



ES9997

Using Design Results from Analysis for Reinforcement Design in Revit

Thomas Fink
CEO, SOFiSTiK AG

Stefan Maly
VP Products, SOFiSTiK AG

Learning Objectives

- Learn how to use the Revit analytical model for analysis and design with Robot Structural Analysis and other software
- Learn how to use design results of other software to automatically design reinforcement layouts in Revit
- Learn how to check the proposed reinforcement layout against the required reinforcement
- Learn how to produce 2D reinforcement drawings out of a 3D model

Description

From the very beginning of Revit Structure software, it was possible to export the analytical system of a structure into several analysis and design software packages. This workflow is well accepted nowadays. This class will demonstrate how easy it is today to generate reinforcement layouts for buildings automatically from design results. Even if reinforcement objects have been created manually, it is possible to check the proposed reinforcement against required values from design results. Despite various examples of how to use a 3D reinforcement model for fabrication, it is still the norm in the construction industry to use 2D drawings for construction and therefore it is still necessary to produce 2D drawings from the model. We will demonstrate how this can be achieved quickly and easily using third-party software.

Your AU Experts

*After receiving his diploma in structural engineering at Technical University in Munich, **Thomas Fink** has worked in structural engineering and software development for over 30 years. He is co-founder and CEO of SOFiSTiK AG, a leading German supplier of software for analysis, design, and detailing. He was on the board of the German section of buildingSMART for more than 10 years, and chaired the working group "innovations" of the Bavarian Chamber of Building Engineers. Whenever he has time, he loves to fly balloons and to sail.*

Email: thomas.fink@sofistik.de

***Stefan Maly** studied structural engineering and computational mechanics in his German hometown Munich, Bavaria. He started his career at SOFiSTiK in 2003 as student worker, after that he worked as consultant to customers for several international bridge and building projects. 2008 he took responsibility for product management of the FEA software product line of SOFiSTiK. Since 2015 he is responsible for all products as vice president. He is a chartered engineer in the state of Bavaria and*

member of IABSE. He lives in nearby Munich with his wife and is father of two daughters. His private interests are architecture and aviation and long-distance running whenever his time allows.

Introduction

This paper will focus on the workflow after creation of a model in Revit. We assume that the analytical model, the loads and support conditions have already been assigned. The remaining steps are analysis, design and detailing. Detailing consists of two steps, creating the model and deriving construction documents out of it.

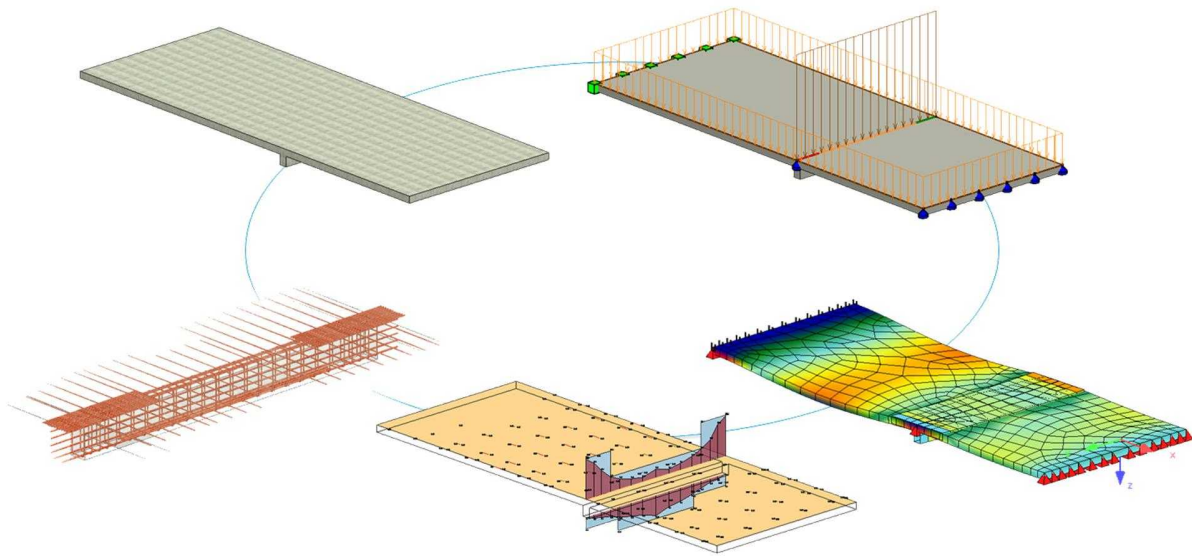


FIGURE 1: BIM WORKFLOW (STRUCTURAL ENGINEERING)

Obtaining Design Results in Revit using the Robot Structural Analysis Link

Revit Structure does not perform any structural analysis and design. It is therefore necessary to export the model to other analysis and design software. Several commercial products are available which plug into Revit and import the analytical model into their environment. The Autodesk 360 | Structural Analysis cloud service can be employed, using the 'Analyse in Cloud' function in Revit, however this service does not yet allow for RC design. In the following the principle workflow for obtaining design results using the Robot Structural Analysis Link for a simple concrete frame structure shall be described.

The example system is a simplified version of the 'Frame in Axis 2' taken from the European Commission's JRC Scientific and Policy Report: Eurocode 2: Design of Concrete Buildings - Worked Examples (http://eurocodes.jrc.ec.europa.eu/doc/1110_WS_EC2/report/1110_WS_EC2.pdf). Two characteristic loadcases, deadload and liveload have been assigned in Revit. The 2-span continuous beam is assumed to be supported by three 250x250mm columns, 3m high. The beam section ignores the effective width effects and is assumed to be 250x400mm made of C25/30 grade concrete.

Generation of Rebar Objects Using Results from Analysis and Design

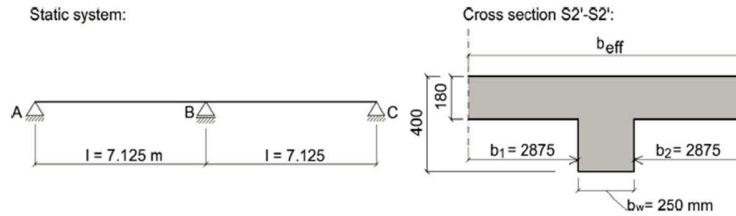


FIGURE 2: SYSTEM, CROSS SECTION

After finalization of the modeling in Revit the analytical model shall be exported to Robot for calculation. The following figures illustrate the necessary steps.

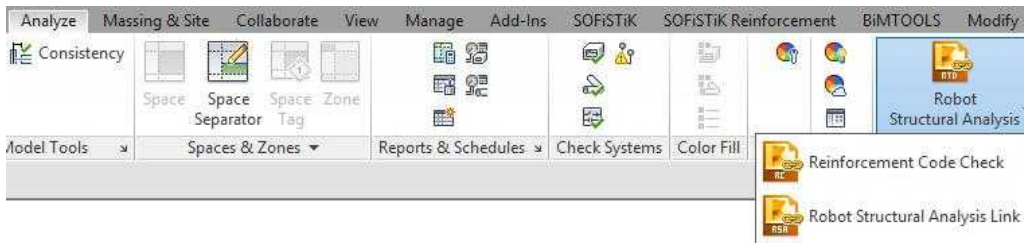


FIGURE 3: STEP 1 - ROBOT STRUCTURAL ANALYSIS LINK IN REVIT

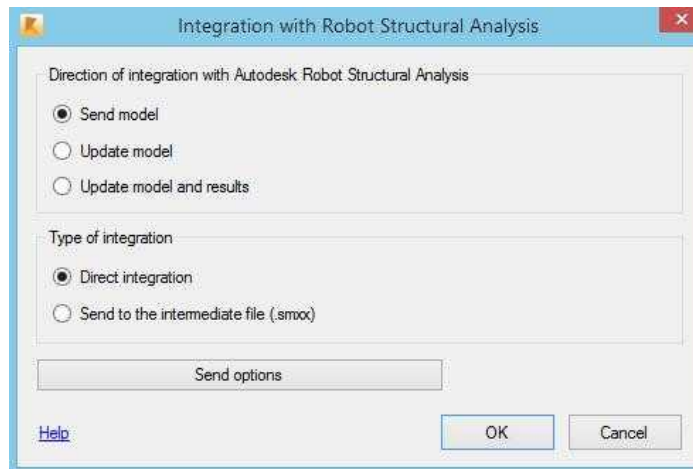


FIGURE 4: STEP 2 - SEND MODEL USING DIRECT INTEGRATION

After the completion of the data exchange the model can be viewed in Robot. In order to obtain a simple force envelope for design according to Eurocode, a manual combination is defined in Robot. The combination uses the (fixed) partial factors of 1.35 for deadload and 1.50 for liveload effects, hence creating a maximum hogging/support moment for ultimate limit state design.

Generation of Rebar Objects Using Results from Analysis and Design

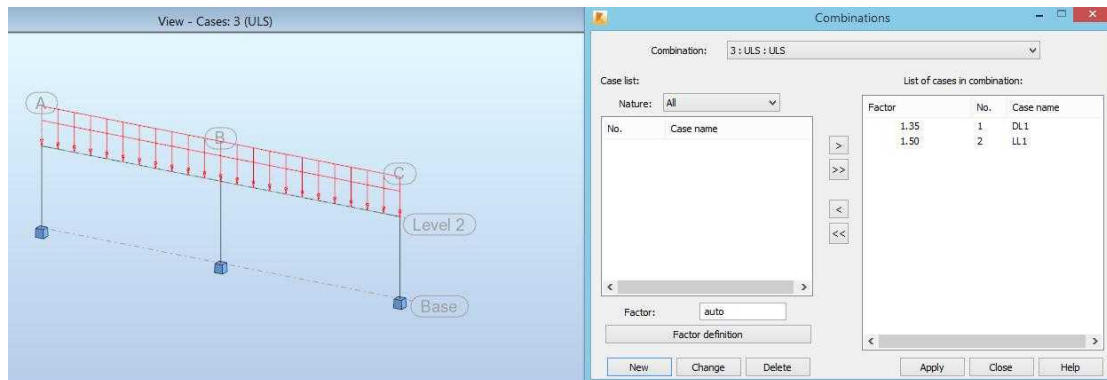


FIGURE 5: STEP 3 - ULS COMBINATION

After calculation, the results of this combination can be viewed. The figure shows the bending moment M_y in ULS (M_{yEd}) for the beam in Robot.

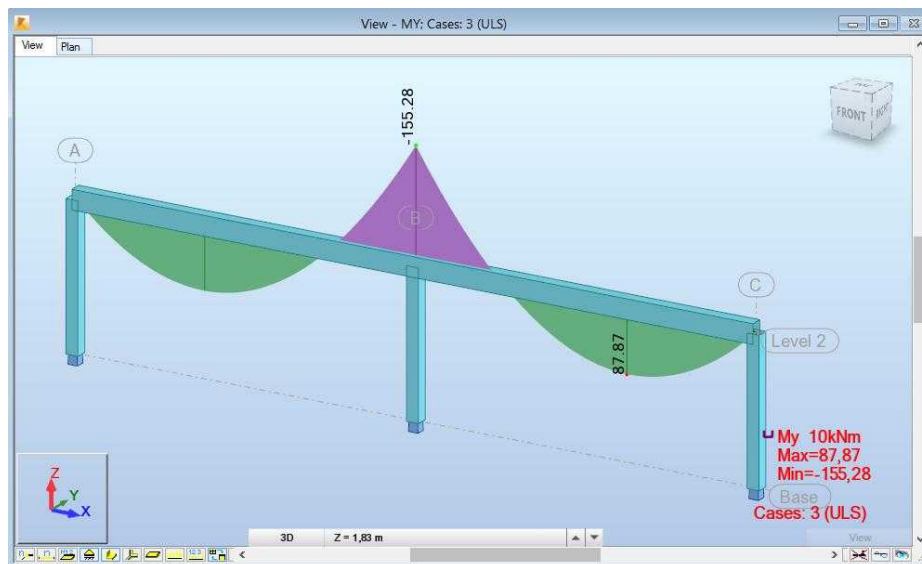


FIGURE 6: STEP 4 - M_y (ULS) FOR THE BEAM

Using the function 'Required Reinforcement of RC Beams/Walls' in Robot a design according to EN 1992-1-1 is performed. The following figure shows the Robot results of the design in tabular format.

Generation of Rebar Objects Using Results from Analysis and Design

Bar/Position (m)	Top required reinforcement (My) (cm2)	Top reinforcement - distribution (My)	Bottom required reinforcement (My) (cm2)	Bottom reinforcement - distribution (My)	Transversal reinforcement - type/distribution
1/ 0,40	0,0	-	1,96	2f16	2f6 34*14.0+79*6.0+34*14.0
1/ 2,08	0,0	-	5,64	3f16	
1/ 3,76	0,0	-	4,98	3f16	
1/ 5,44	1,39	2f16	1,25	2f16	
1/ 7,13	11,53	6f16	0,0	-	
1/ 8,81	1,39	2f16	1,25	2f16	
1/ 10,49	0,0	-	4,98	3f16	
1/ 12,17	0,0	-	5,64	3f16	
1/ 13,85	0,0	-	1,96	2f16	

FIGURE 7: STEP 5 – REQUIRED MEMBER REINFORCEMENT IN ROBOT

Now, the Robot Analysis Link in Revit can be used to update the model including results and required reinforcement.

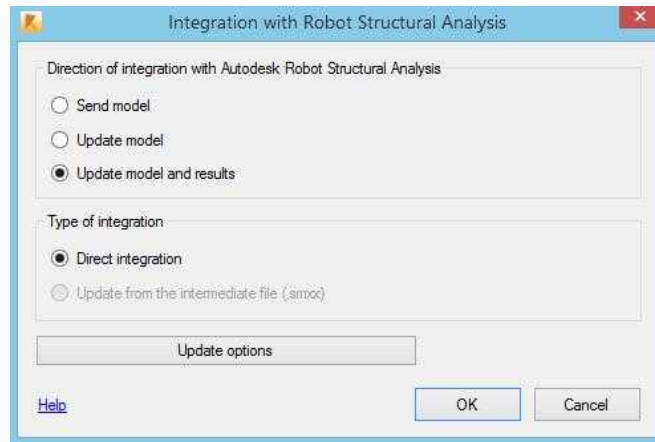
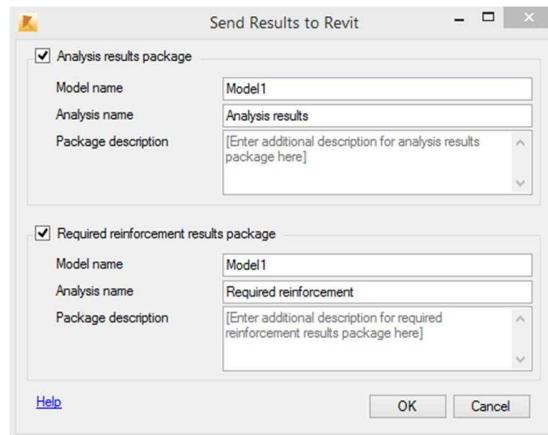


FIGURE 8: STEP 6 – UPDATE OF MODEL IN REVIT

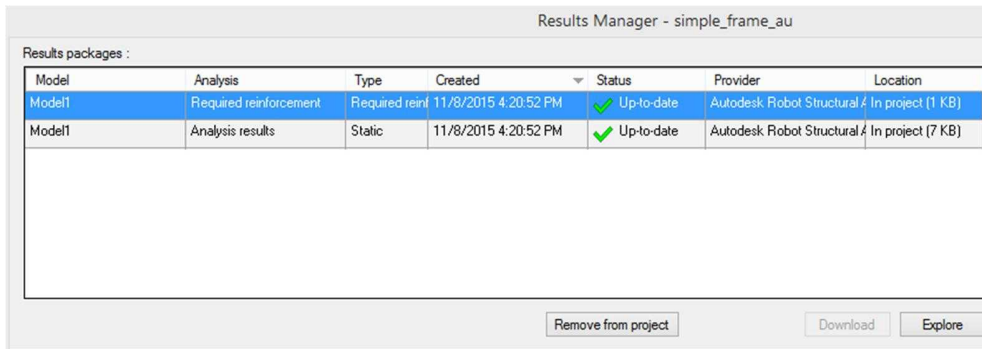
The results are saved as Results Packages in Revit. During the send process the user is prompted for additional information and modification of the description and name of the packages for analysis and required reinforcement results. Using the 'Results Explorer' function in Revit the imported results can be checked. The following figures show the necessary steps and dialogs.

Generation of Rebar Objects Using Results from Analysis and Design



The 'Send Results to Revit' dialog box contains two sections, both with checked checkboxes. The first section, 'Analysis results package', has 'Model name' set to 'Model1', 'Analysis name' set to 'Analysis results', and a 'Package description' field with placeholder text. The second section, 'Required reinforcement results package', has 'Model name' set to 'Model1', 'Analysis name' set to 'Required reinforcement', and a 'Package description' field with placeholder text. At the bottom, there are 'Help', 'OK', and 'Cancel' buttons.

FIGURE 9: STEP 7 - STORAGE AS RESULTS PACKAGES



The 'Results Manager - simple_frame_au' window displays a table of results packages. The table has columns for Model, Analysis, Type, Created, Status, Provider, and Location. Two rows are visible, both with a status of 'Up-to-date' and a green checkmark icon. Below the table are buttons for 'Remove from project', 'Download', and 'Explore'.

Model	Analysis	Type	Created	Status	Provider	Location
Model1	Required reinforcement	Required reinf	11/8/2015 4:20:52 PM	✓ Up-to-date	Autodesk Robot Structural 4	In project (1 KB)
Model1	Analysis results	Static	11/8/2015 4:20:52 PM	✓ Up-to-date	Autodesk Robot Structural 4	In project (7 KB)

FIGURE 10: OPTIONAL STEP - RESULT MANAGER FOR CHECKING THE RESULTS PACKAGES

Finally the required reinforcement is displayed along the members in Revit. This Step completes the workflow to obtain design results in Revit using the analysis link.

Generation of Rebar Objects Using Results from Analysis and Design

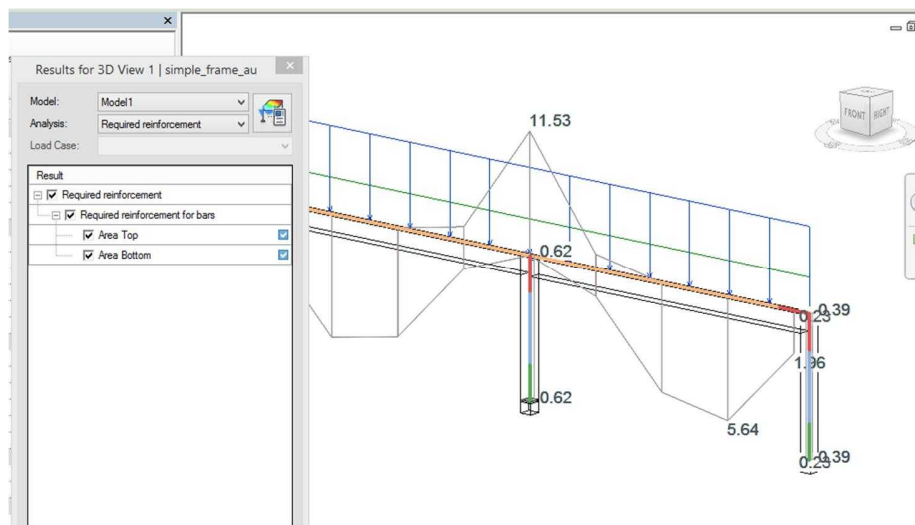


FIGURE 11: FINAL STEP - DESIGN RESULTS IN REVIT

The required area of reinforcement is shown on different locations of the structure. The longitudinal reinforcement is shown for the two layers of beams (top and bottom) and columns (sides). For slabs and walls, the amount of reinforcement is shown in each direction for each layer, plus additional information is provided on the orientation of the reinforcement.

Generation of 3D Reinforcement in Revit Structure

There are several ways to generate the rebar model in Revit. In this paper the focus shall be automated generation of reinforcement using the third party product 'SOFISTIK Reinforcement Generation' which is available via the Autodesk App Store for Revit 2016.

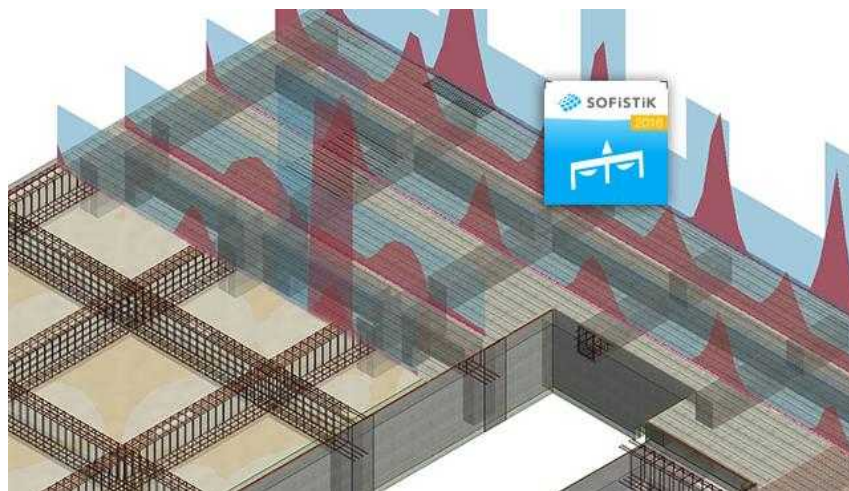


FIGURE 12: SOFISTIK APP REINFORCEMENT GENERATION

In a first step a 3D View of the simple frame in Revit is prepared, the toolbox of Reinforcement Generation (RCG) is shown as well in the figure below.

Generation of Rebar Objects Using Results from Analysis and Design

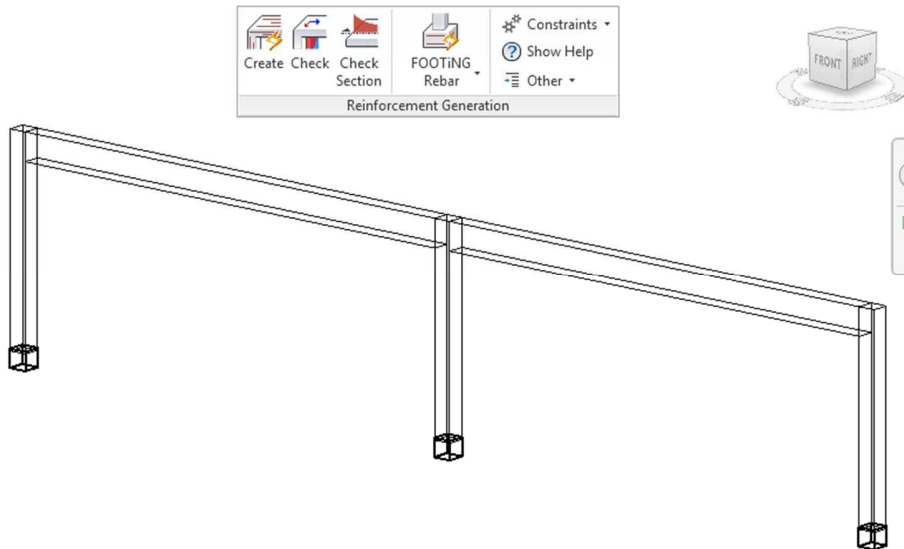


FIGURE 13: STEP 1 - WIREFRAME VIEW OF THE SIMPLE FRAME IN REVIT

Using the 'Create' button of RCG, the user is able to generate an automatic reinforcement proposal for beams, columns, walls and slabs. To demonstrate the workflow, only the reinforcement for the beam shall be generated using the app. After clicking the 'Create' button the user is prompted to a dedicated dialog. Here the Revit results packages are selected as data source automatically, as no other design data is available at this step.

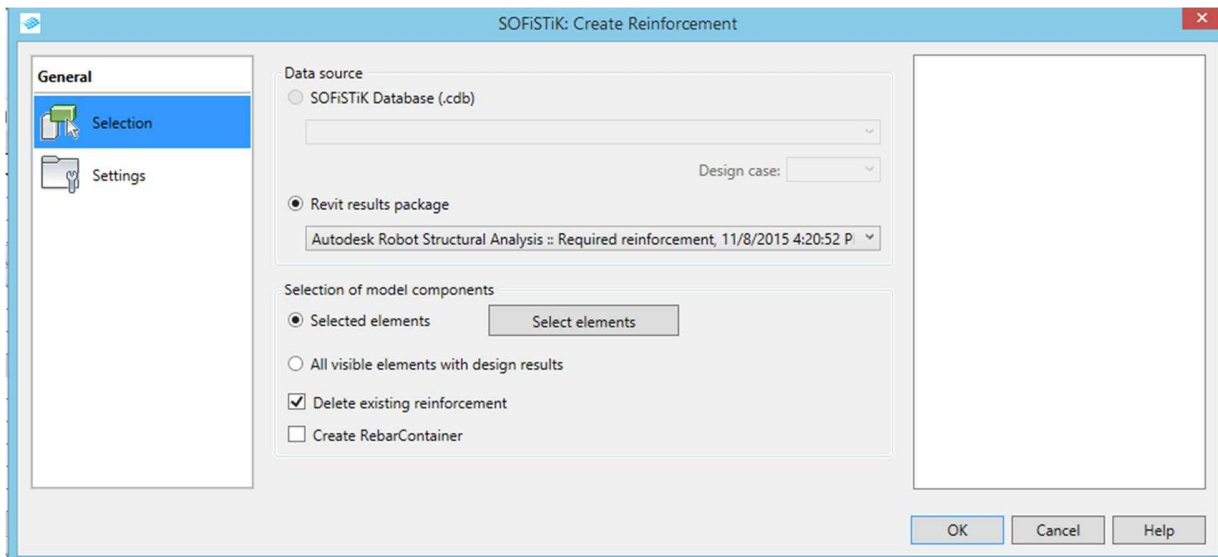


FIGURE 14: STEP 2 - CREATE REINFORCEMENT DIALOG

After selection of the beam members, further parameters can be set. The app offers dedicated settings for the assignment of base reinforcement in the upper and lower layer as well as for the stirrups. The

maximum required reinforcement from the design for the respective layers is shown and allows easy plausibility checks.

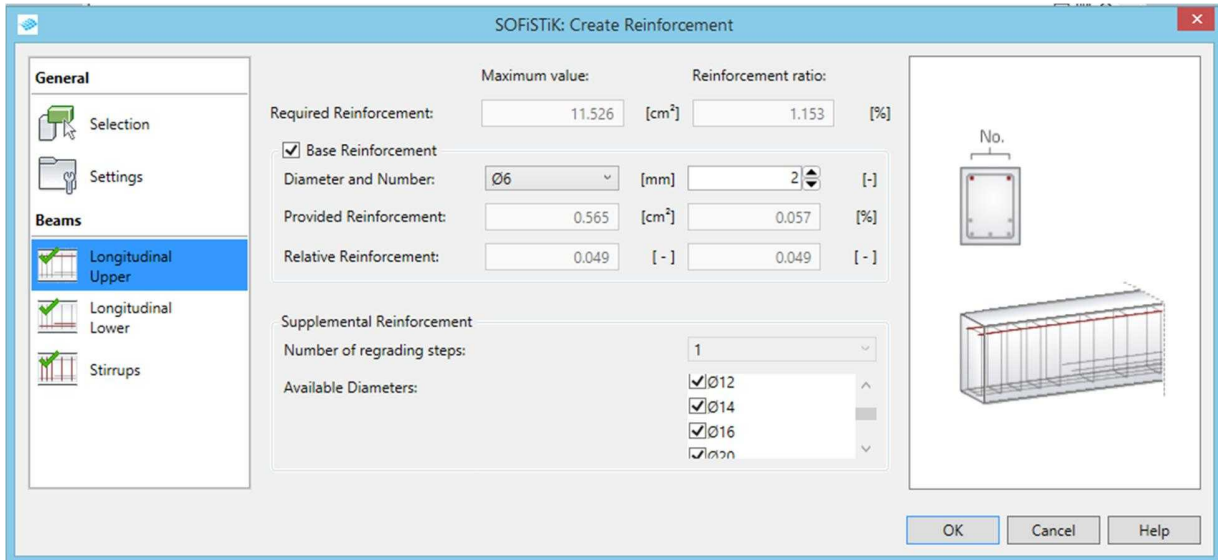


FIGURE 15: STEP 3 - BEAM REINFORCEMENT PARAMETERS

After confirming the defaults for the upper and lower layer, the final step shows that there is no required shear reinforcement from the previous design in Robot. In this case nominal shear reinforcement of diameter 8mm stirrups at 200mm center spacing is entered.

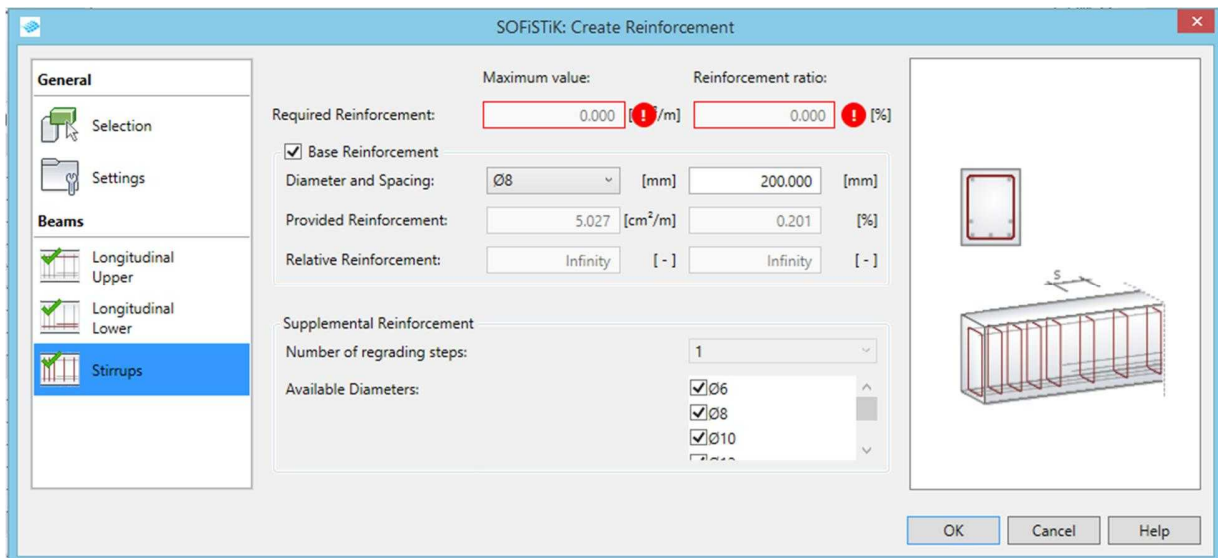


FIGURE 16: STEP 4 - MANUAL STIRRUP REINFORCEMENT INPUT

After confirming the inputs with 'OK' the Reinforcement Generation App will insert Revit rebar objects in the chosen members. Using a 3D view with medium level of detail and solid rebar display mode the generated reinforcement can be inspected.

Generation of Rebar Objects Using Results from Analysis and Design

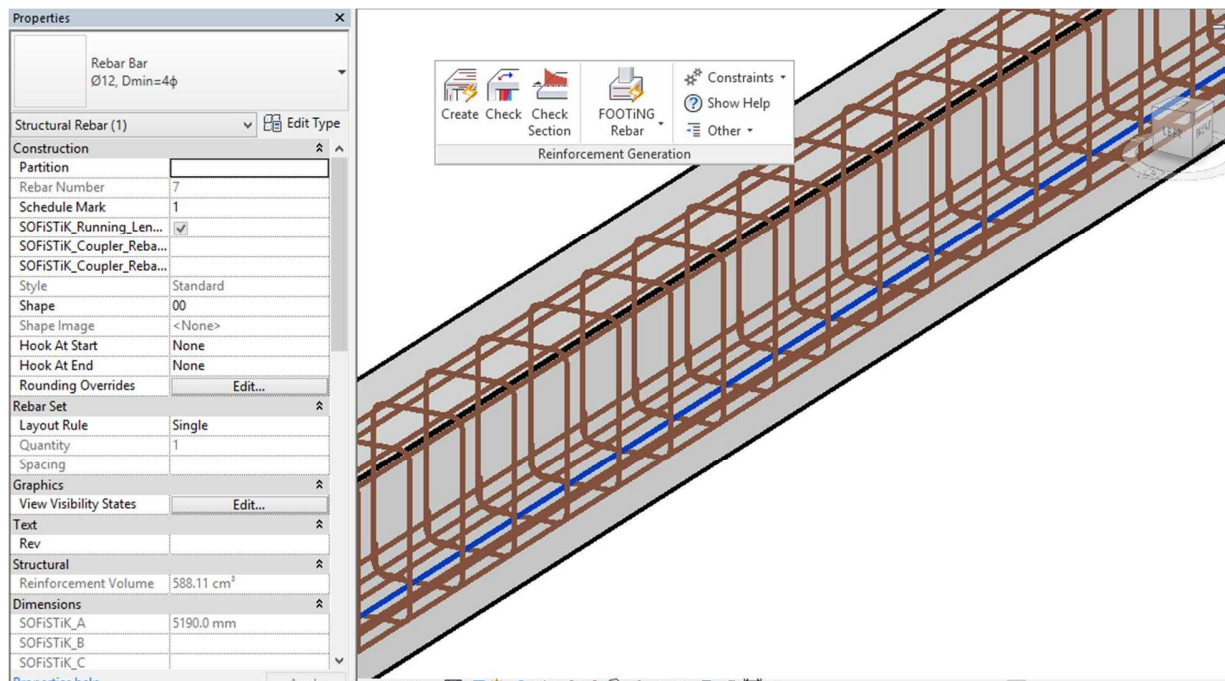


FIGURE 17: GENERATED REINFORCEMENT (MIDSPAN)

To further optimize the rebar layout several adjustments might be necessary, e.g. additional rebars might be placed manually. The Reinforcement Generation app displays required and provided reinforcement in a graphical representation which enables the user to easily verify whether the requirements have been fulfilled. The user might use the 'Check' button and select objects again in order to obtain the results shown below.

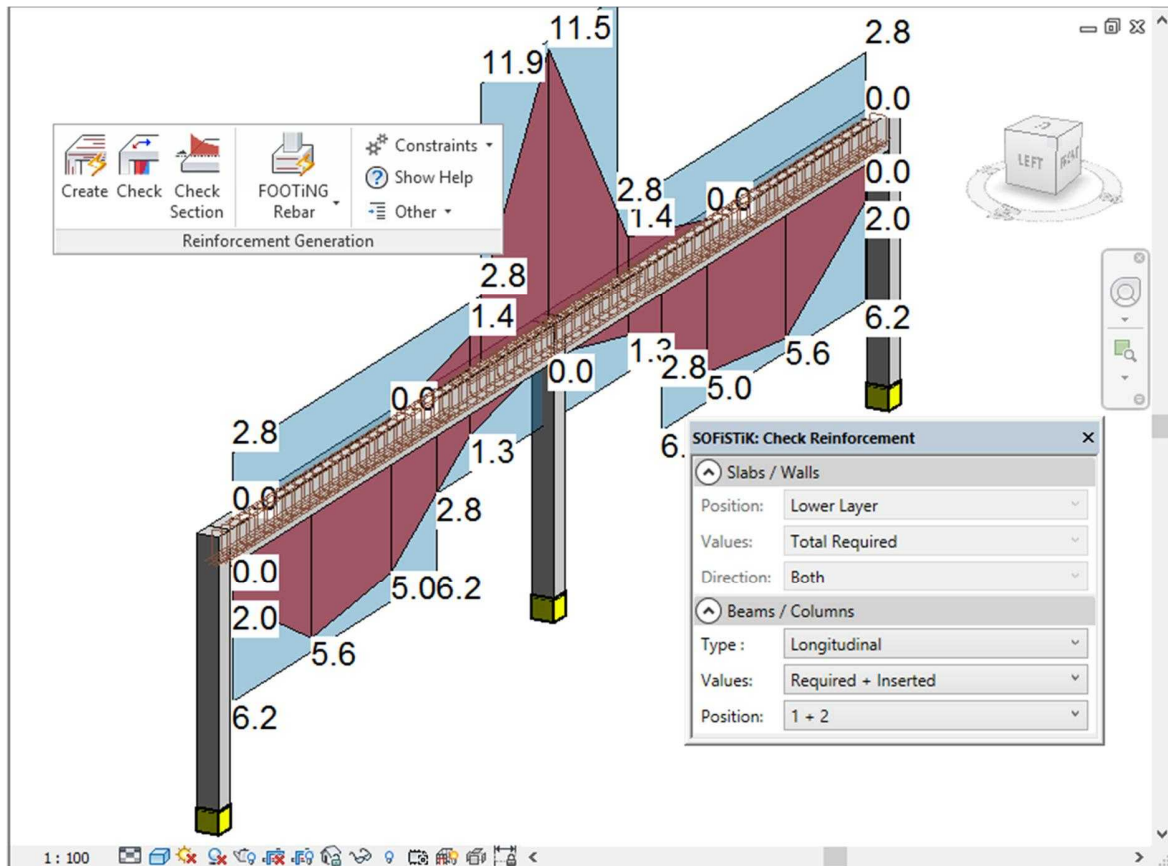


FIGURE 18: FINAL STEP - CHECK REINFORCEMENT (BLUE: PROVIDED)

Generation of Rebar Objects in Revit Structure

Input Data

Essentially, we require the architectural model (i.e. the concrete layout) and the analytical model based on the architectural model for analysis and design. It is essential, that analysis and design has been started from an integrated model in Revit. Design results (e.g. required reinforcement), has to be stored in the Revit model as Result Sets.

Rules

The generation of the reinforcement bars is based on the calculated required reinforcement stored for the various structural members. The generation process is controlled by specific rules which are provided in a separate configuration file and which can be modified by the user. The rules are used to control design code requirements as well as user specific settings.

The procedure thereby follows generally the approach of rule-based 'Expert Systems' in which the knowledge of an expert required to accomplish some task is separated from the general program logic and encoded in a specific rule base.

As an example, the definition of the anchorage length in Eurocode EN 1992-1-1:2004 reads:

(2) The design value of the ultimate bond stress, f_{bd} , for ribbed bars may be taken as:

$$f_{bd} = 2,25 \eta_1 \eta_2 f_{ctd} \quad (8.2)$$

where:

f_{ctd} is the design value of concrete tensile strength according to 3.1.6 (2)P. Due to the increasing brittleness of higher strength concrete, $f_{ctk,0,05}$ should be limited here to the value for C60/75, unless it can be verified that the average bond strength increases above this limit

η_1 is a coefficient related to the quality of the bond condition and the position of the bar during concreting (see Figure 8.2):

$\eta_1 = 1,0$ when 'good' conditions are obtained and

$\eta_1 = 0,7$ for all other cases and for bars in structural elements built with slip-forms, unless it can be shown that 'good' bond conditions exist

η_2 is related to the bar diameter:

$\eta_2 = 1,0$ for $\phi \leq 32$ mm

$\eta_2 = (132 - \phi)/100$ for $\phi > 32$ mm

(2) The basic required anchorage length, $l_{b,rqd}$, for anchoring the force $A_s \sigma_{sd}$ in a straight bar assuming constant bond stress equal to f_{bd} follows from:

$$l_{b,rqd} = (\phi / 4) (\sigma_{sd} / f_{bd}) \quad (8.3)$$

Where σ_{sd} is the design stress of the bar at the position from where the anchorage is measured from.

Values for f_{bd} are given in 8.4.2.

These requirements may be defined in the SOFiSTiK rule base as (the '\$' precede comment lines):

\$ 8.4.2 Ultimate bond stress

$$f_{bd} = 2.25 * \eta_1 * \eta_2 * f_{ctd}$$

\$ 8.4.2. Bond conditions

$$\eta_1 = 0.7 ; \text{isBondGood} : \eta_1 = 1.0$$

$$\eta_2 = 1.0 ; d_{asl} > 0.032 : \eta_2 = (0.132 - d_{asl}) * 10$$

\$ 8.4.3 Basic anchorage length (based on total yield strength)

$$l_{b,rqd} = (d_{asl} / 4) * (f_{yd} / f_{bd})$$

Each of the rules in the configuration file consists of an IF ... THEN ... type expression which consists of a condition and an assignment which expresses the value of a variable as a function of other variables. A variable consists of its value and a valid domain (min/max), which can also be modified using arithmetic expressions. Most of the variables will be provided and set by the system during runtime, like for

example the concrete tensile strength f_{ctd} , which will be set within Revit from the material definitions, but it would also be possible to define new variables.

The rules can be entered in any order. The program sorts the rules internally depending on their mutual dependencies and evaluates them accordingly, provided that no circular dependencies are given. This 'declarative style' of programming allows to encode the logic in a more readable way, for example it allows to list rules in the same order they are given in the design code. This in turn makes it easier for users with no specific programming background – which is usually the case with the design engineer - to control and modify the settings.

User Preferences

Many engineers (and experienced draftsmen) follow rules of thumb like “main reinforcement in beams with height > 0.4m is at least of diameter 16mm” or “in case main reinforcement in beams is a certain diameter (i.e. 16mm), supplementary bars should not differ more than two values from main reinforcement (i.e. 12mm, 14mm, 16mm, 20mm or 25mm only). Such kind of settings could also be defined in the rule base, for example:

```
d_asl = [0.010, 0.012, 0.014, 0.016, 0.020, 0.024, 0.028, 0.032]
```

```
Is_Beam {  
  d_asl <= 0.028  
  Section_Height >= 0.40 : d_asl >= 0.016  
  Section_Height <= 0.60 : d_asl >= 0.020  
}
```

After defining the allowed values for the diameter of the longitudinal bars 'd_asl' the range of diameters is restricted according to the section dimensions. The Variables 'Is_Beam' and 'Section_Height' are set during runtime by Revit.

Optimization

Good reinforcement design is always a compromise between several goals. For example, we want to have as little reinforcement mass as possible, but we also want to limit the variation of bars. These kinds of requirements can be controlled by weighting factors, which may be used to influence the outcome.

These factors can also be set within the rule-base. Currently are supported:

F1: try to use as less bars as possible (increase rather diameter than number)

F2: try to reduce difference between inserted and required reinforcement

F3: try to avoid using multiple layers of reinforcement (bending beams only)

The following sample will demonstrate how different settings lead to different results:

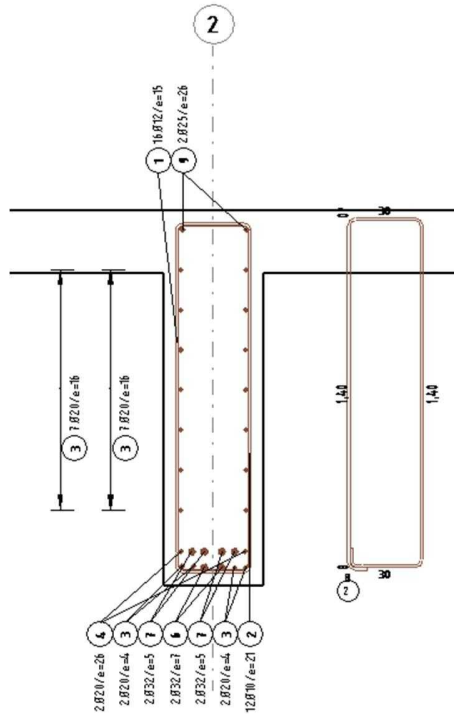


FIGURE 19: REINFORCEMENT GENERATED WITH STANDARD SETTINGS

We have a heavy two-span transfer T-beam supporting columns from three storeys above. Its section is 150cm * 40cm and the required maximum bottom reinforcement is 54,7cm² in the middle of the span. Over the support in the middle, 77,2cm² top reinforcement is required according to design.

With standard settings the result is as shown left for the in-span bottom reinforcement. The program generates 6 bars with diameter d=20mm and 6 bars with d=32mm, resulting in 67,1cm². Over the column we get 2 bars d=25mm and 11 bars d=28mm, resulting in 77,6cm², which is almost the theoretic optimum. So further optimization of the top reinforcement over the support is not necessary.

In a second attempt we increase the weighting factor F2 from 1 to 5. As expected the generated reinforcement in the field is reduced to 58,1cm² using 8 bars d=25mm and 6 bars d=20mm. The number of the bars has been increased by one.

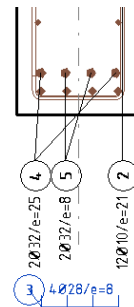


FIGURE 20: REINFORCEMENT GENERATED WITH MODIFIED SETTINGS

A third attempt is to help the software by choosing 2 bars d=28mm as basic reinforcement. This results in 4 bars d=28mm and 4 bars d=32mm with an area of reinforcement of 56,8cm² (picture on the right).

At present the development process is not fully finalized. Further investigation into optimization rules and weighting is in process.

Basic Reinforcement

During the development of the software it became obvious that in many cases users want to have a direct influence, in particular, on the choice of the basic reinforcement. Therefore a set of dialogs has been created, in which users can see the basic reinforcement proposed by the software and in which it can be modified. All reinforcement except this basic reinforcement will be created automatically.

Figure 20

Generation of Rebar Objects Using Results from Analysis and Design

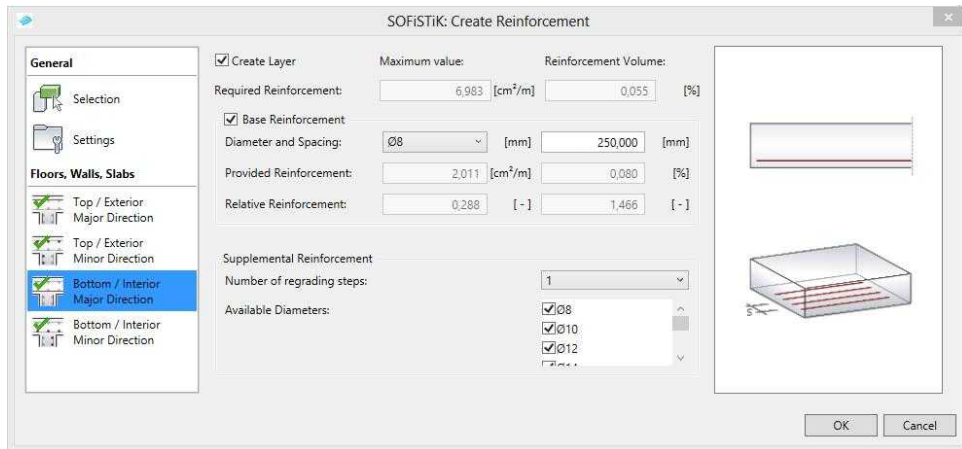


FIGURE 21: SETTINGS FOR SLABS

Check Proposed Reinforcement against Required Reinforcement

Reinforcement values

Obviously the resulting reinforcement layout has to be checked properly. Users may want to see

- Reinforcement required, i.e. design results
- Reinforcement distributed in area or area by length, i.e. proposed reinforcement
- Difference between the values above.

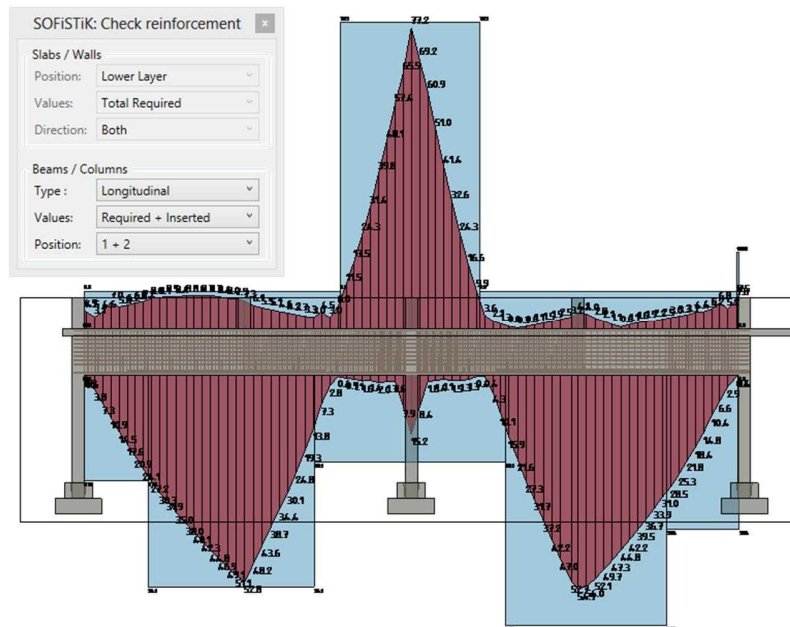


FIGURE 22: CHECKING LONGITUDINAL REINFORCEMENT FOR BEAMS

For beams and slabs a presentation showing required reinforcement is common and well arranged. For slabs a presentation as scaled crosses has the advantage, that values of two directions and the angle of the reinforcement can be displayed in a single view.

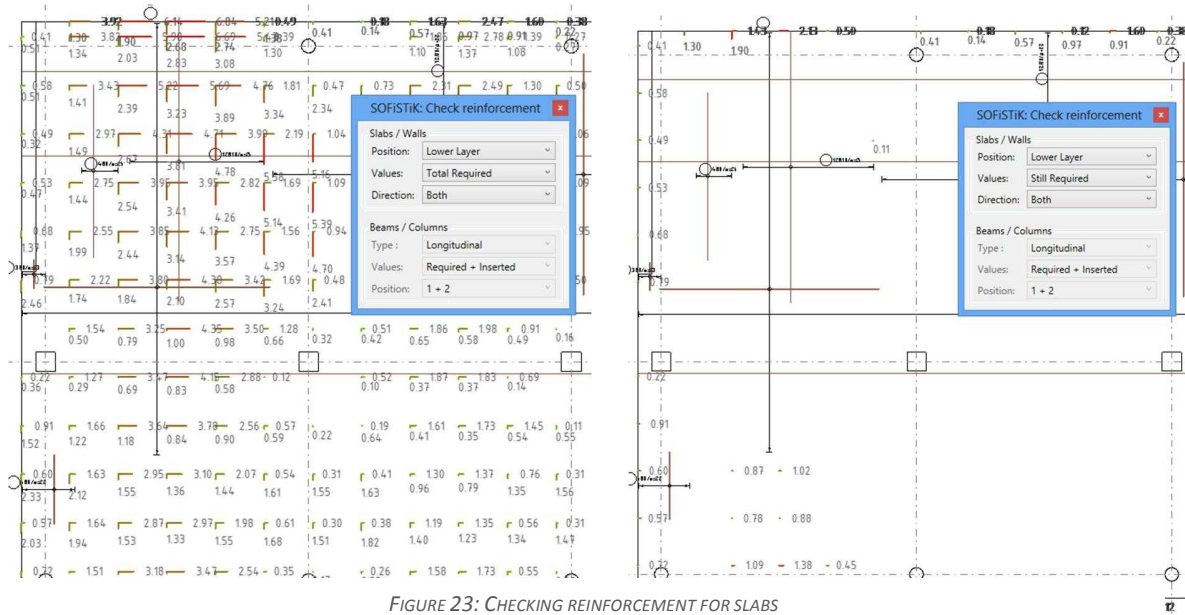


FIGURE 23: CHECKING REINFORCEMENT FOR SLABS

As an alternative there is the possibility to draw model lines which cut the members. Reinforcement values perpendicular to this line can be displayed similar to beams and columns.

Bar Marks, Annotations, Schedules and Shop Drawings

It is very difficult if not impossible to produce proper shop drawings with pure Revit Structure. SOFiSTiK Reinforcement Detailing is a software developed to accelerate and simplify this process. It can be downloaded from Autodesk Exchange Apps. The app is very well integrated into Revit Structure and rely on the use of detailing and annotation families which can be modified by the users to follow local or company standards.

Bar marks

In most countries bar marks are numbered per drawing sheet, e.g. bar mark #1 on sheet 1 is generally different to bar mark #1 on any other sheet. Consequently rebar schedules are produced for each single sheet separately. To achieve this behavior, each rebar has to be assigned to a sheet. In Revit a sheet is composed out of several views, which can only be on one single sheet. Therefore all visible rebars on a certain sheet can be assigned to this sheet with one simple command. In case there are rebars visible which are already assigned to another sheet, a warning will be given and the user has the choice to disregard this rebar for the current assignment.

After assigning rebars to a sheet, bar marks can be created with a simple command.

Running Length

In many cases it is desired to cut rebars on site according to the formwork. These rebars can be marked as “Running Length”. Regardless of their length in the model, there will be one rebar in the schedule per diameter only. The length is the added length of all rebars marked as running length with a certain diameter.

Filters to display top and bottom layer separately

In many cases it is desired to display top and bottom reinforcement of slabs in two different views. To achieve this, a parameter has been introduced which indicates the layer of each reinforcement element. This parameter can be set by the user or automatically within the software. SOFiSTiK Reinforcement Generation fills this parameter automatically, whereas in Reinforcement Detailing we have a tool, which determines the layer and writes this information into the parameter.

With Revit Filters this parameter can be used to control the visibility of each rebar according to the content of the parameter.

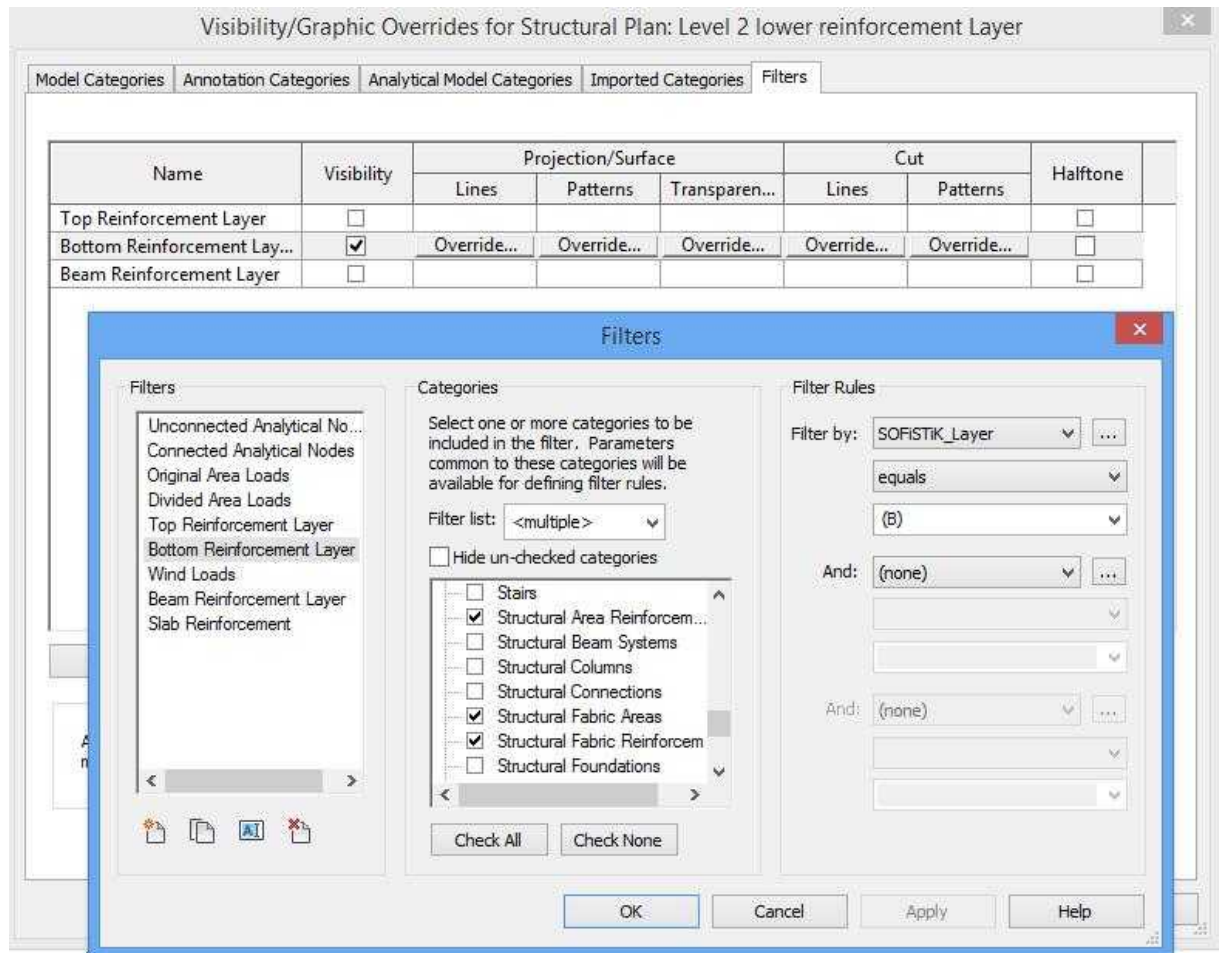


FIGURE 24: FILTER RULES TO DISPLAY BOTTOM REINFORCEMENT ONLY

Adjust, Stretch, Split and Browse

SOFiSTiK Reinforcement Detailing includes a set of tools which enhance productivity and complement the functionality of Revit Structure.

A rebar set can be exploded, the resulting rebars may be stretched to the cover of a suitable face of the rebar host.

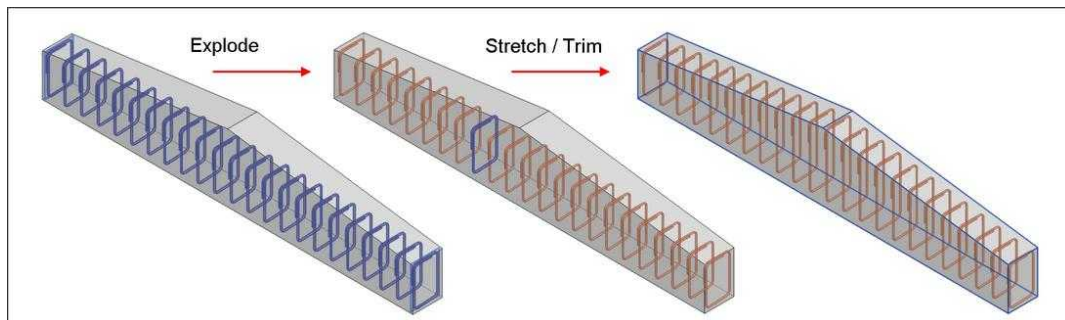


FIGURE 25: REINFORCEMENT OF HAUNCHED BEAM

In some cases rebars should not be perpendicular to the driving line of the set. For this there is an easy way to rotate a set of single bars around the definition line of the set.

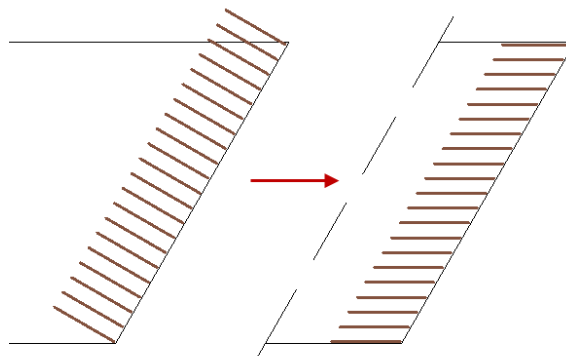


FIGURE 25: REINFORCEMENT SET TURNED PARALLEL TO AN EDGE

Revit can create straight rebars of any length. Normally the lengths are arranged and grouped in a practical manner for transportation and site placement. There is a command which splits rebar sets using different rules. A staggered split pattern provides laps in rebars in different positions.

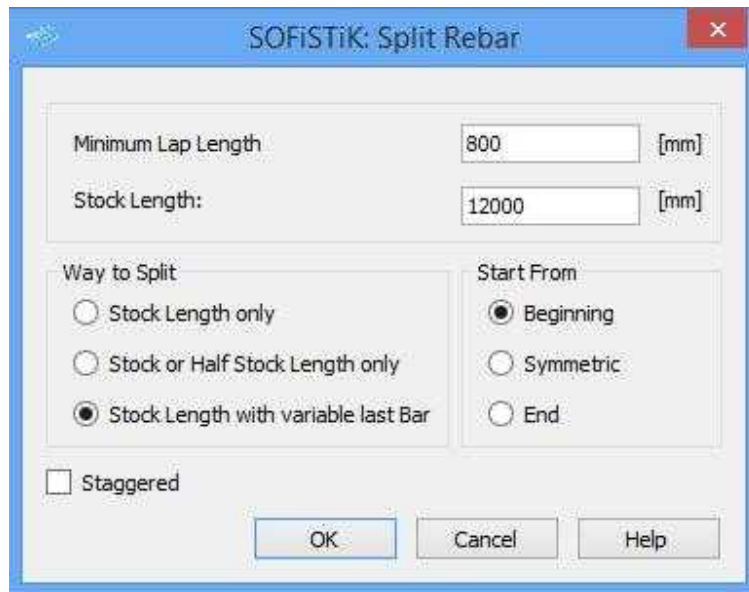


FIGURE 26: DIALOG TO CONTROL SPLITTING OF REBARS

Visibility of rebars (view unobscured or view solid) can be defined in Revit for each view. To change this for all rebars in a view can be very time consuming. To assist with this there is a tool helping users to set this behavior for all rebars in active or all views in one step.

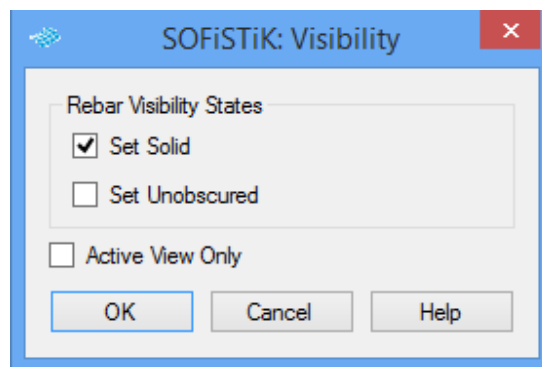


FIGURE 27: SET VISIBILITY OF ALL REBARS WITH ONE COMMAND

A useful tool for checking bar marks and assignment of rebars to schedules is the command 'Browse'. All instances of a selected bar mark or of all bar marks assigned to a certain sheet are displayed in red. In addition all rebars not assigned to a sheet can be marked with this tool.

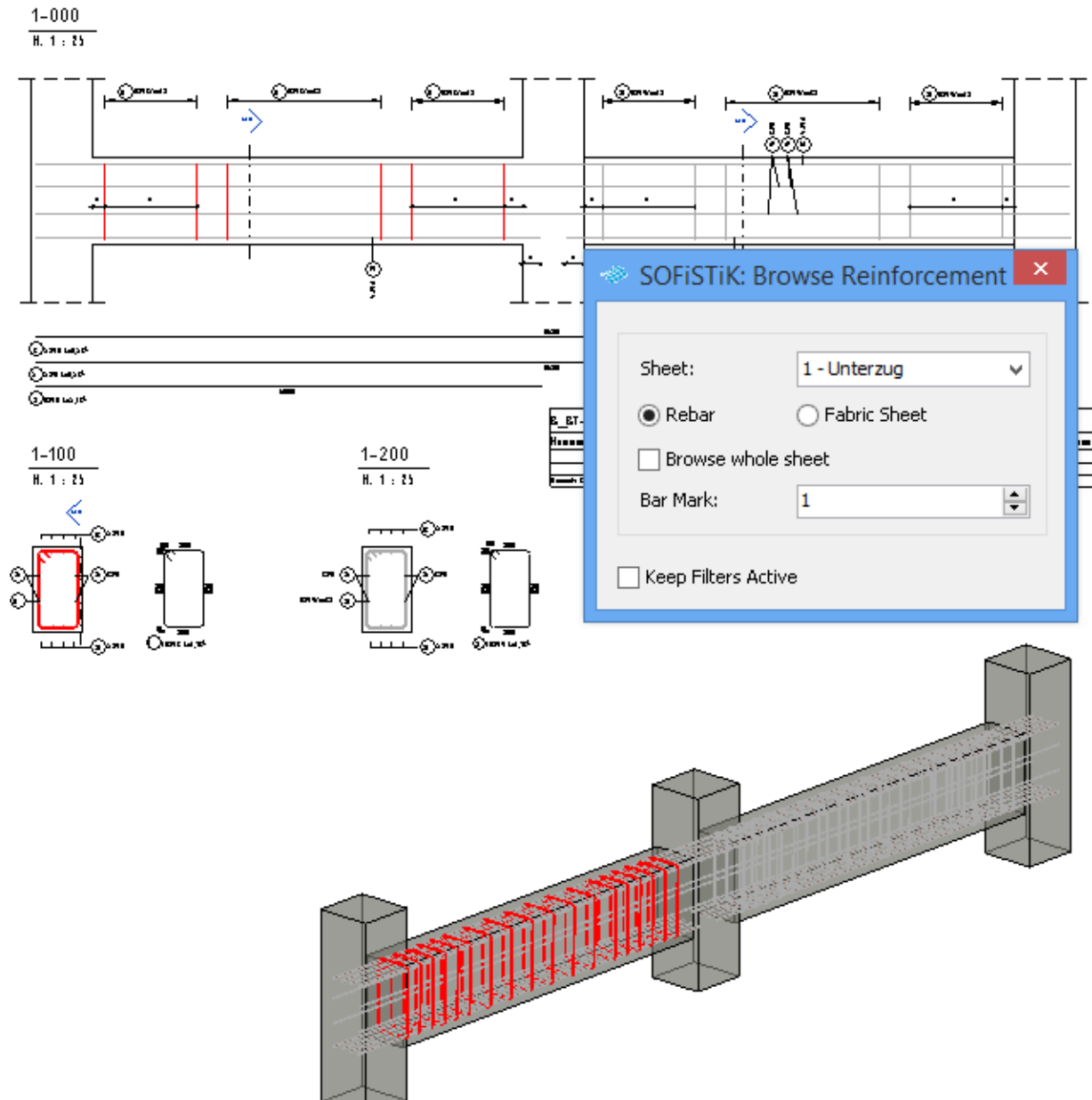


FIGURE 28: BROWSE REBAR FOR CHECKING PURPOSE

Annotation of Rebars

Around the world, the annotation of rebars is treated differently. Annotation of rebars in sections are normally placed outside the section whilst annotations of slab and wall reinforcement can be inside. SOFiSTiK Reinforcement Detailing attempts to automate this process in a way that all annotations are clearly readable.

Rebar Tags

Rebar sets with a single rebar are annotated with an annotation object with a leader line. For sets with more rebars a combination of a dimension element and an annotation object are used. To achieve proper placement of annotations, SOFiSTiK Reinforcement Detailing uses the offset of the bounding box of all selected members.

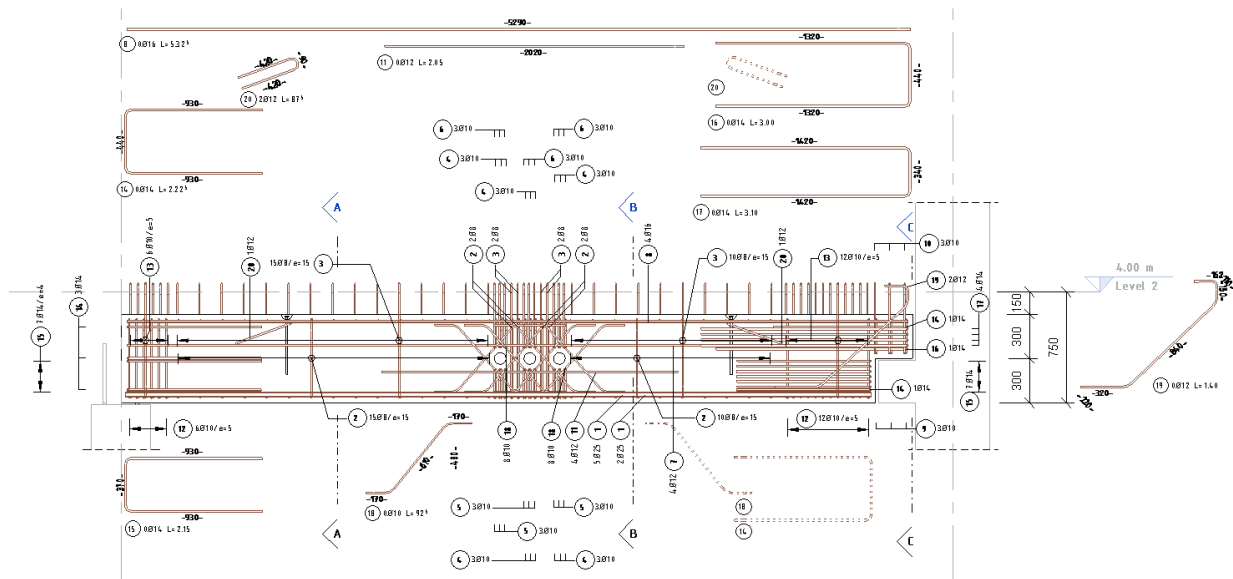


FIGURE 29: TAGGING AND REBAR DETAILS

Hide and Tag

For clarity, not all rebars of a rebar set are normally visible on a shop drawing. For this, 3 different commands are available:

- First is “Hide” which makes the original rebar set invisible on the sheet and inserts a detail element displaying the rebars which shall be visible. This functionality has been introduced by Revit 2015.
- Second is “Hide and Tag”, which hides all rebars but one and annotates this rebar with a distribution line.
- Third is “Hide and Detail” which additionally displays the bar shape turned into the drawing plane.

In all three cases the detail elements are automatically updated in case the rebar set has been changed.

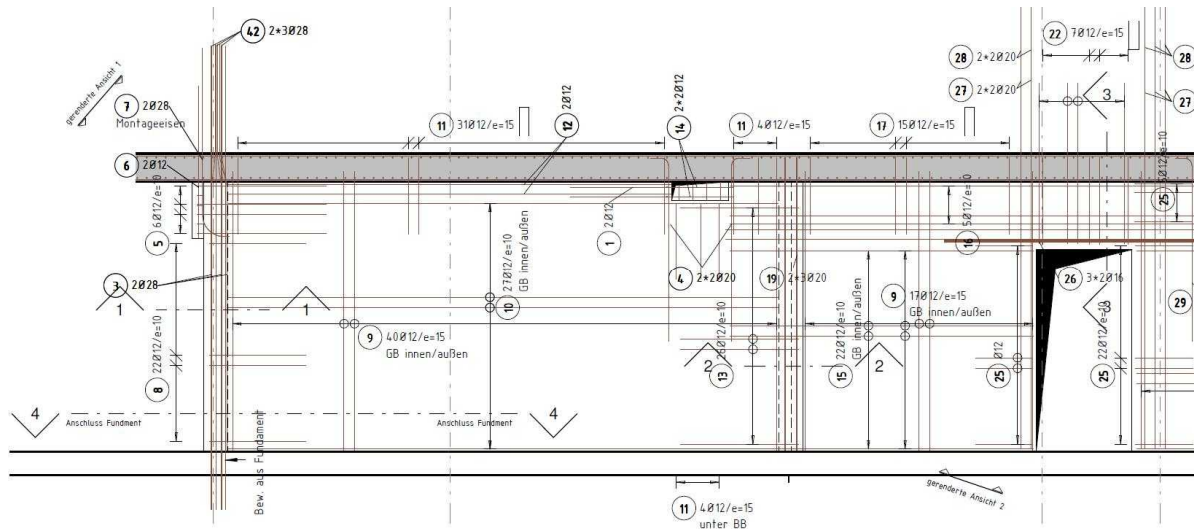


FIGURE 30: WALL WITH HIDDEN REBARS

Enhanced Features with SOFiSTiK FEA Extensions

Covering most features of Autodesk Robot Structural Analysis, equivalent tools from SOFiSTiK provide features which are missing in Robot:

- There is a way to define properties of edges of slabs and walls like linear hinges directly in Revit to be used by the FEA software. This allows easy definition of connections between walls and slabs without carrying moments.
- Transfer of required shear reinforcement of beams from design into Revit to generate and check stirrups.

Summary

Using Autodesk Revit Structure, Autodesk Robot Structural Analysis together with Apps from SOFiSTiK, the complete workflow of analysis, design and detailing of concrete structures is supported. All relevant data can be stored within a Revit project. Compared to traditional workflows, which rely on transferring data from application to application, big improvements can be achieved regarding performance and quality.