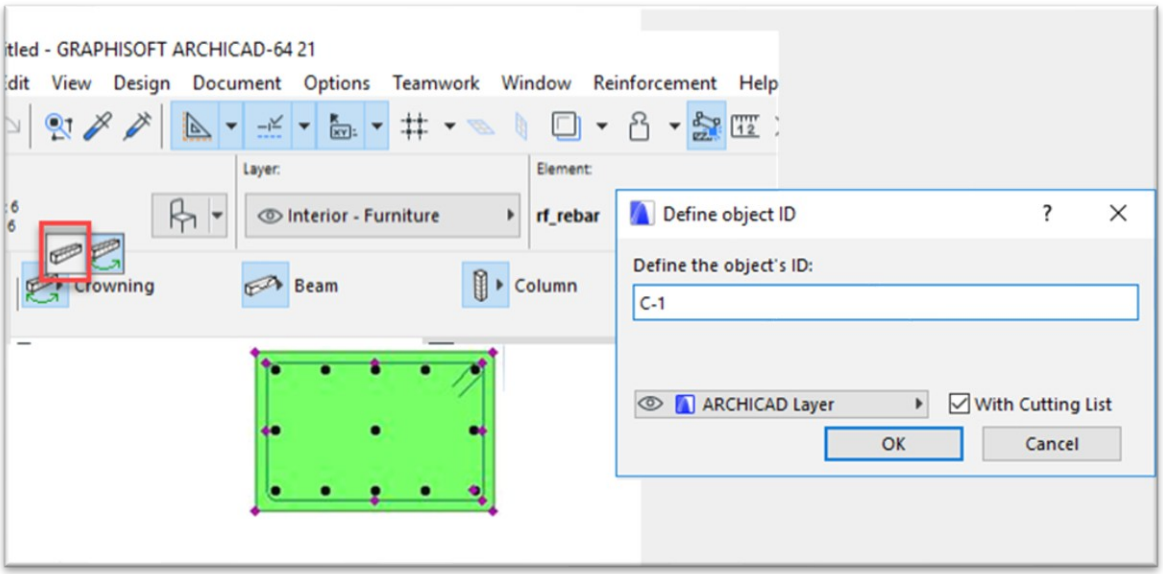


When you are finished with the beam objects rebar structure, select all the rebar, stirrup abd concrete elements in the crowning and make sure that the „**Use for Listing**" parameter is selected.

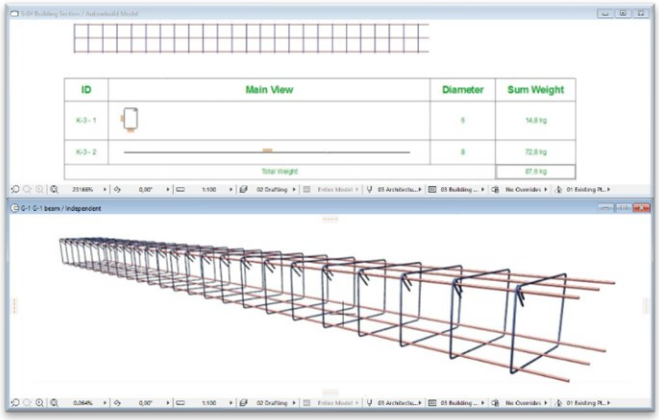


Select all rebars, stirrups and concrete elements in your construction on the Floor Plan (section view is not supported for this command) and choose the „**Floating palette / Crowning / Create Crowning**" command.

In the opening dialog fill out the name (ID) of the new crowning (if you choose a name, which already exists, the application will ask you to choose another name). After the definition you can place this crowning with this name to the model. The rebar calculation table will contain this name for the components (rebars and stirrups) of this crowning.
In the dialog, you can see a check box with a name „**With Cutting List**". If you click 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 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.



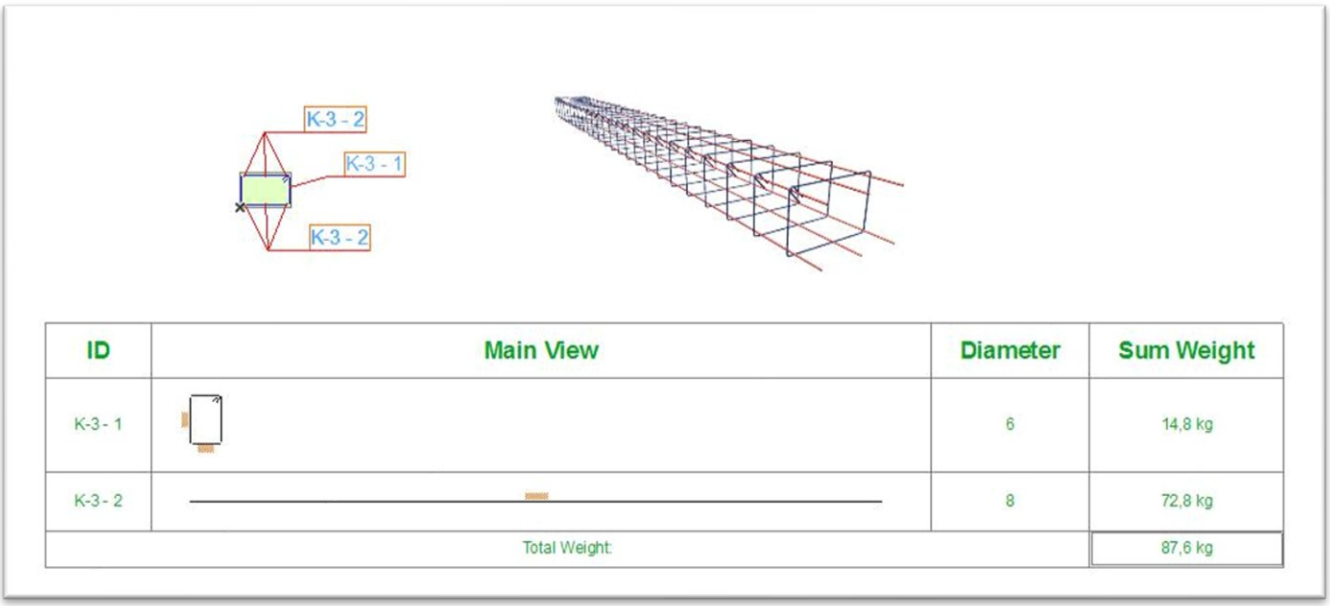
In the crowning you created, all the components' (rebars and stirrups) ID will contain the crowning name and an individual, internal ID behind it. This time the user can **not** change these names (e.g. if you used **K-1** as a name of the crowning, the rebars and stirrups number will looks like: **K-1**-1**, K-1**-2**, K-1**-3**…**).

When you place the crowning on 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. During the rest of the working process you can find here the components of the crowning, in addition, 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 (find more information about the crowning changes in point 2.6).

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 overhang.
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

2.7 Edit Objects

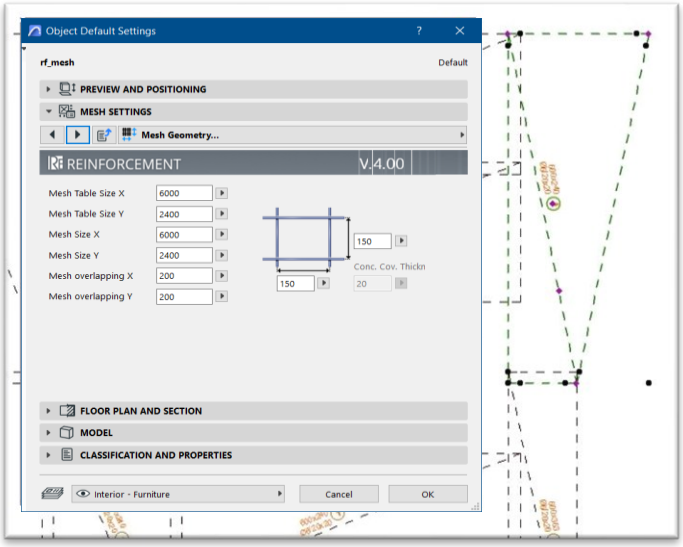
EDITING METHODES IN GENERAL

Changes of any kind of the objects on the 3D model do not have influence on the already placed cutting lists. If you edit any objects on the model or change some parameters on it, it is recommended to refresh the cutting lists with the command „**Floating palette / Cutting lists placement**".

During the refreshing 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.

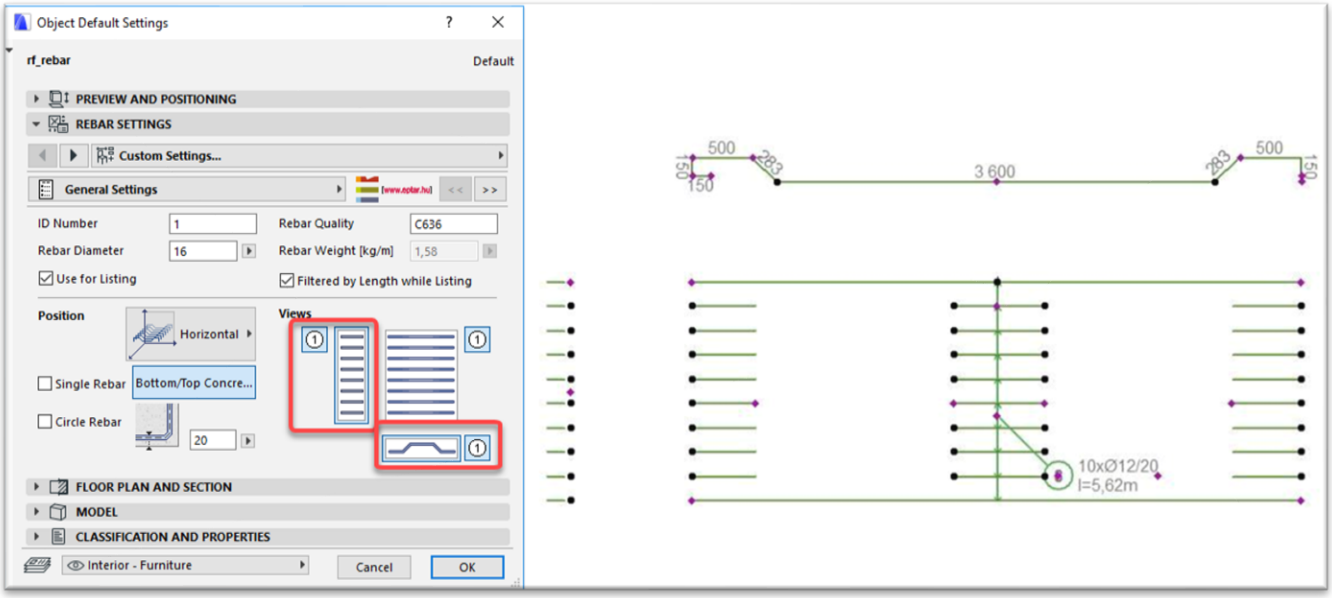
EDIT BASIC OBJECTS

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, open the „**Object Settings**" dialog and change the requested values. Click the „OK" button to apply the changes on the object.

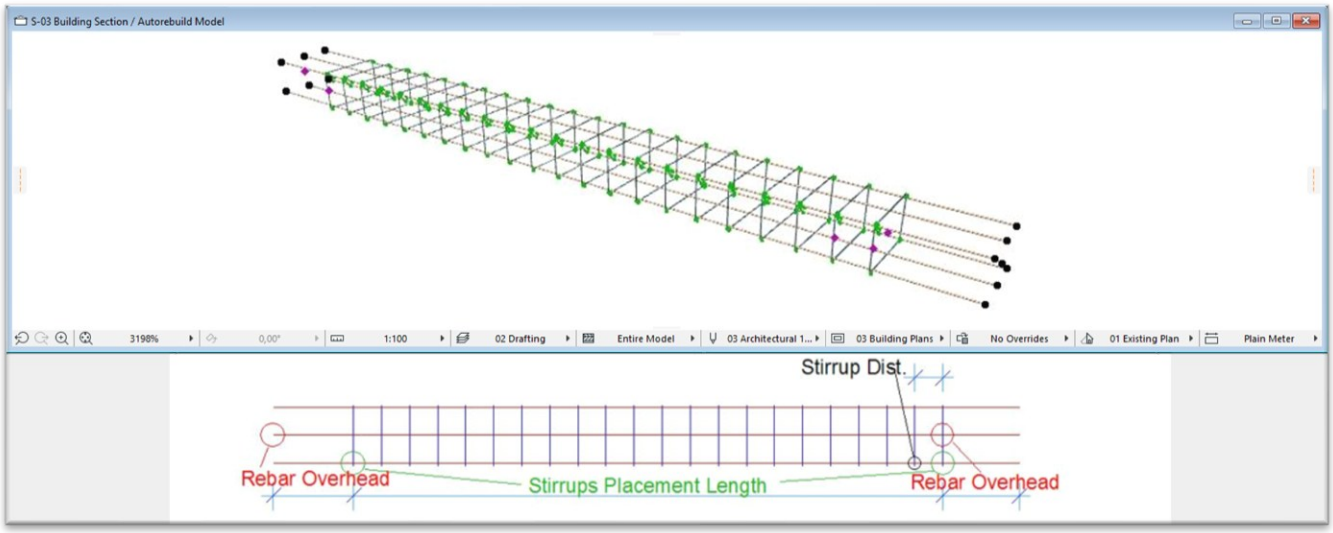
To edit rebar and stirrup objects, it would be helpful if the side view (section view) of the element could be seen.To see these open the „**Object Settings**" dialog, select the „**General Settings**" tab page and switch on the requested views with the icons on the bottom. Click „OK" and the external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



EDIT COMPLEX ELEMENTS

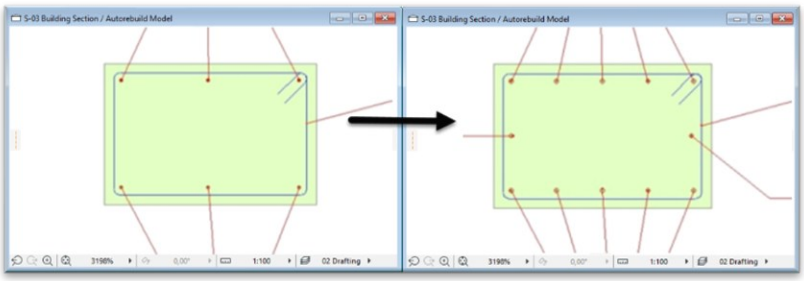
Edit Crowning object

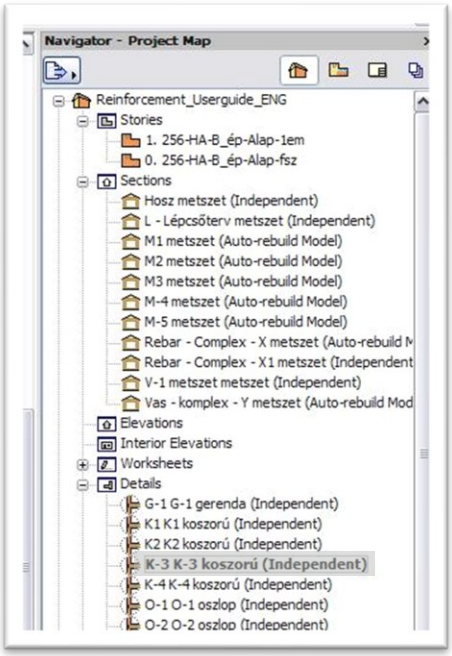
You can edit the crowning object on the Floor Plan and the 3D view with the editable hotspots. You can change the full length of the construction, the distance of the stirrups and the rebars overhang from the last stirrup element (necessary for corners).



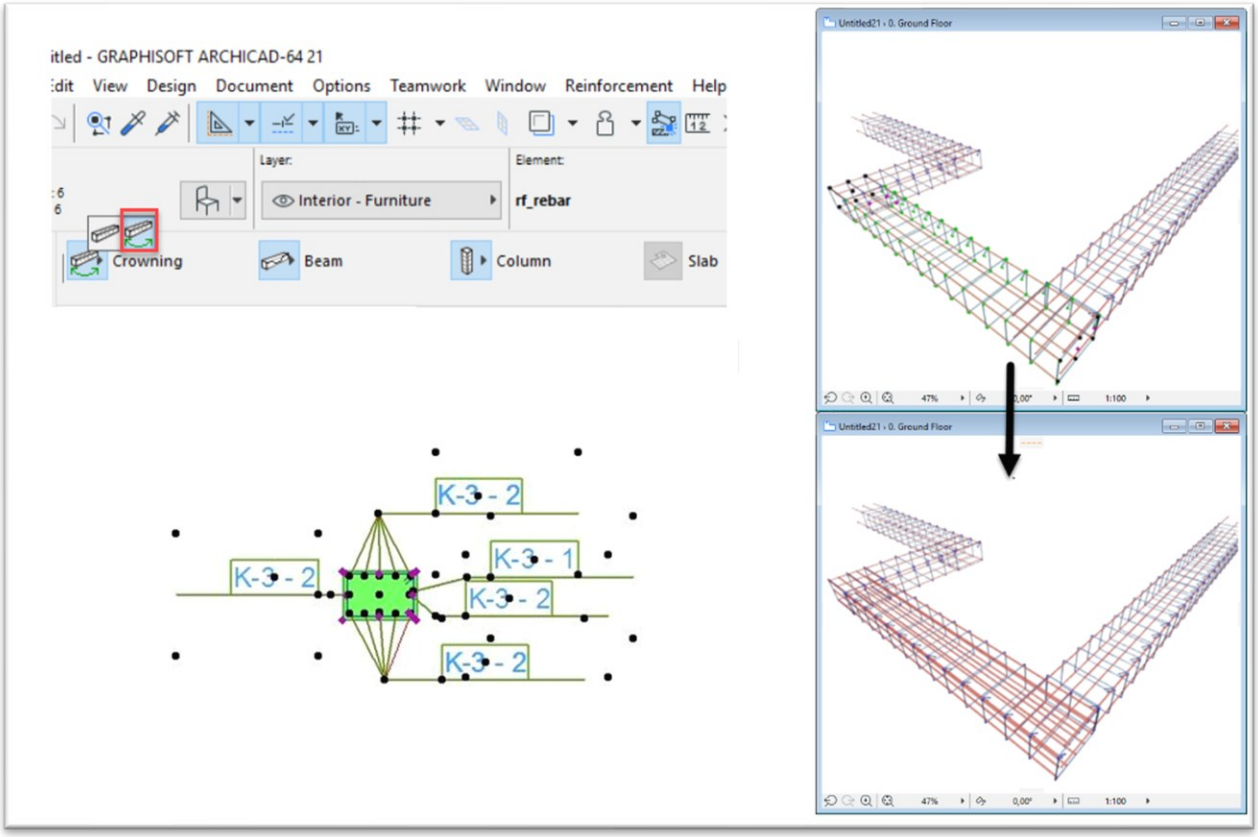
You can check the components of the crowning objects in the „**Objects Settings**" dialog under the „**General Settings**" tab but these components are not available to change in this view, you can edit the components only with the help of the command „**Floating palette / Crowning / Edit Crowning**".

Select the „**Detail Window**" with the name of the crowning to edit it (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.

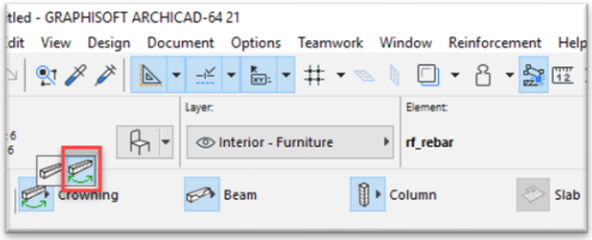




If you don't want to change a few types of elements on the model, select the objects you want to change and select the edited crowning components as well. Click on the „**Floating palette / Crowning / Edit Crowning**" command.
The solution will refresh the selected crowning objects with the selected components (basic elements). Now you created a new crowning structure but there is some untouched crowning on the model, which is not affected, so you can not overwrite the original crowning number, you have to define a new name.

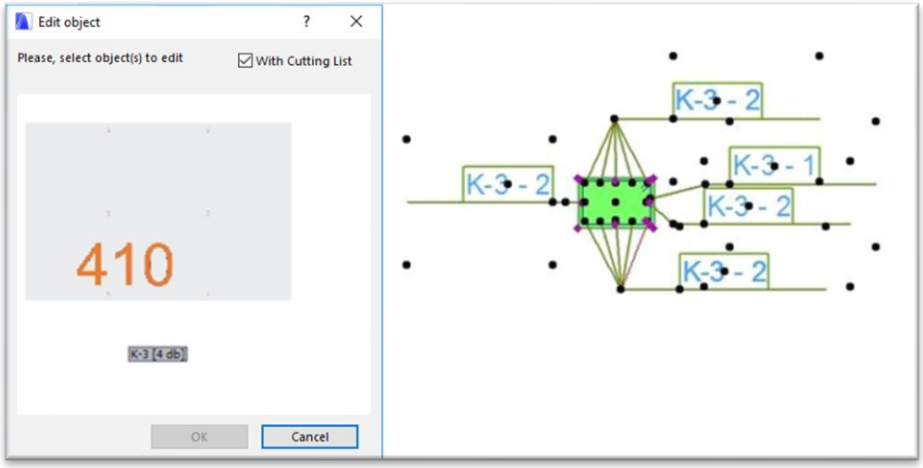


If you want to change all the crownings in the model, select only the crowning components on the Floor Plan.
Click on the „**Floating palette / Crowning / Edit Crowning**" command. The solution opens a dialog that shows the available crowning composites on the model. You have to select the number of the crowning structure, which you want to refresh. Click „OK" and the solution will update all the crowning objects (structures) on the Floor Plan that have the same number which you have selected. The solution also updates the detail windows as well.



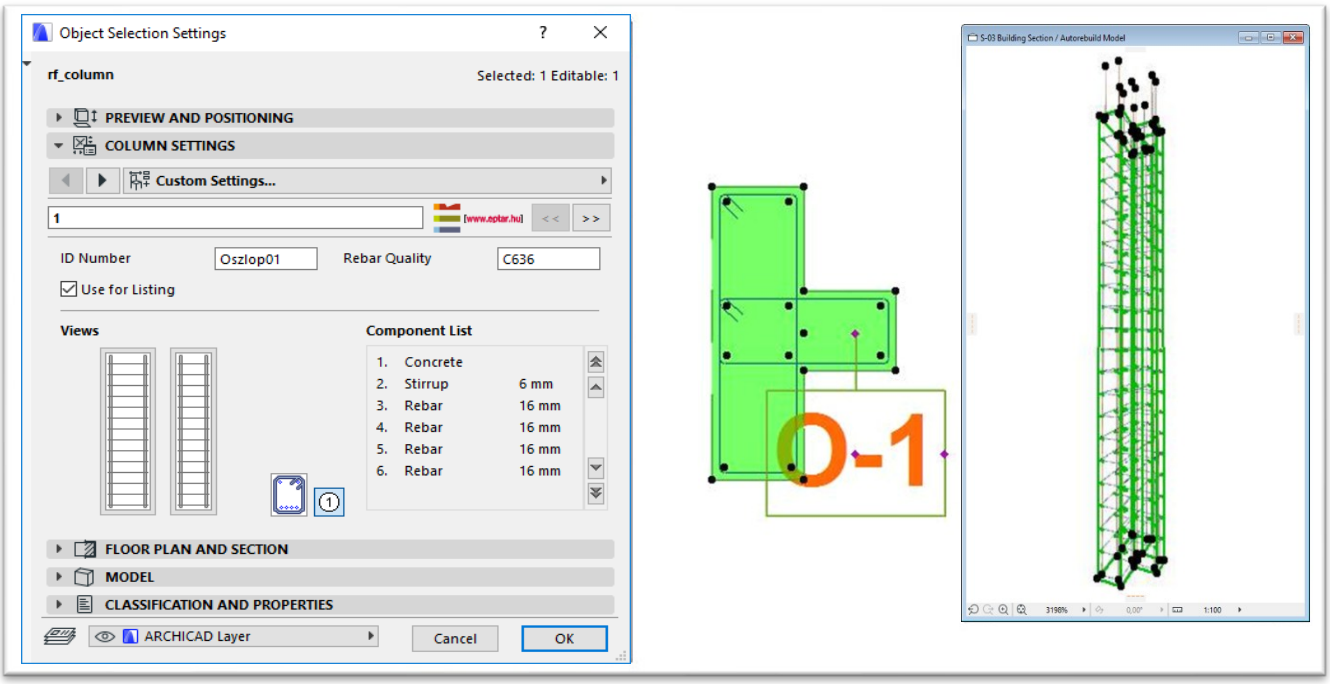
The application does not refresh side views in the „Detail drawing" window, so it is recommended to delete these views and tables and to place them again.

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

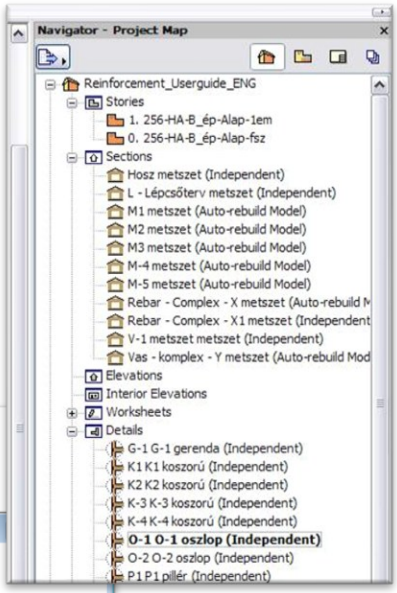


Edit Column object

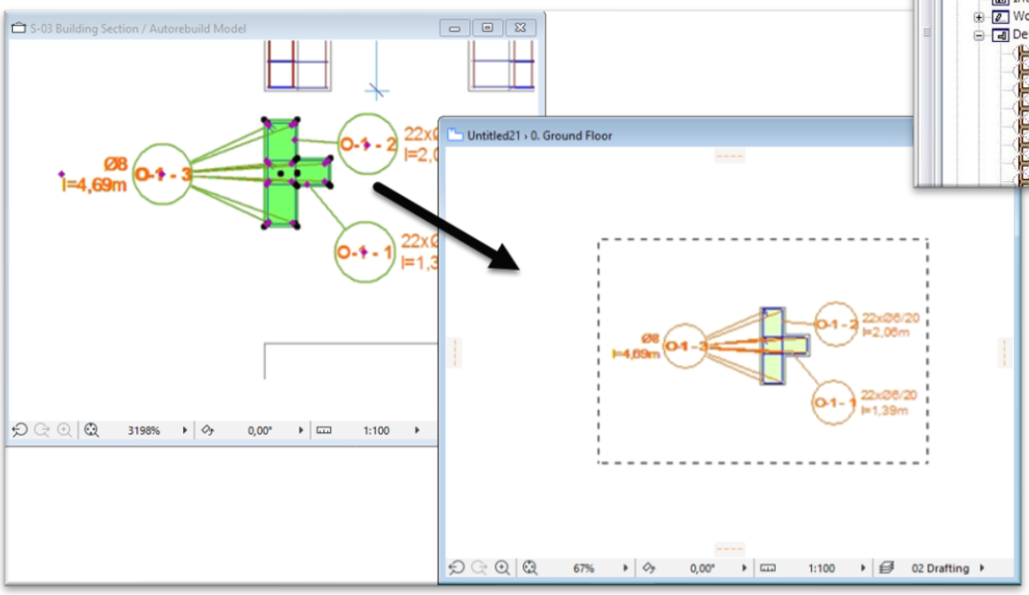
Column elements are not possible to edit on the Floor Plan or in 3D model view. In the „**Objects Settings**" dialog you can change only the pen colors, listed or not listed check box, ID name and some 2D representation mode.



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 „**Floating palette / Crowning / Edit Crowning**".

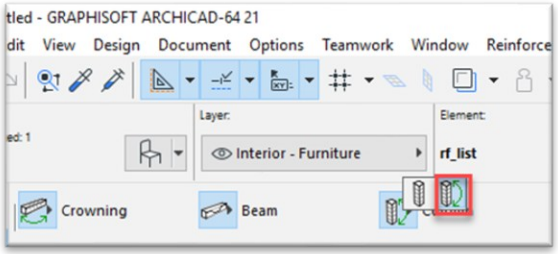


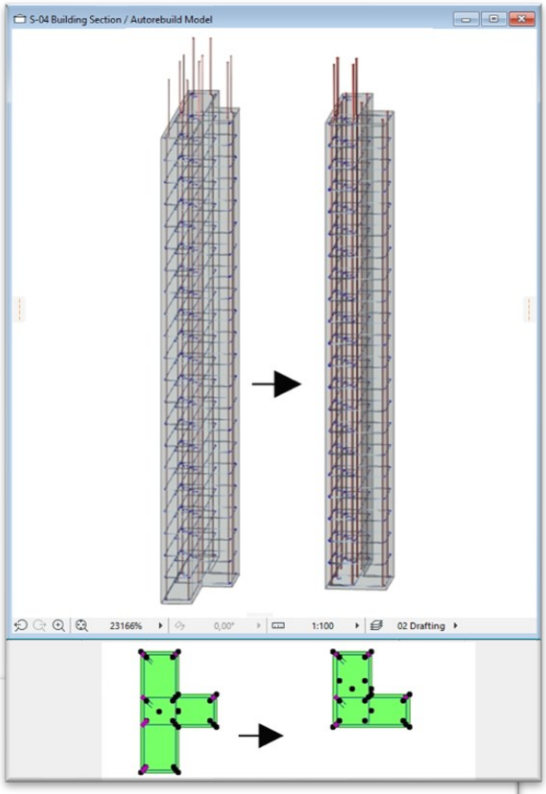
Select the „**Detail Window**" with the name of the column to edit (e.g. O-1).
Copy the section view components from the detail window to the Floor Plan.

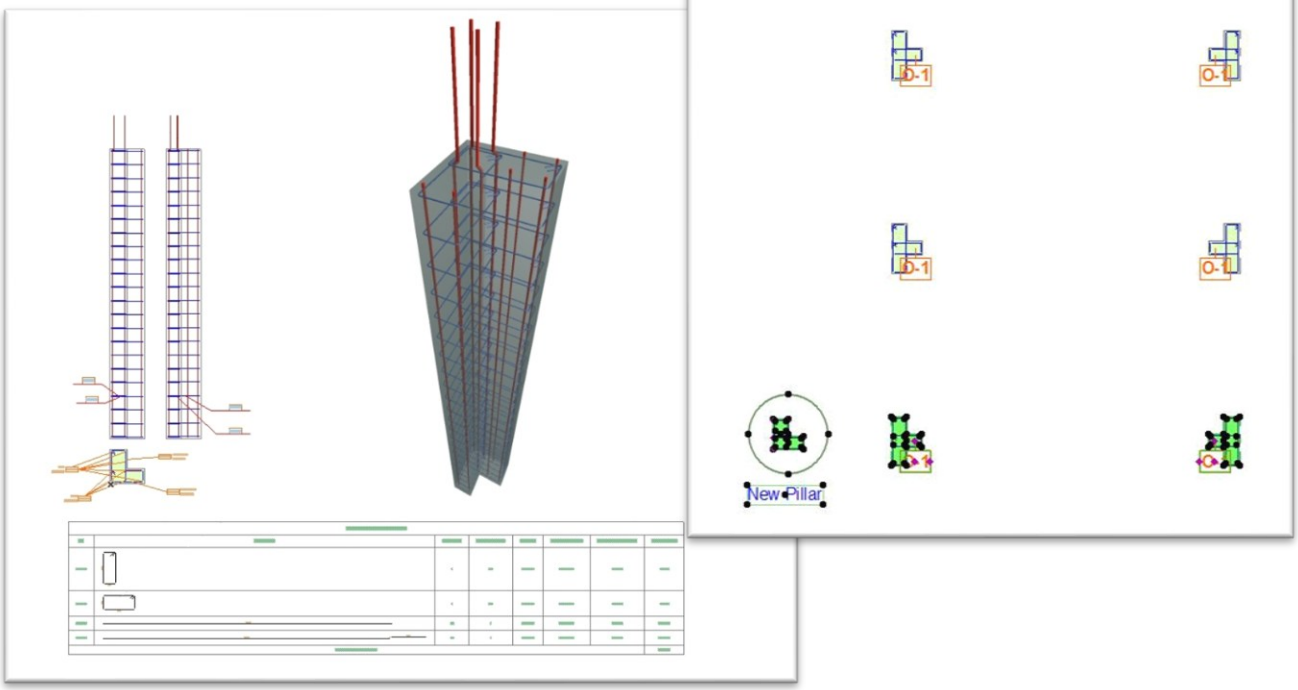


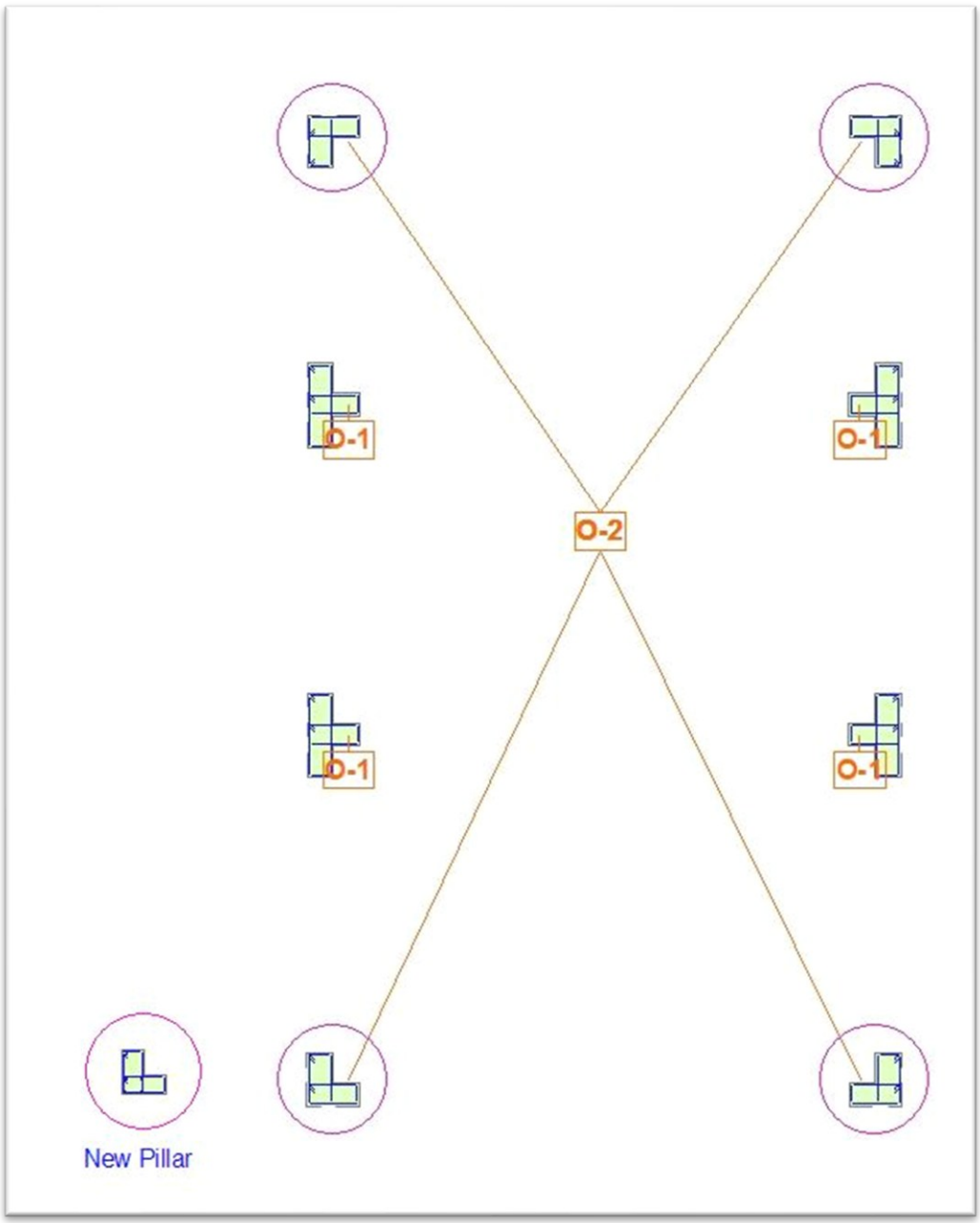
Change the necessary parameters, delete components or add new ones into the construction.

If you want to change a few pieces of elements on the model, select the objects you want to change and select the edited column components too. Click on the „**Floating palette / 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 but you have to define a new name.







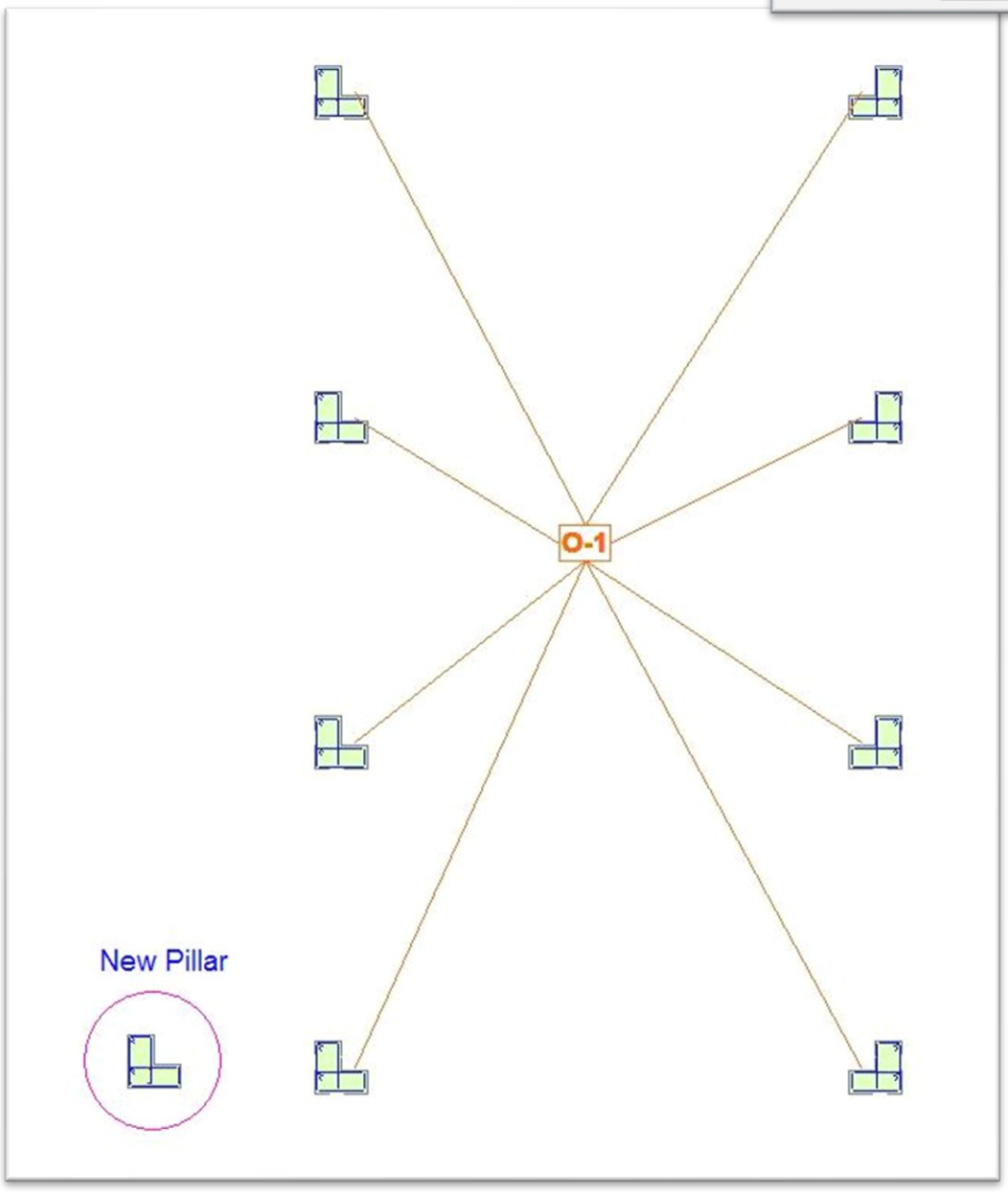

O-1
O-1
O-2
O-1
O-1
New Pillar

If you want to change all the columns in the model, select only the column components on the Floor Plan.
Click on the „**Floating palette / Column / Edit Column**” command. The 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” and the solution will update all the column objects (structures) on the Floor Plan which have the same number that you selected. The solution also updates the detail windows.



The application does not refresh side views in the „Detail drawing” window, it is recommended to delete these views and tables and to place them again.

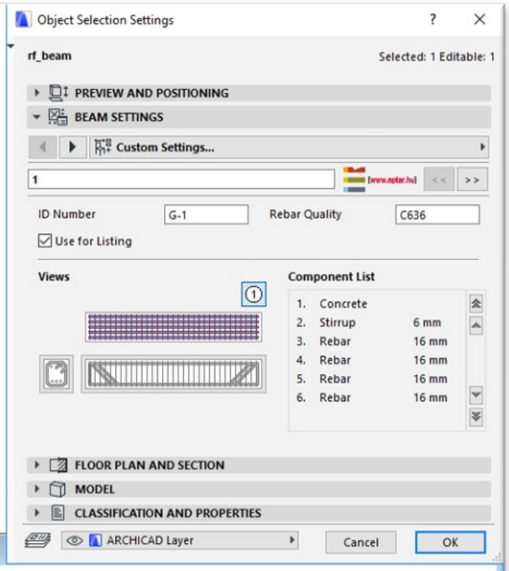
When you update the columns, the cutting lists which contains these types of columns will not be updated automatically. We suggest to update the cutting lists after these types of changes.



Edit Beam objects

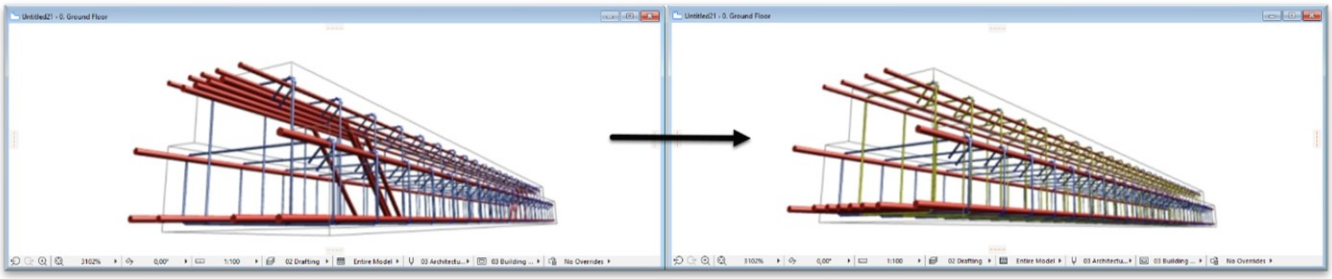
It is not possible to edit beam elements on the floor plan or in 3D model view. In the „**Objects Settings**” dialog you can change only the pen colors, listed or not listed check box, ID name and some 2D representation modes.
You can check the components of the beam objects in the „**Objects Settings**” 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 „**Floating palette / Beam / Edit Beam**” command.



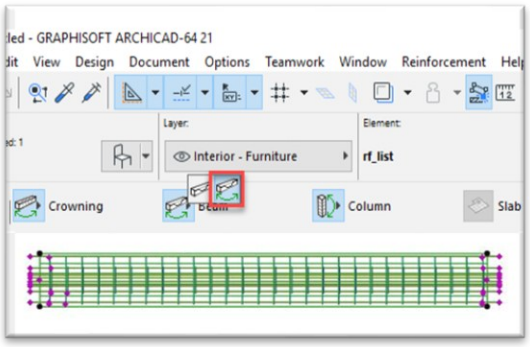




Select the „**Detail Window**” with the name of the column to edit (e.g. G-1). Copy the section view components from the detail window to the Floor Plan.
Change the necessary parameters, delete components or add new ones into the construction.



If you want to change only a few pieces of elements on the model, select the objects you want to change and select the edited beam components as well. Click on the „**Floating palette / 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 but you have to define a new name.

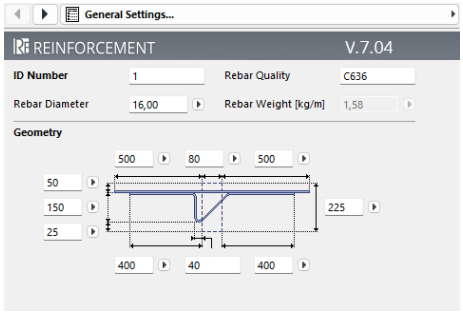


If you want to change all the beams in the model, select only the beam components on the Floor Plan. Click on the „**Floating palette / Beam / Edit Beam**” command. The solution opens a dialog, which shows the available beam composites on the model. You have to select the number of the beam structure, which you want to refresh. Click „OK”, the solution will update all the beam objects (structures) on the Floor Plan, which have the same number which you selected. The 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.

2.8 The new thermal bridge breaker

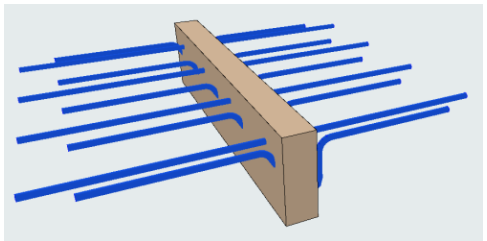


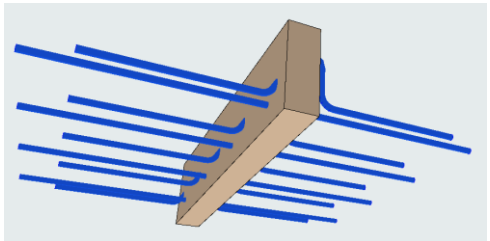
This new object is an experiment aimed at ensuring the Reinforcement add-on can meet every challenge—making it possible to model any kind of reinforcement using complex elements.

Until now, the program did not offer a comprehensive way to handle thermal bridge breaks; they had to be assembled manually. This new object is a base element whose dimensions and materials can be adjusted.

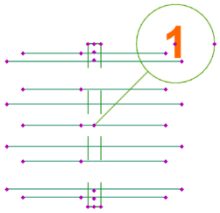
Since this is an element typically needed in specific, unique cases rather than general applications, it must be placed and configured manually.

On the first page of the settings window, the main parameters can be adjusted, such as the identifier, rebar diameter and quality, and geometric dimensions.
The second page allows you to set the number and position of longitudinal and shear rebars.
The third page includes the usual label settings, and the fourth page contains the 3D appearance parameters.





As seen, the element doesn’t have many parameters, but it still allows for the proper and effective design of a thermal bridge break in reinforcement.



In both the floor plan and 3D view, selecting the object reveals many editable hot spots, so—just like with other elements—its dimensions and geometry can also be modified manually.

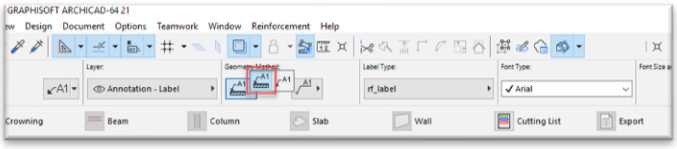
2.9 Deleting Elements

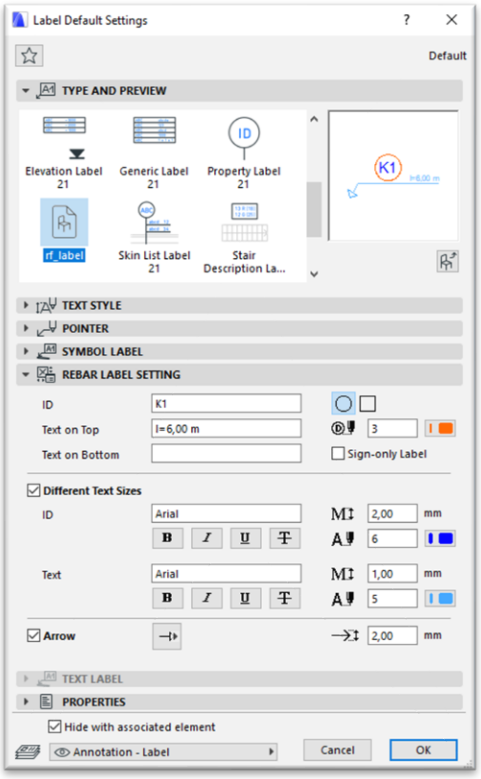
To delete any rebar construction from the 3D model, you can use the ArchiCAD default process. Select the object on any views and click the „DEL" button or select the „**Edit / Delete**" command.
On the other hand, the deleted objects will not be deleted from the cutting list until you do not refresh them. To clear the deleted objects from the cutting list, use the „**Floating palette / List Placement**" command.

2.10 Using ArchiCAD Label element

You can place default ArchiCAD label elements connected to the rebar objects.
Select the „**Label**" (

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

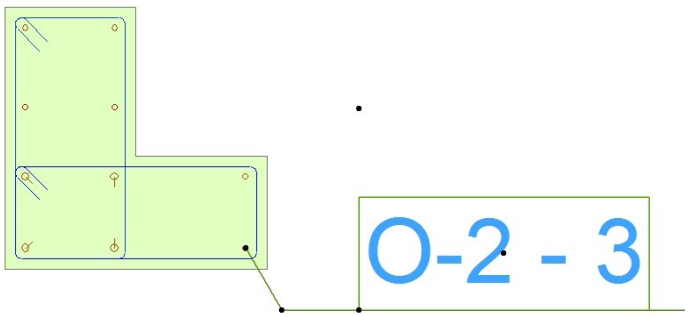




On the working area select the „**Label connection to objects**" placement method. In this case, the settings of the label, such as text and shapes, will be set directly on the Reinforcement object interface, in the Label tab. By choosing the **"Independent"** placement method, the changes are set in the Label Default settings.

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 on 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 its hotspots to the right position.

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

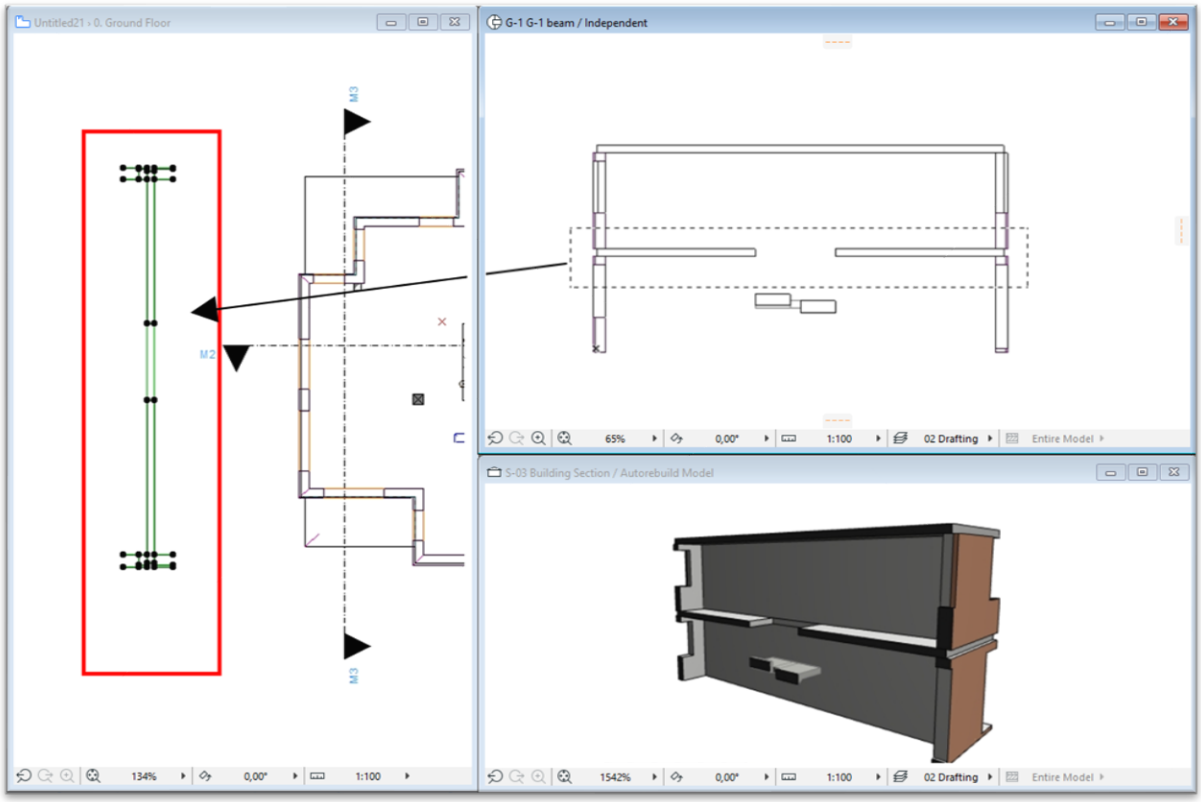


2.11 Creating Section drawings

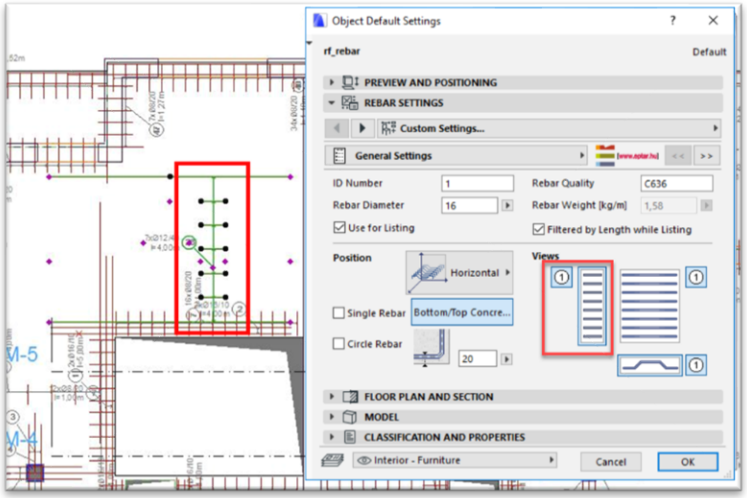
You can create section drawings in two different ways:
- by placing the model section view on the Floor Plan and move the side views of the rebar objects or
- by working on the section view of the model, picking up the parameter sets of rebar objects and placing 2D objects to the drawing.

Both processes have benefits and disadvantages, depending on what is your preferred working method you can use either of them.

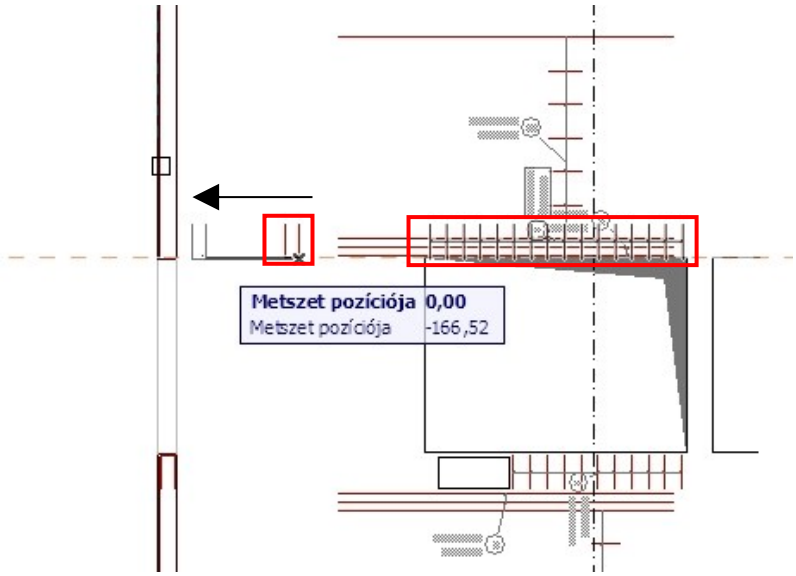
Creating section view on the Floor Plan
Create an ArchiCAD section line on the Floor Plan where you want to define the section. Open the section view and mark the working area which shows your affected slab. Copy the contour from the section window and place it on the Floor Plan, to the right position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



Select one of the rebar objects which crosses the section line on the Floor Plan. Open the „**Object Settings**" dialog and click the side view icon on the „**General Settings**" tab page.

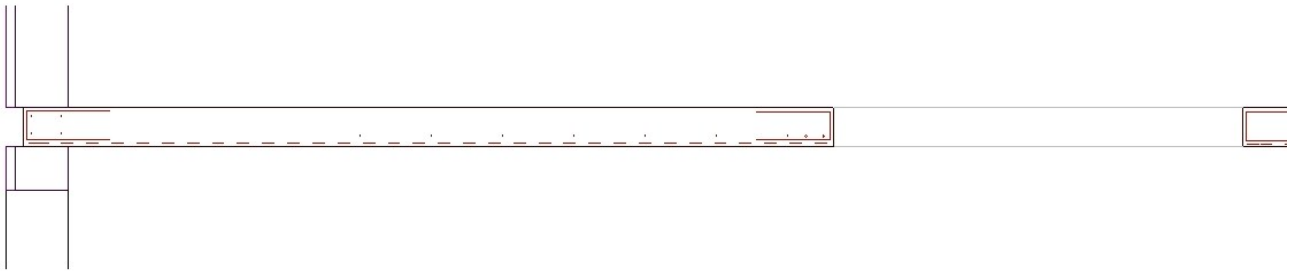


With the help of the editable hotspots move the section view of the object to the right position and rotate it to the right direction if it is necessary.
Select a mesh object in the cross section line, open the „**Object Settings**” dialog and click the „**Show Section**” check box. Select the direction 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 as necessary.



In version 2.0 the mesh which has been placed by Eptar add-on has no such section view (check box is grey). In this case, draw a single line to represent the mesh on section.

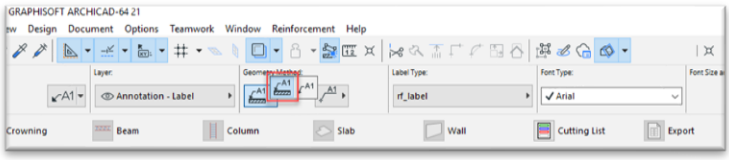
Use the process above to all objects, which



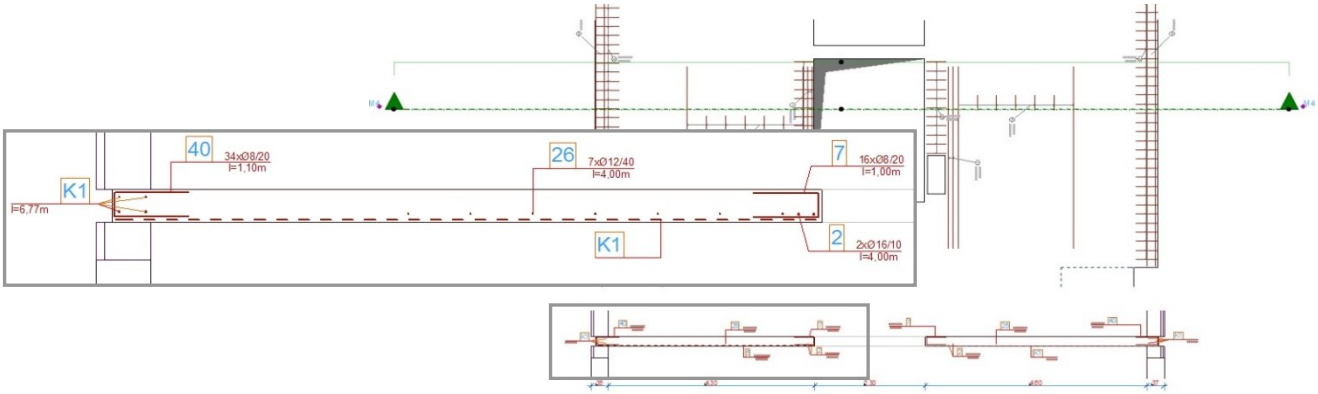
is affected in the section view.

To place labels for the section drawing use ArchiCAD label objects. Select the „**Label**” (

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



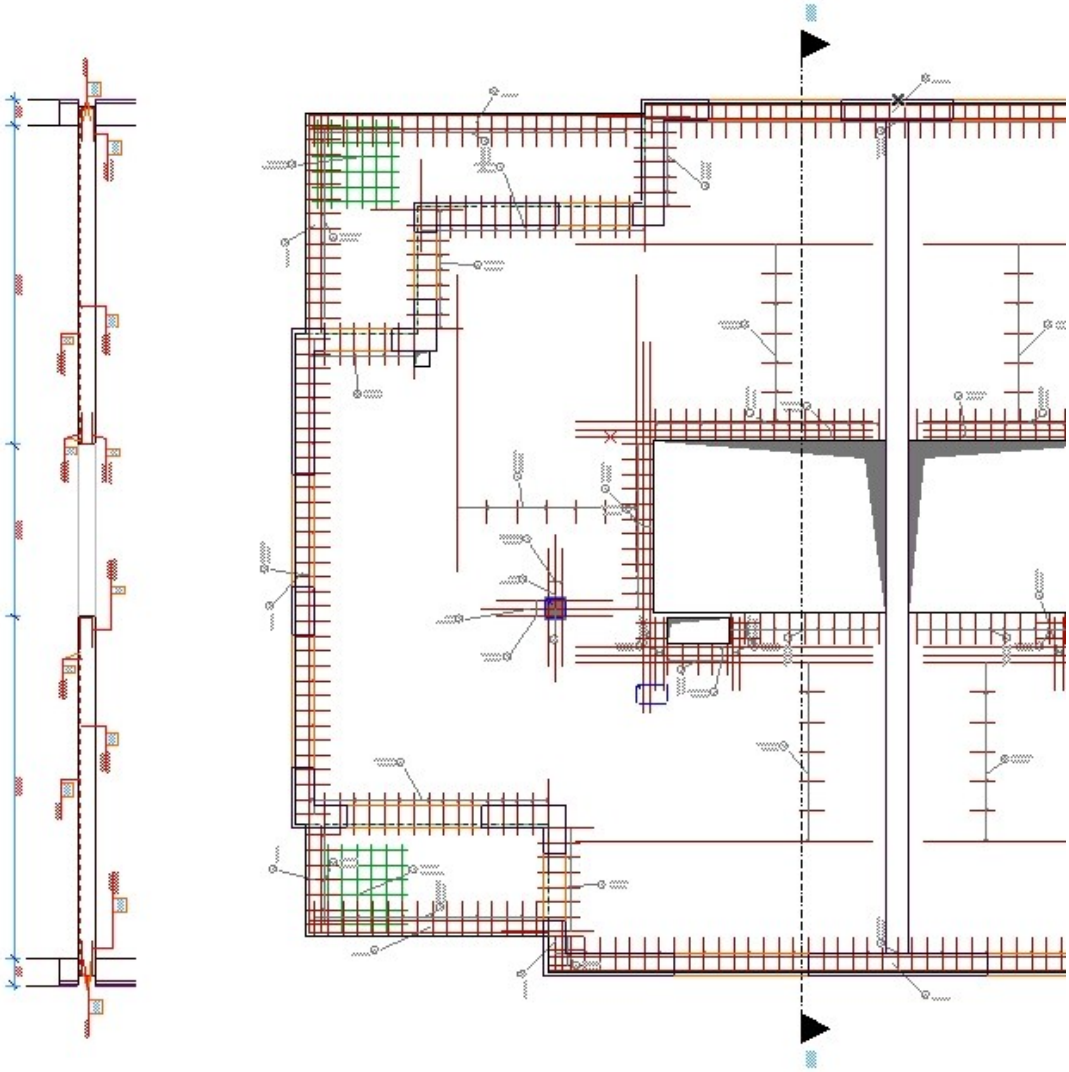
On the working area, select the „**Label connection to objects**” placement method.
Click on a hotspot or hotline of the object on the Floor Plan to place and connect the label to the object..The label will be placed on the Floor Plan connected to the object you have selected. If you want to move the 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, the label will be update at all parameter changes and the 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 and disadvantages
The benefit is to see all the changes in the section rebars that you make on the model.

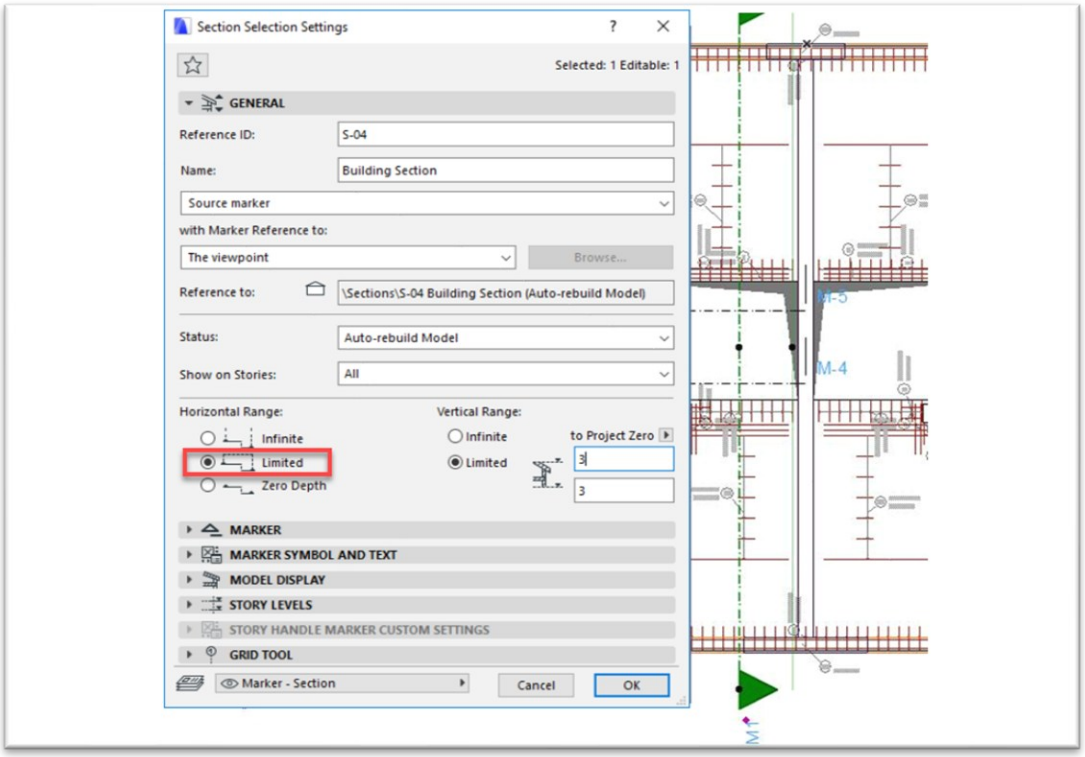
On the other hand, if you move or copy the objects on the model side, the section views will move or be multiplied with the objects together which can cause a confused section drawing.

If you use this section generation method, it is recommended to generate the section drawing during the last phase of the planning process, when you are not planning to change the structures in the model.



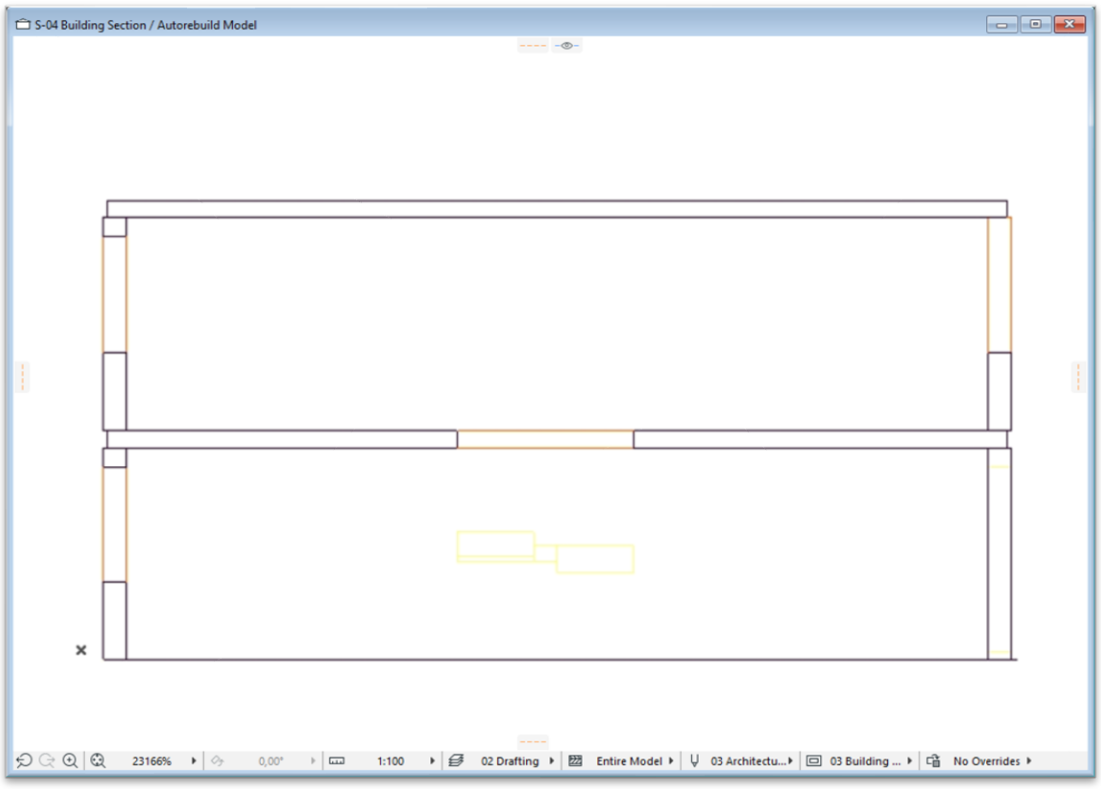
Defining section view in Section Window

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 the section window’s „**Horizontal Distance**” properties to „**Limited**”. Set up approx. 2 m distance on the Floor Plan.

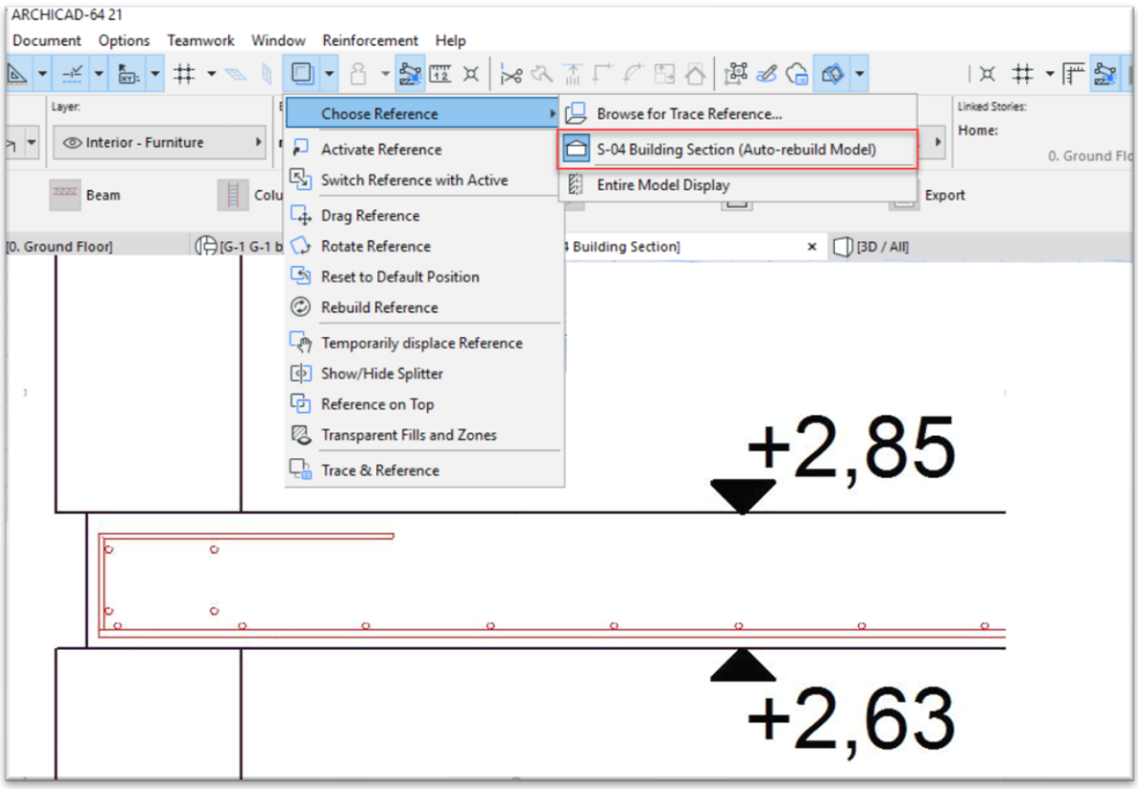


.

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.



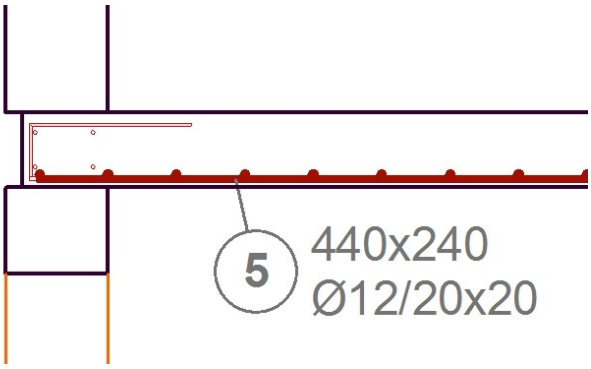
Set up the „**M-1**” Section as a „**Trace**” view in Section „**V-1**”. Open the „**M-1**” section view and set up the „**Layer Sets**” to show the slabs in the model in „**Wireframe**” mode. If you followed these steps, the rebar objects will be visible in the „**M-1**” section.



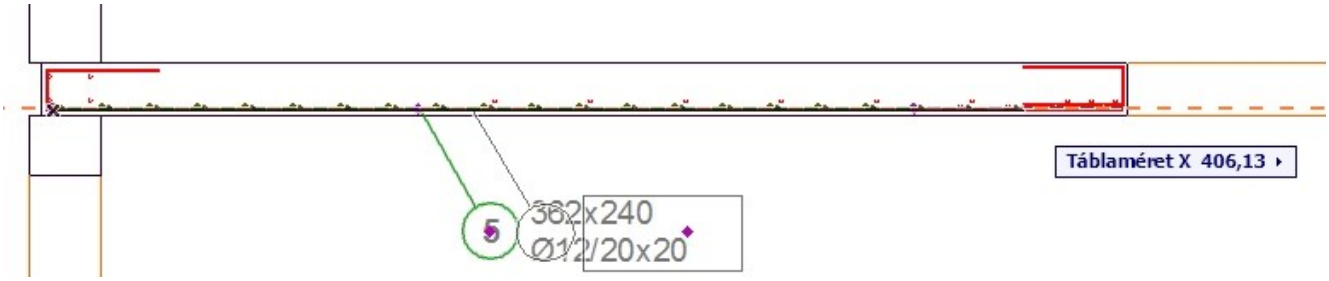
Open the „**V-1**” section window. In the section view you can see the contour lines of the structures and the 3D section models of the rebar structures (based on the trace drawing). Move your mouse over one of the rebar objects and click on it, while pushing the **Alt** button (**Alt + Click**).
With 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 to the desired place but somewhere close.



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.

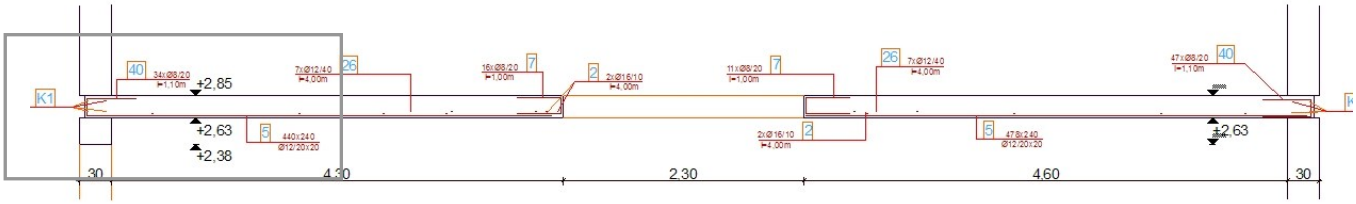


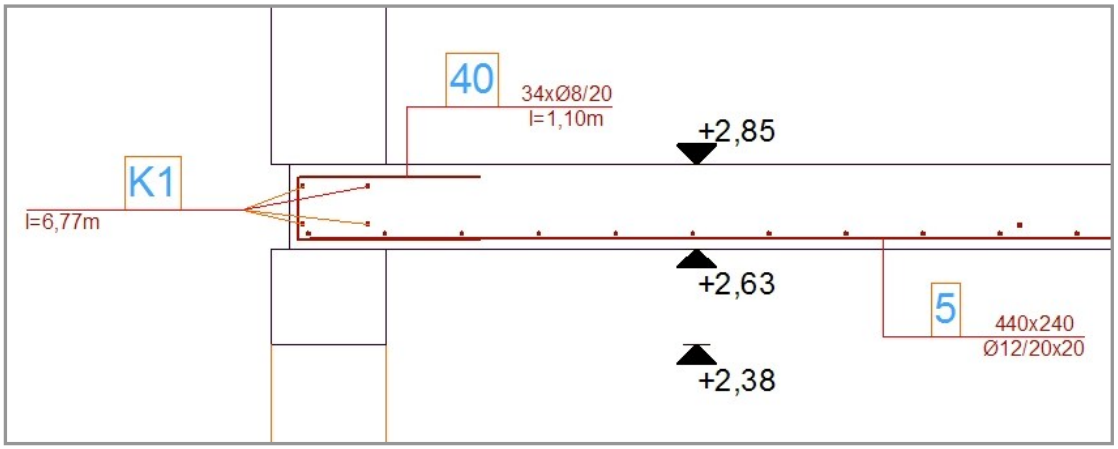
If you choose mesh objects, maybe the mesh section view will be shorter or longer than the original object section view. In this case, you can edit the length of the mesh's section view with the hotspots on the edges.



Repeat the process above until you place all the elements into the section view that is necessary to show.

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



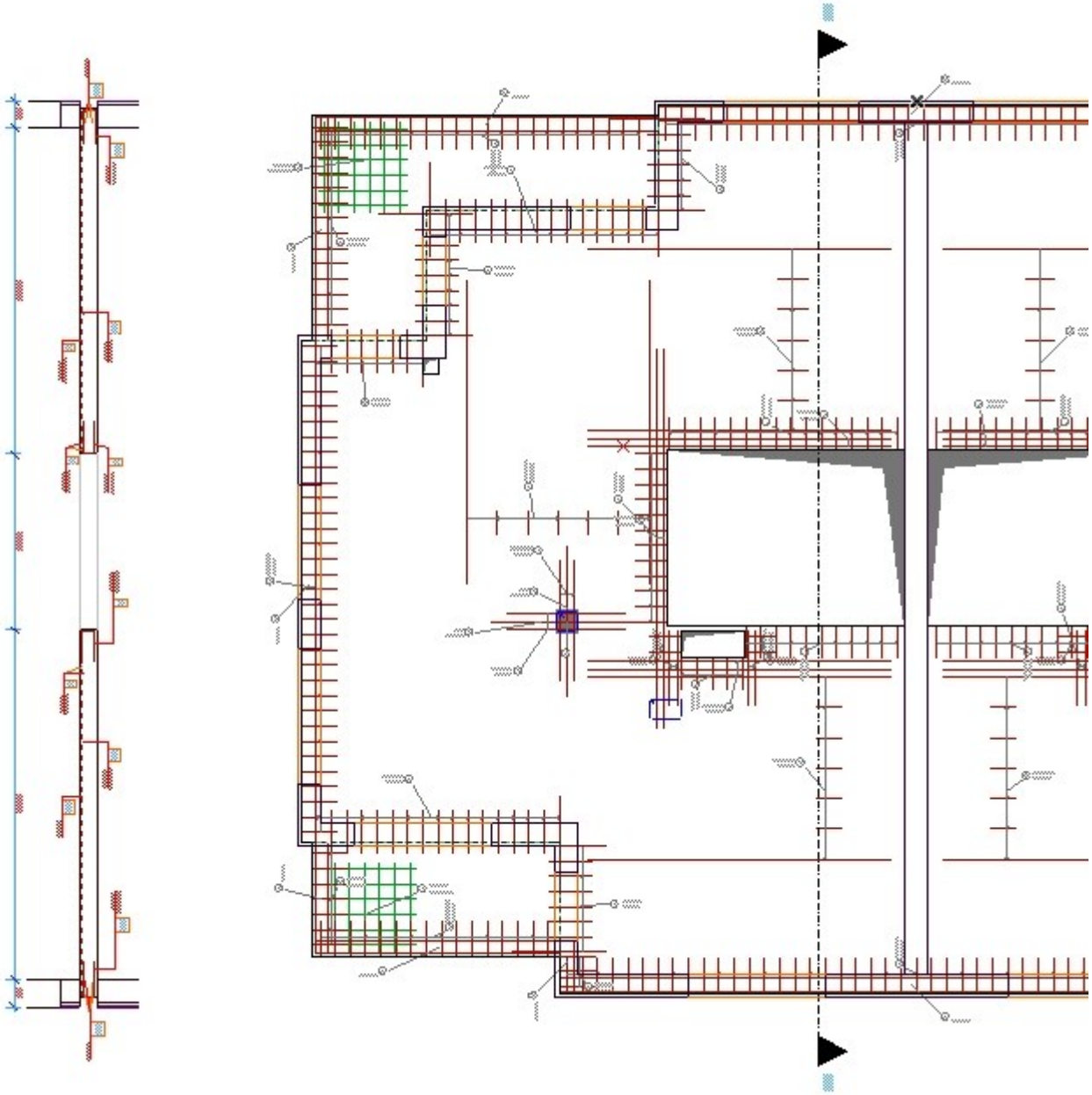


Benefits and disadvantages
In this solution you follow the ArchiCAD working methods and you generate the section view in the section window, so the section documentation is not in the model space.

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 cases.

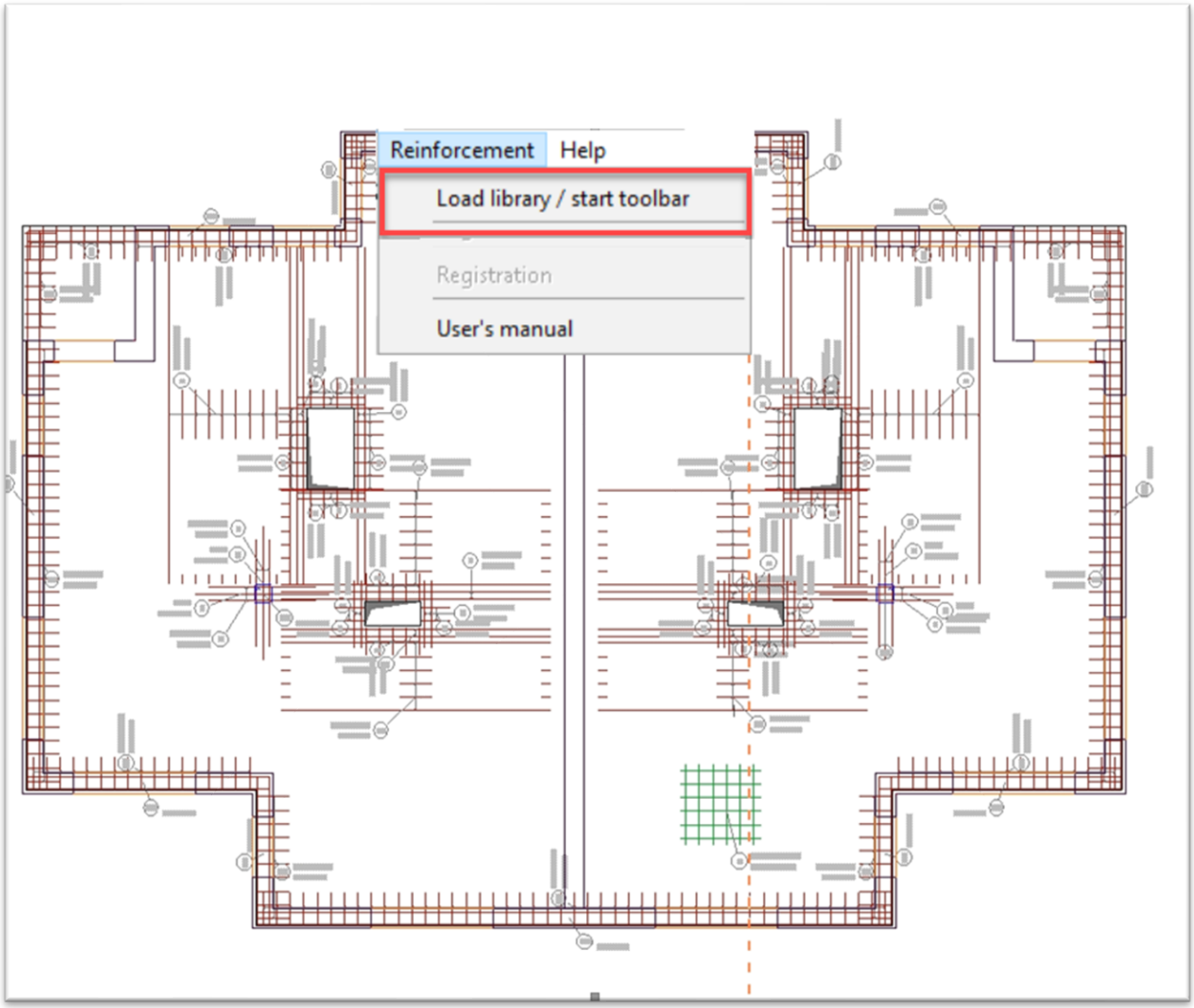
Suggestion:
If you use this section generation method, we suggest to generate the section drawing at the last phase of the planning process, when you are not planning to change the structures in the model.

.



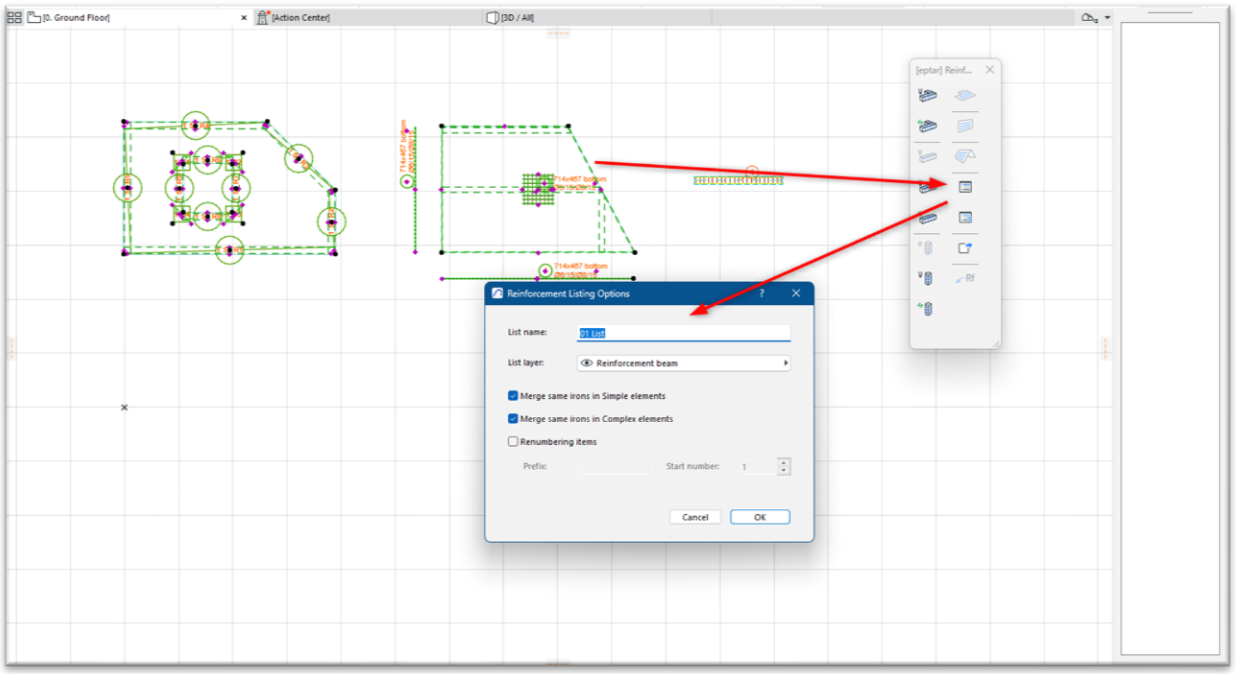
Chapter 3

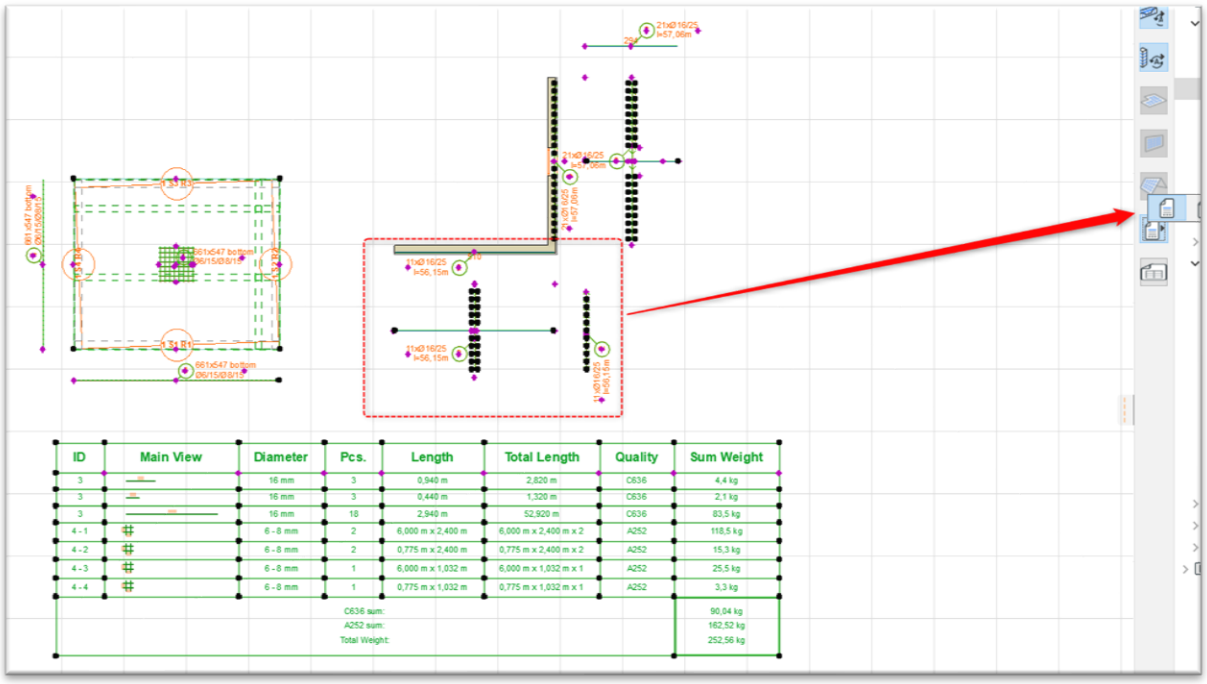
3.1. Cutting list placement and ID check



Reinforcement solution helps the users to create the necessary reinforcement drawings and models, it also generates cutting lists and checks the ID mistakes as well.
In point 2.5 (complex structures) we have already written about the cutting list definition which is available to be placed on the Floor Plan. You can place the same cutting list for several construction drawings as well.

In the 4.0 version you can create the Cutting List the same way as previously: select reinforcement elements, then click on the “Cutting List” icon, then place the list on the floorplan. This list will contain all rebars, meshes, stirrups of all selected elements.
Up until now if you wanted to add more bars or other elements to this existing list, you needed to select all elements associated with the Cutting List element and the Cutting List itself, then you needed to click on the “Cutting List” icon to update its content.

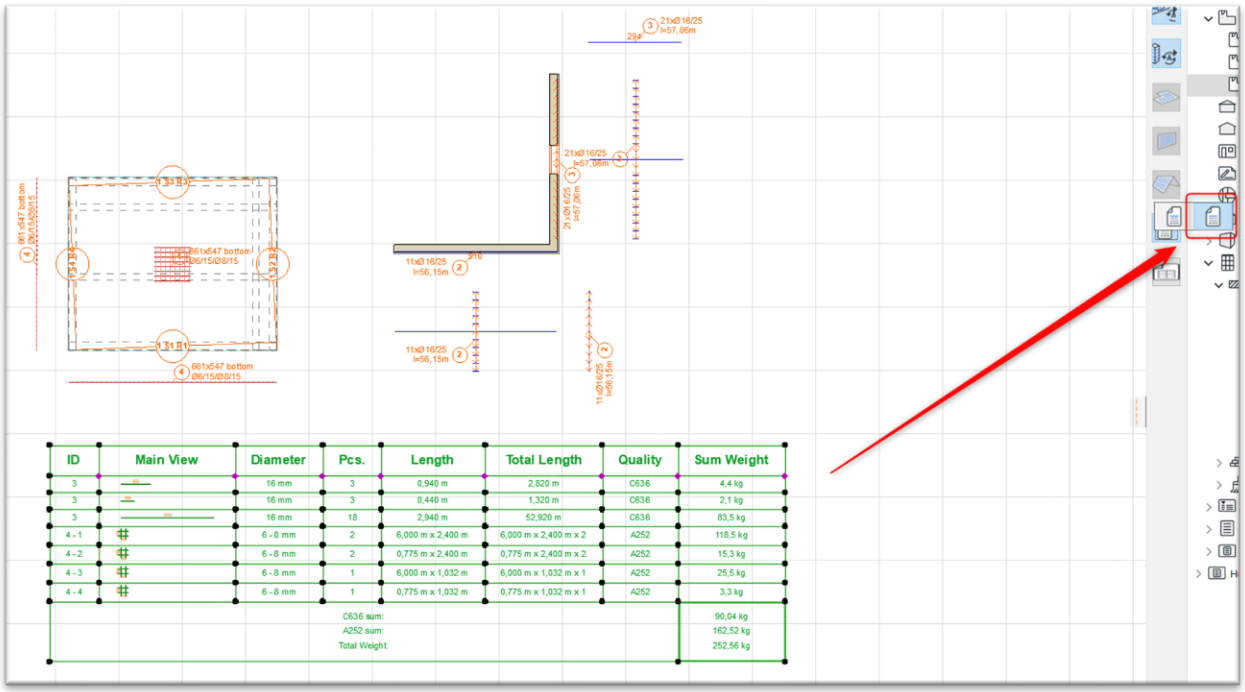


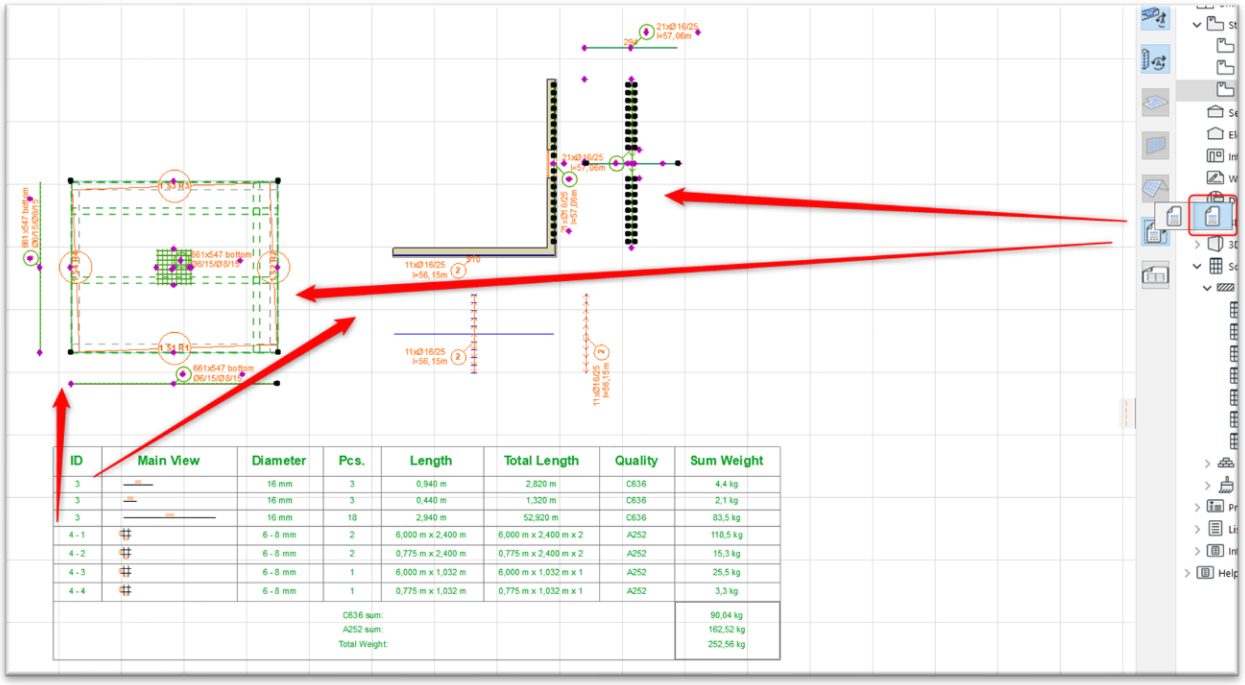


In the new version if you select the placed Cutting List element, and click on the second “Cutting List” icon,

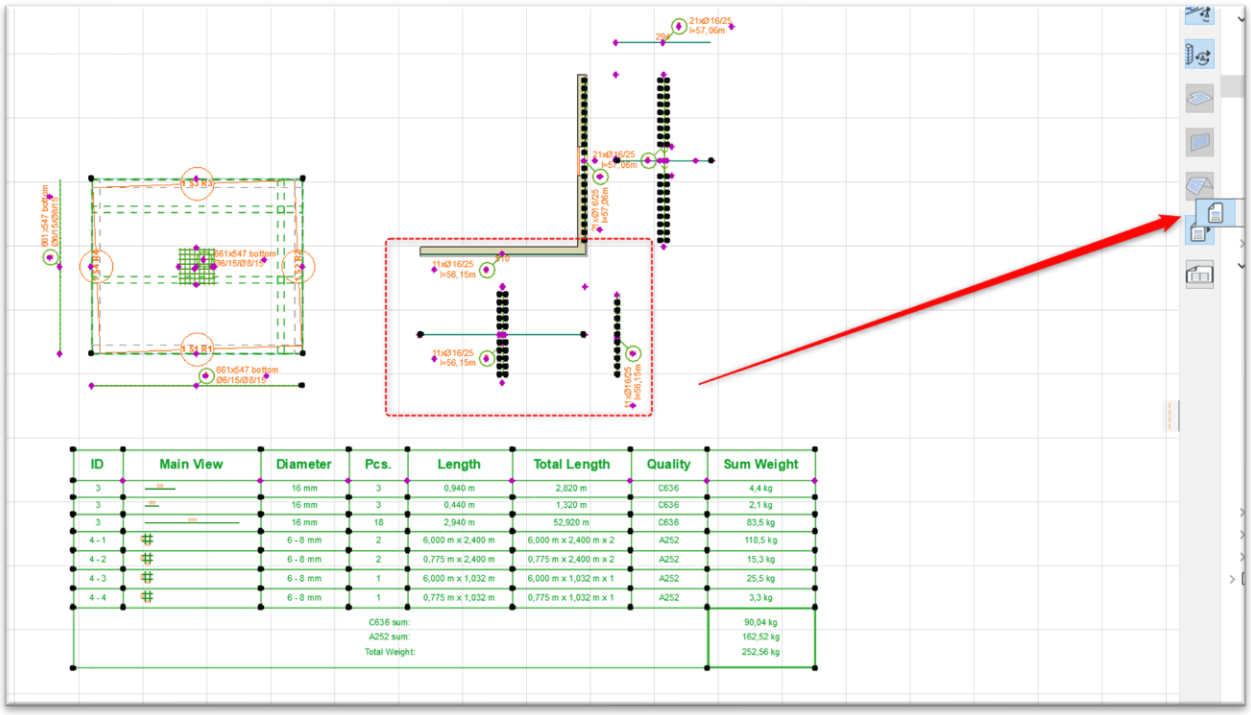


the program will find automatically select all associated elements that are listed in it.

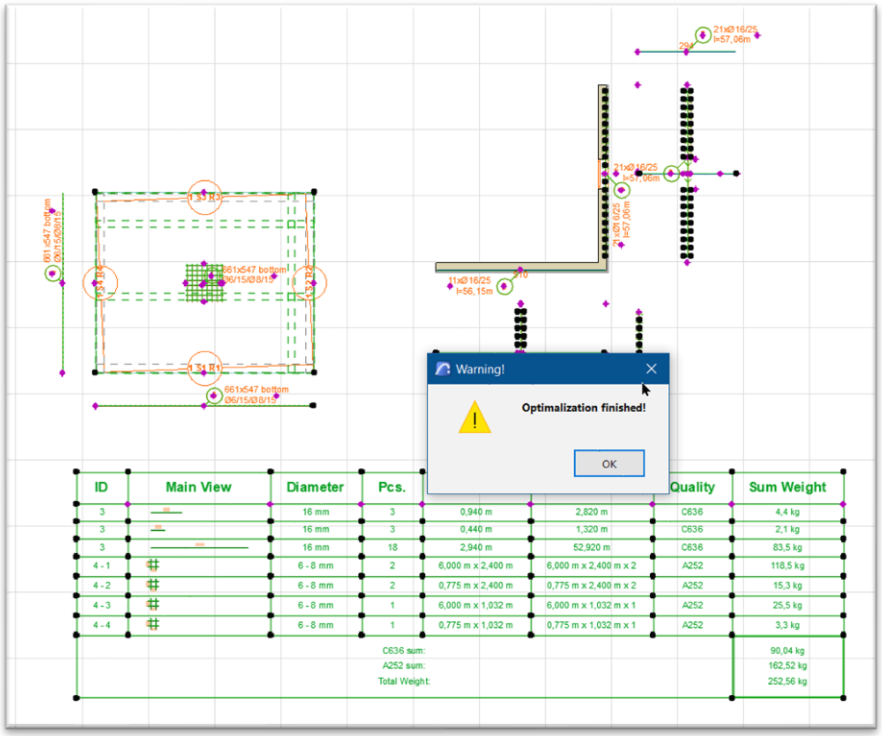




To add a new item, you should add to the selection this new element, plus select the Cutting List element, and click on first “Cutting list” icon in the palette.

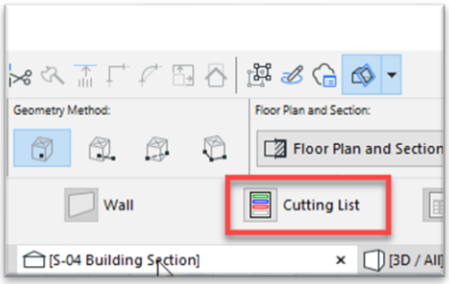


The list will be updated, and the newly added item will be associated with this list.

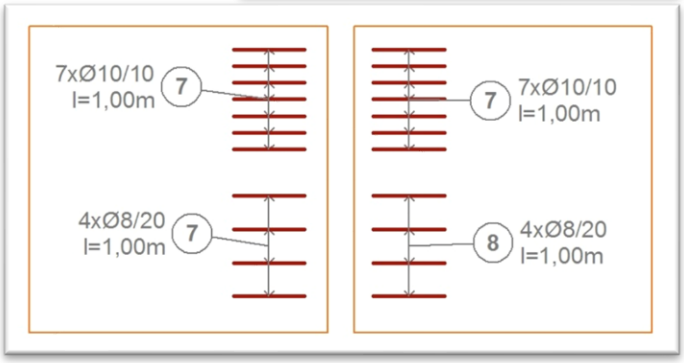


Of course the process also works backwards, when you would like to remove one or more elements from the Cutting List.

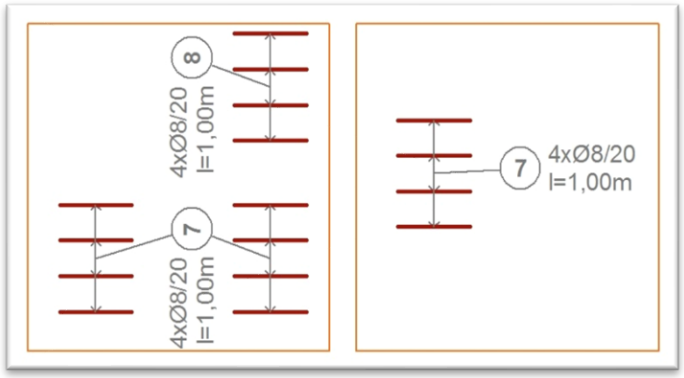
Create Cutting list



Select the Floor Plan view which you want to collect in one cutting list. Select the necessary layers and hide the unnecessary object layers. Choose the „**Reinforcement / Cutting List placement**" command. The 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 rebar types on the **visible layers on the active Floor Plan view**.

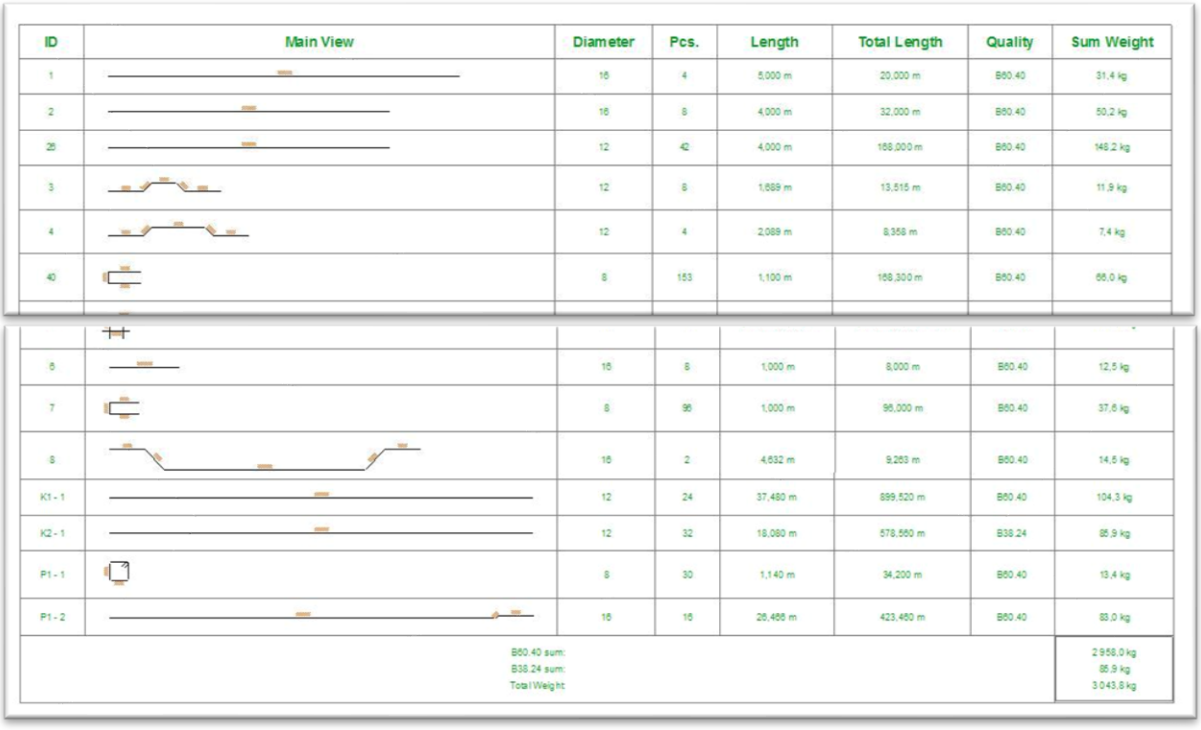


In case you placed objects with the same ID, but the parameter values (type, diameter, shape, and length) are different then the solution will change the ID number of one of them.



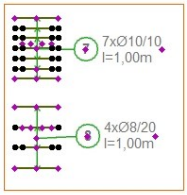
In case you placed objects with different IDs but the parameter values (type, diameter, shape, and length) are the same then the solution will collect them and use them with the same ID. The used ID will be the one which has been placed the most times on the Floor Plan.

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 does not place a new one but refreshes the existing one.

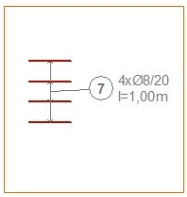


ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1		16	4	5,000 m	20,000 m	B60.40	31,4 kg
2		16	8	4,000 m	32,000 m	B60.40	50,2 kg
26		12	42	4,000 m	168,000 m	B60.40	148,2 kg
3		12	8	1,689 m	13,515 m	B60.40	11,9 kg
4		12	4	2,089 m	8,358 m	B60.40	7,4 kg
40		8	153	1,100 m	168,300 m	B60.40	66,0 kg
5		16	8	1,000 m	8,000 m	B60.40	12,5 kg
7		8	96	1,000 m	96,000 m	B60.40	37,6 kg
8		16	2	4,632 m	9,263 m	B60.40	14,5 kg
K1 - 1		12	24	37,480 m	899,520 m	B60.40	104,3 kg
K2 - 1		12	32	18,080 m	578,560 m	B38.24	85,9 kg
P1 - 1		8	30	1,140 m	34,200 m	B60.40	13,4 kg
P1 - 2		16	16	26,466 m	423,460 m	B60.40	83,0 kg
				B60.40 sum: B38.24 sum: Total Weight			2 958,0 kg 85,9 kg 3 043,8 kg

In case you have more cutting lists placed on the actual Floor Plan view, Reinforcement will cancel the operation, and not update any of the placed cutting lists. In this situation, please select the cutting lists to update and select all the rebars you want to be in the cutting list. Click on 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 that contain the unnecessary cutting lists or place them to independent detail drawings until you run the recalculation method again.

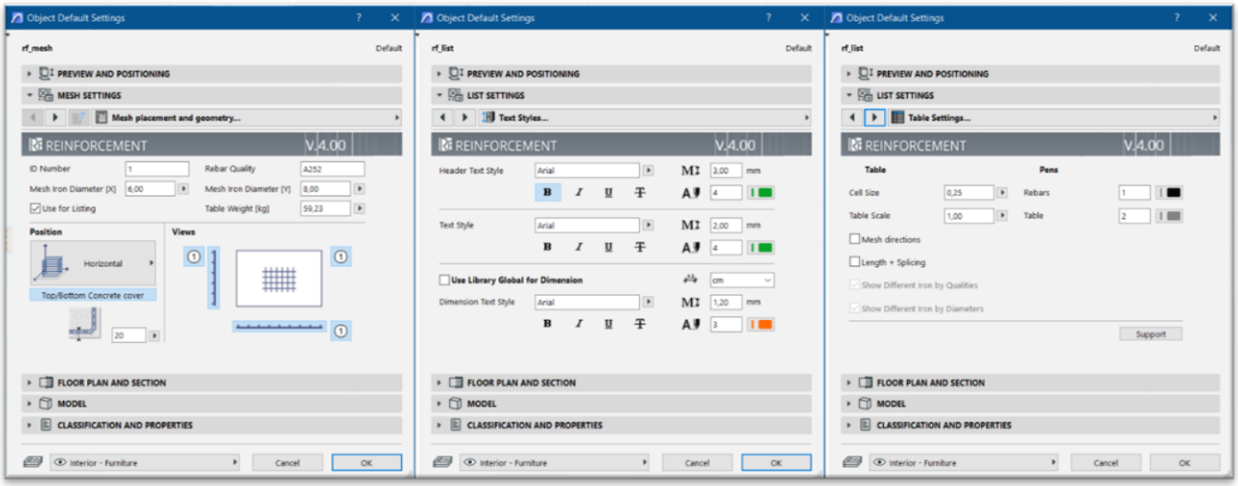


ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
7		10	7	1,000 m	7,000 m	B60.40	4,3 kg
8		8	4	1,000 m	4,000 m	B60.40	1,6 kg
Teljes vasalás összsúlya:							5,9 kg



ID	Rajzolat	Átmérő	Darabszám	Hossz	Teljes hossz	Anyagminőség	Össz súly
7	420 / 180 / 420	8	12	1,000 m	12,000 m	B60.40	4,7 kg
Teljes vasalás összsúlya:							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 boxes in front of the different column names. If you need bigger or smaller



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

rebar drawings, you can customize its size on the „**List Representation**" tab page.

When a listing is made, the program calculates and displays differently the splicing lengths.

ID	Main View	Diameter	Pcs.	Length + Splicing	Total Length	Quality	Sum Weight
1		16 mm	11	4,05 + 0,64 m = 4,69 m	51,56 m	C636	81,4 kg
Total Weight:							81,36 kg

For listing, it is possible to list every rebar of different lengths.
The program denotes the radius of the circular bracket, both in the Floor Plan and in the list.

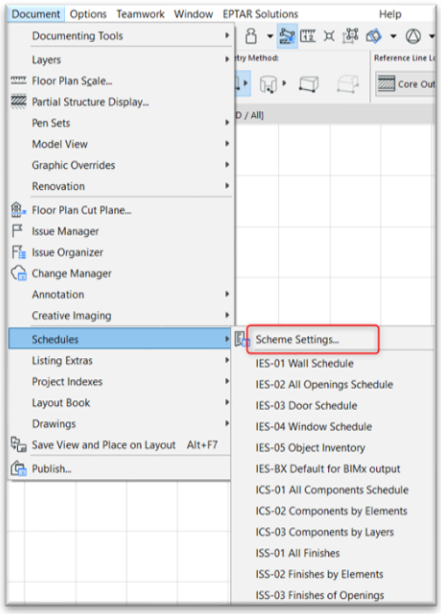
ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1		6 mm	16	1,72 m	27,47 m	B503	6,1 kg
Total Weight:							6,10 kg

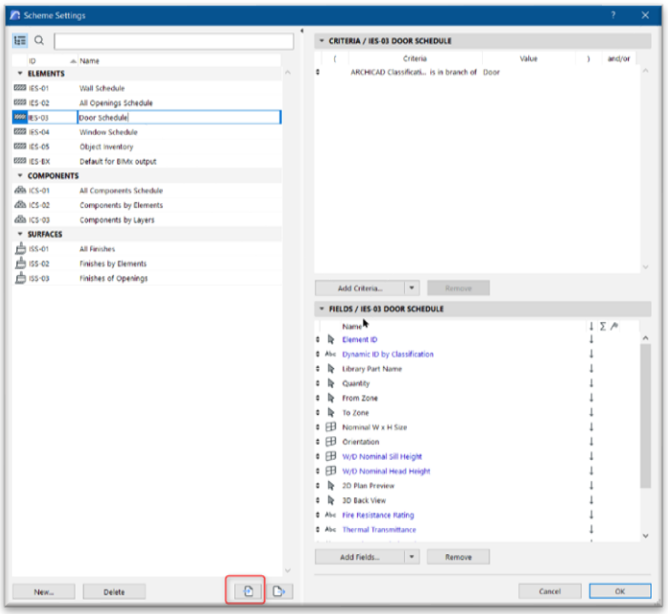
In case of a mesh, the properties of the divergent rails appear automatically in the list.

ID	Main View	Diameter	Pcs.	Length	Total Length	Quality	Sum Weight
1		6 - 8 mm	1	6,00 m x 2,400 m	6,00 m x 2,40 m x 1	A252	59,2 kg
1		6 mm	40	2,400 m	96,00 m	A252	21,3 kg
1		8 mm	16	6,000 m	96,00 m	A252	37,9 kg
Total Weight:							59,23 kg

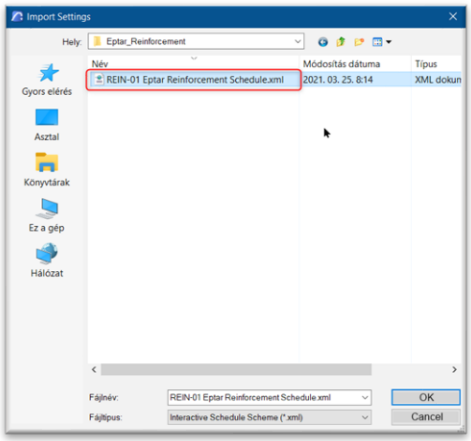
3.2. Using Interactive Schedule

With the help of this implemented ArchiCAD listing method you can also 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.



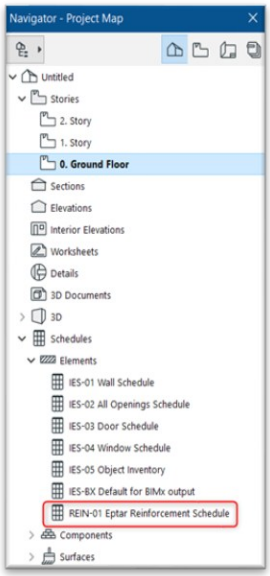


Choose the „**Import...**” button on the right side.
Find the installed Reinforcement Library on your computer and the „**[eptar]-Reinforcement.xml**” file in the same location.
In most of the cases you can find this file here:
„**C:/Program Files/Graphisoft/ArchiCAD/add-ons/[eptar]_Reinforcement**”.

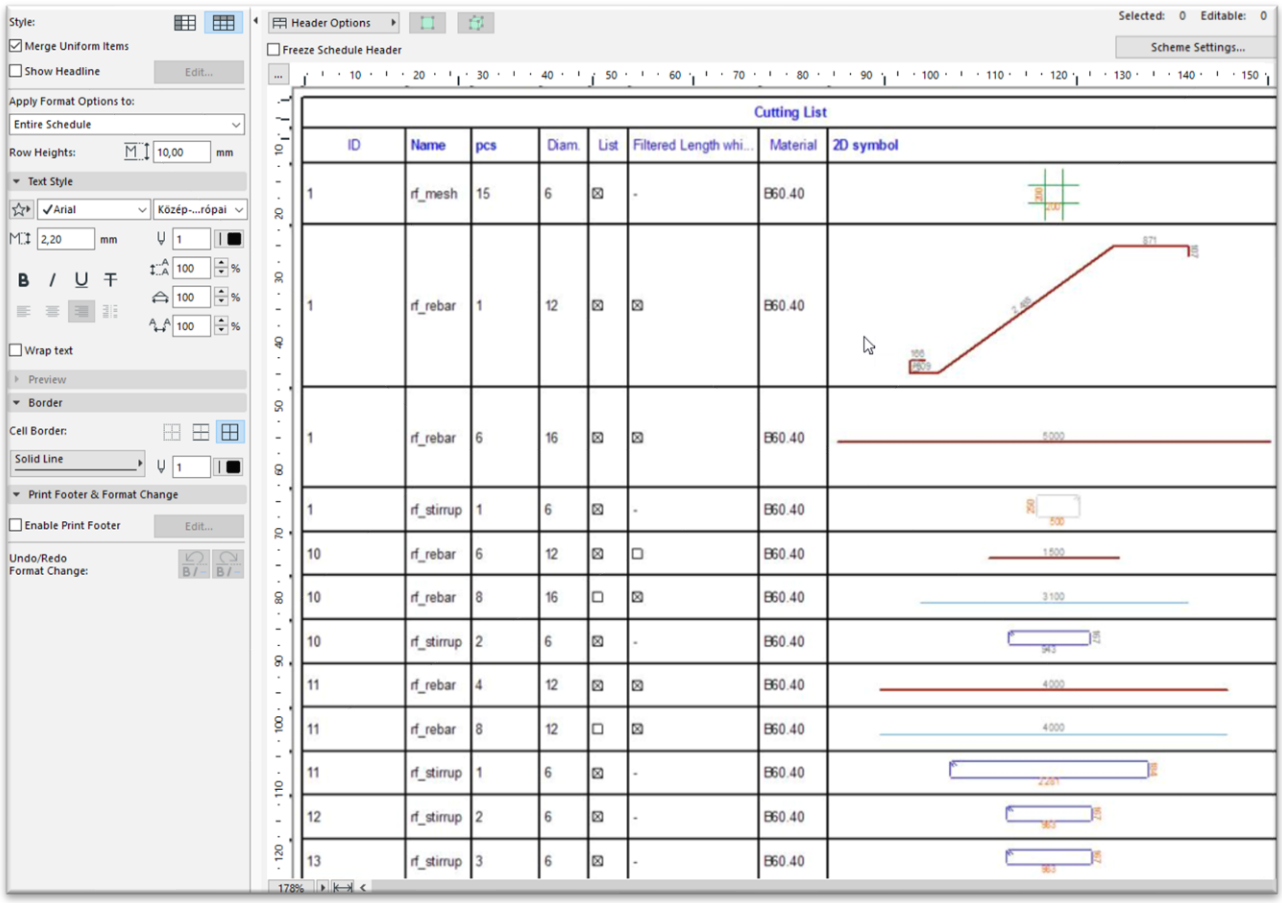


After the loading method choose the **[eptar]-Reinforcement** type from the schemes. Click the „OK” button.

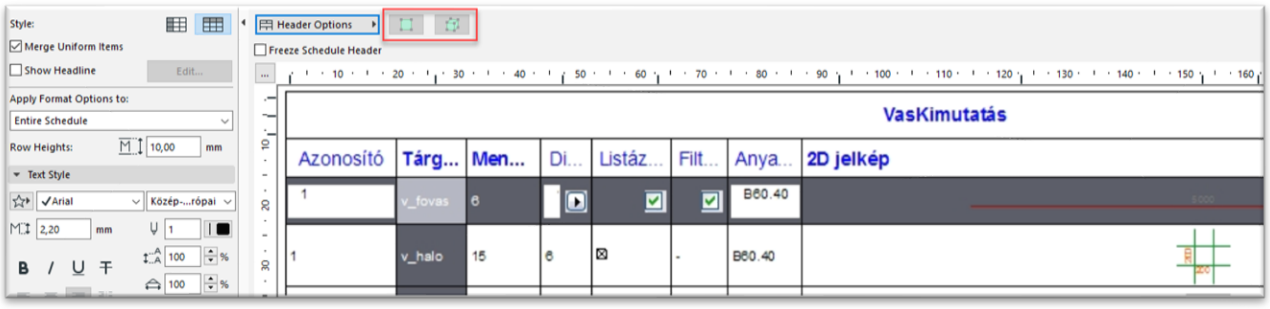
If your interactive schedule doesn’t open automatically, select the „**Schedules/ Element / [eptar]-Reinforcement**” in the „**Navigator**” on the right.



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

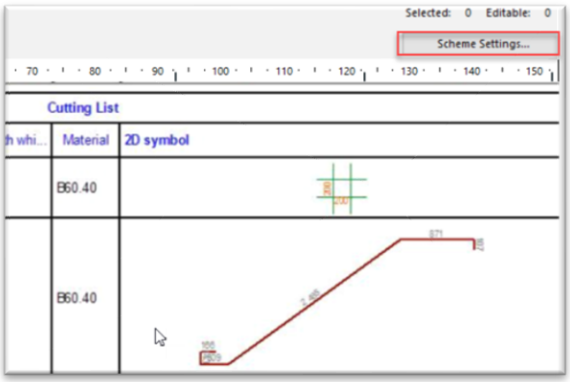


If you can not recognize an object based on the ID or the parameters in the list, you can use one of the buttons on the top of the dialog. Choose one rebar in the list and click the activated button on the header the list. ArchiCAD will show the object on the **Floor Plan** or in the **3D window** immediately.



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

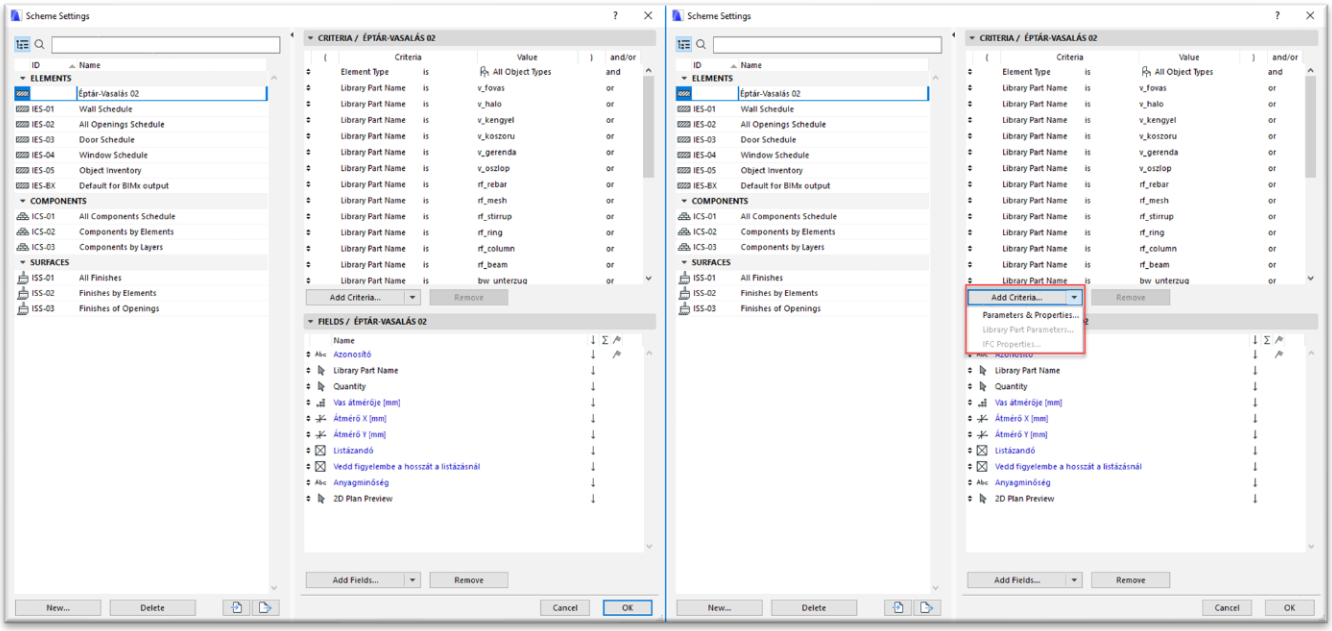
3.3. Creating custom Cutting list



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

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

On the bottom parts of the dialog, under the „**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 that can be used 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 collects all the loaded library parts and shows them in a list. Select the „**[eptar] 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 the „OK” button, your selected parameters will be available immediately in the „**Schedule fields**”.
In the 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 ArchiCAD documentation.

Chapter 4

4.1. Technical support

If you have a problem, question, comment or request regarding the solution, please your local reseller:

Country	Website	Contact
Austria		
Croatia		
Ecuador		
Finland		
Germany		
Greece		
Hungary		
India		
Indonesia		
Italy		
Latvia		
Lithuania		
Norway		
Philippines		
Poland		
Portugal		
Romania		
Serbia		
Slovakia		
Slovenia	https://www.pilon.si/	
Sweden	https://www.nordicbim.com/	
Switzerland	https://www.bimticino.ch/	
Thailand	https://eptar.hu/gfx/logok/applicadthai.png	
	https://www.dcs.co.th/	
Ukraine	https://www.facebook.com/graphisoft.ukraine/	
Vietnam	https://greends.com.vn/	

contact Éptár Kft.

EPTAR Kft.
1145 Budapest, Szugló u. 61-63.
+36-1 225-7355
www.eptar.hu
info@eptar.hu