AUTODESK UNIVERSITY

MSF125329

Introduction to Structural Precast for Revit

Tomasz Fudala - Technical Marketing Manager for Structure
Autodesk

Bogdan Teodorescu - Principal Product Owner
Autodesk

Learning Objectives

- Learn how to set up the environment
- Learn how to split floors and walls according to user-defined rules
- Learn how to reinforce slabs and walls according to user-defined rules
- Learn about shop drawings and CNC generation

Description

Prefabrication can lead to reduced project schedules, greater productivity, and better on-site safety records. Structural Precast for Revit 2018 software supports multiple stages of the precast workflow. The purpose of this class is to reveal the design workflow for precast elements (walls and slabs) using advanced tools that automate the process of generating precast assemblies, shop drawings, and computer numerical control (CNC) files.

Tomasz Fudala

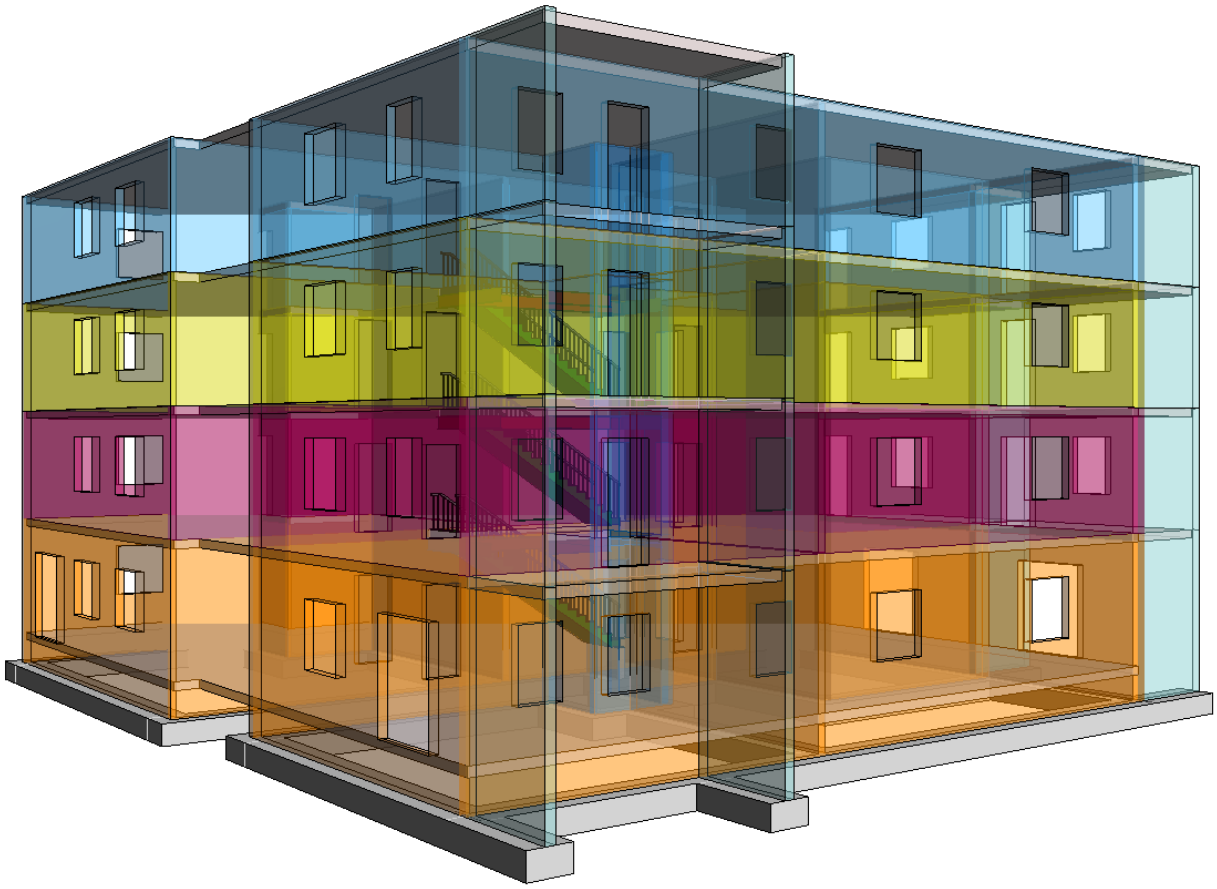
Tomasz Fudala is the Technical Marketing Manager for Structure at Autodesk. He has over 14 years of experience in the software industry and a comprehensive background and vast knowledge of structural solutions in the Autodesk portfolio. He achieved a Master of Science degree in Structural Engineering from the Cracow University of Technology, Poland. Find him on Twitter @tomekf

Bogdan Teodorescu

Bogdan Teodorescu is the liaison to the network of partners working on Revit Structure software and will be responsible for improving the Revit software integration in the precast sector. He graduated from Technical University of Civil Engineering of Bucharest (UTCB), and he has a master's degree in structural analysis for steel structures from Imperial College London. He has accumulated significant experience in designing steel and precast buildings, and he also held a teaching assistant position at UTCB.

Structural Precast for Revit

Autodesk® Structural Precast Extension for Revit® is a BIM-centric offering for modeling and detailing precast elements that improves productivity and precision for engineers, detailers and fabricators working on typical building projects in the precast industry. As an app for Revit, Structural Precast for Revit provides Revit users access to powerful tools for automatic rule-based segmentation, reinforcement, shop drawings and CAM files generation of precast planar concrete elements. The product is mostly suitable for typical building projects, made up of slabs and walls produced in factories with a high level of automation.



Key benefits and business values

BIM-centric precast concrete solution helping our users in three main ways:

Better connect structural design to precast fabrication

- Improve collaboration working within a BIM environment
- Streamlines coordination of structural, architectural and MEP designs by using Revit as a single platform
- Help accelerate the detailing process by integrating precast detailing expertise alongside design

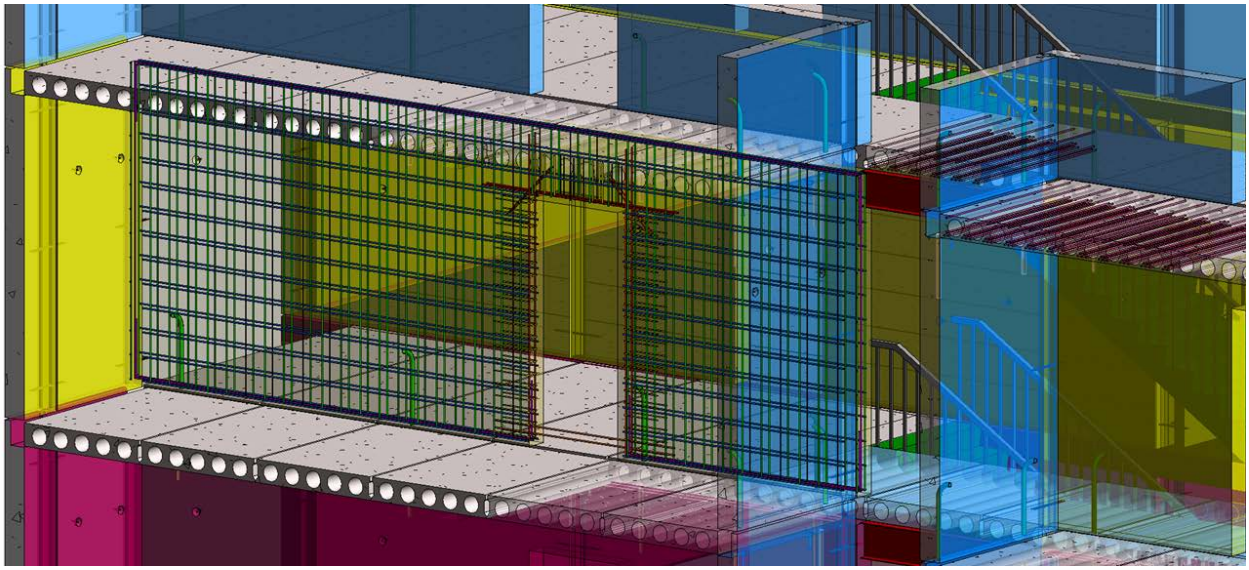
Streamline detailing to fabrication with model-based deliverables

- Better estimate quantities and cost at early bidding and execution stages
- Help improve accuracy of precast deliverables and help speed time to fabrication using CAM files
- Use various LOD levels to express the main element's characteristics or go deeper into details like segmentation, main and secondary reinforcement

Extend fabrication to the field to minimize waste

- Share models with contractors for use in the field
- Perform project review sessions of the 3D models
- Extend use of model for the building lifecycle

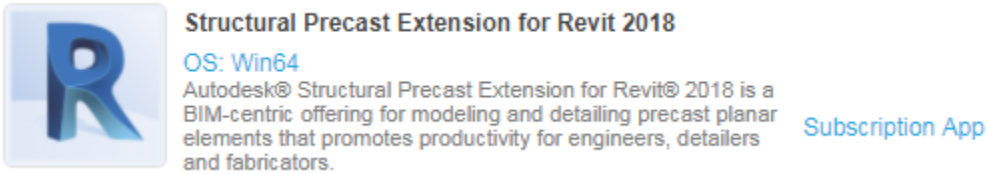
This approach for precast concrete results in creating relevant, more accurate and up to date documentation, it also drives the end-to-end workflows in a paperless way.



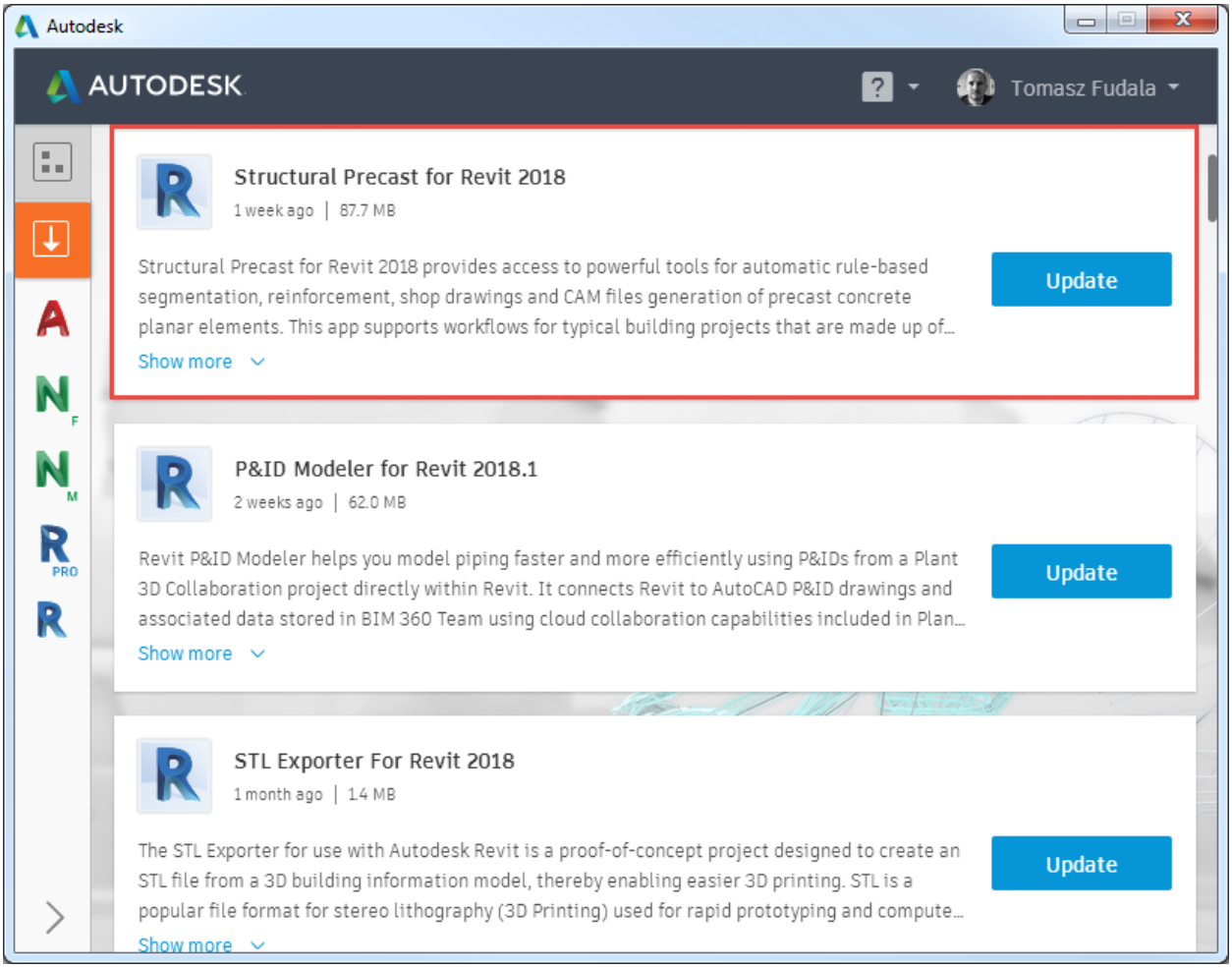
Installation & Configuration Settings

Autodesk Structural Precast Extension for Revit 2018 was released along with Revit 2018.1 release.

The extension must be download from the Autodesk App Store and then installed on the top of Revit 2018.1

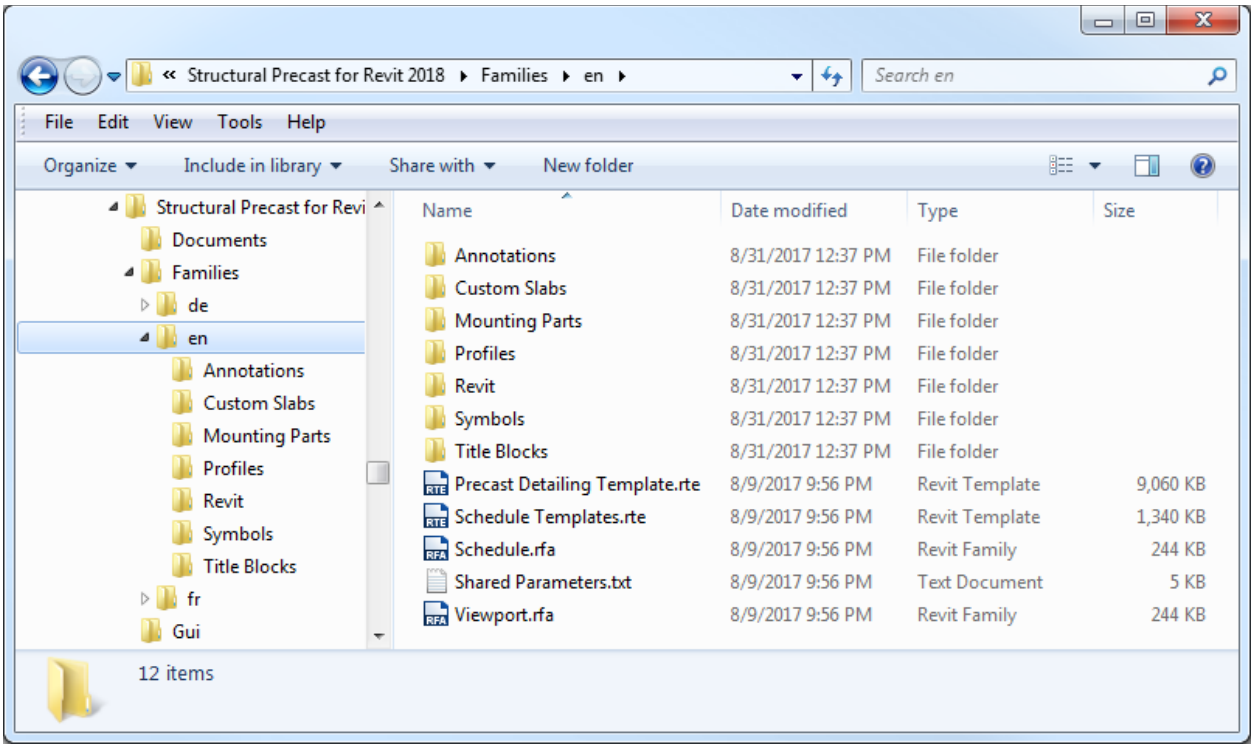


…or you can also access the installer from within the Autodesk Desktop App:



At this moment, the extension is available in English, French and German languages.

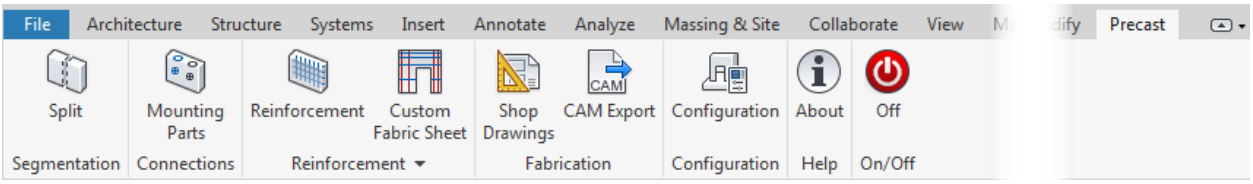
Once installed, several Revit families and templates are placed in a dedicated location… *C:\ProgramData\Autodesk\Structural Precast for Revit 2018\Families\.*



The installed files include:

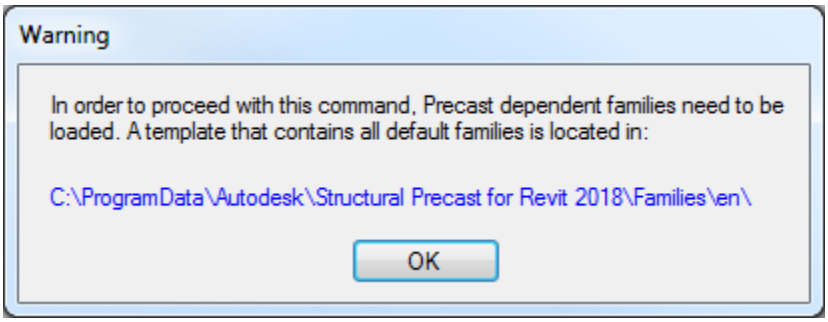
- Annotations
- Custom slabs (sample slab families)
- Mounting parts (lifters and other connection parts)
- Profiles
- Revit (rebar shapes)
- Symbols
- Title blocks

Here is what the Precast tab looks like once the extension is installed.



You must be wondering what the On/Off button does, an answer is pretty straight forward as it enables or disables the extension. You can use this tool to turn the extension off to replenish hardware resources and improve Revit performance if there is such need.

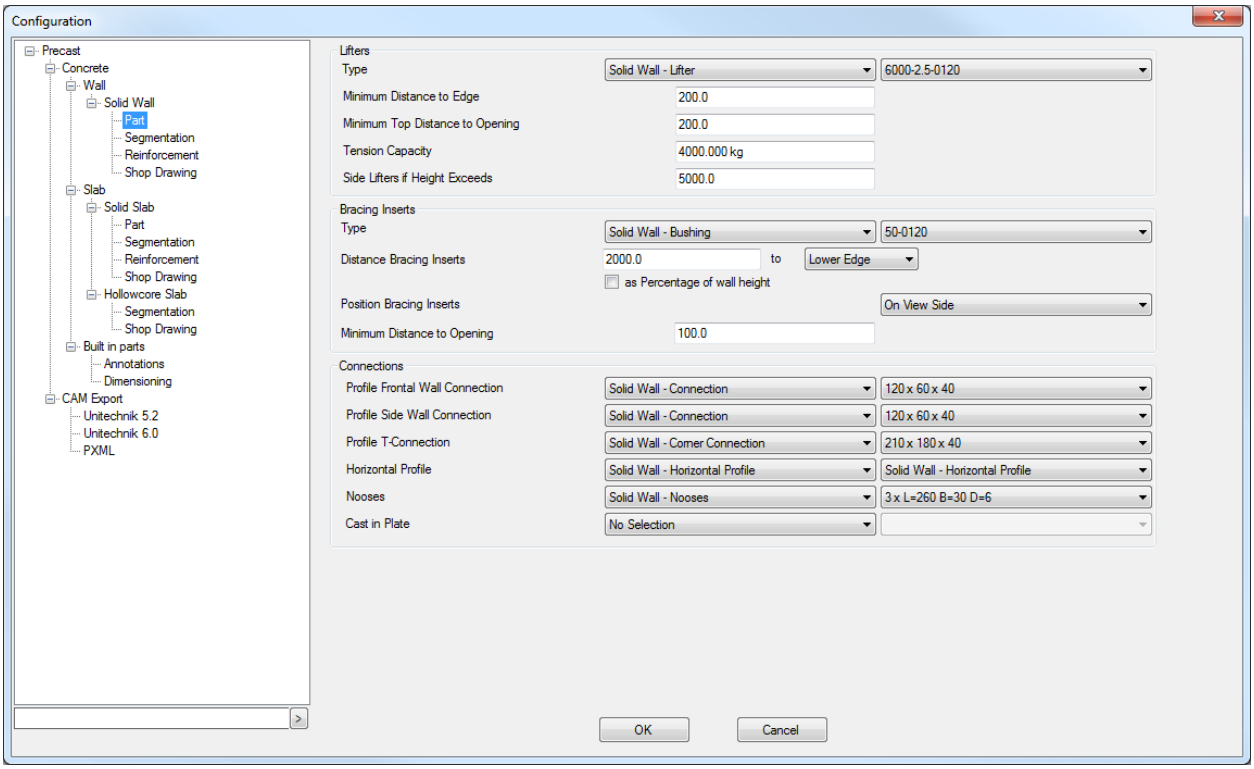
When you attempt to run the first command, you are presented with the following warning:



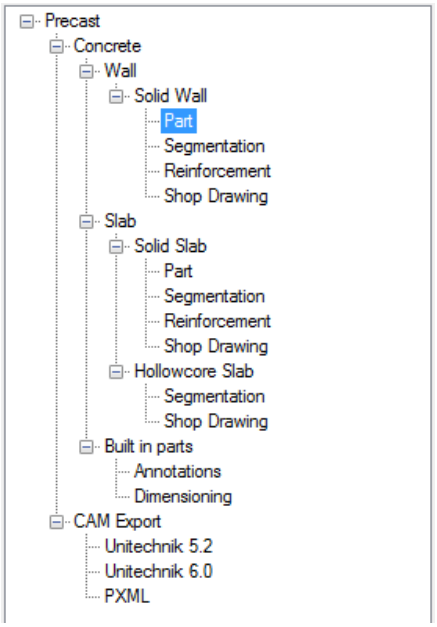
This is basically saying some content needs to be loaded for this extension to work properly.

That's why it is a good practice to start working on a new project running the Configuration tool first.

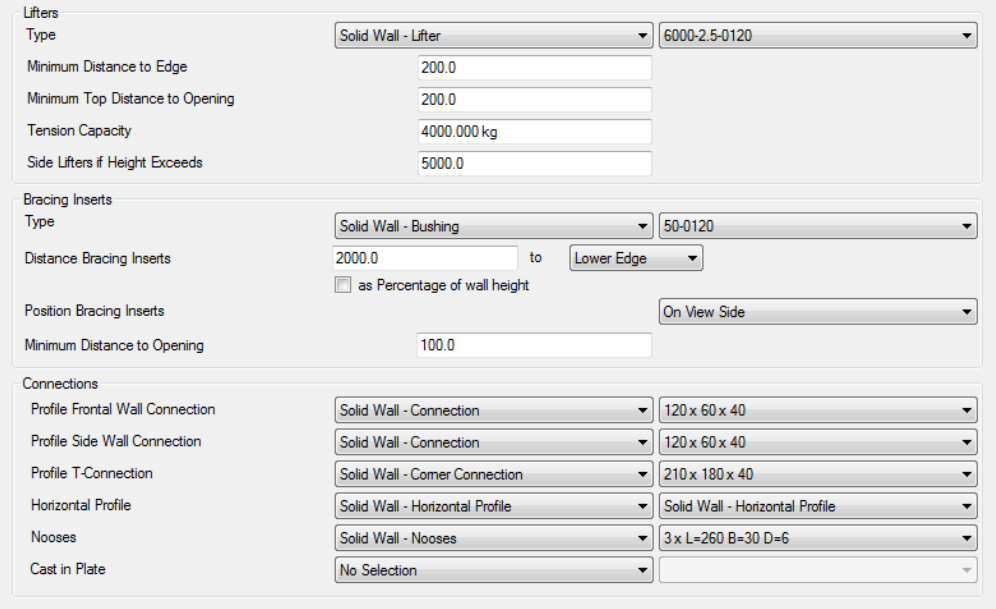
Before you start using Structural Precast Extension for Revit you should configure these settings for walls, slabs and built in parts. The settings include position numbering, part information, segmentation rules, reinforcement definition, and project drawing defaults for walls and slabs, as well as settings for annotation and dimensioning in parts.



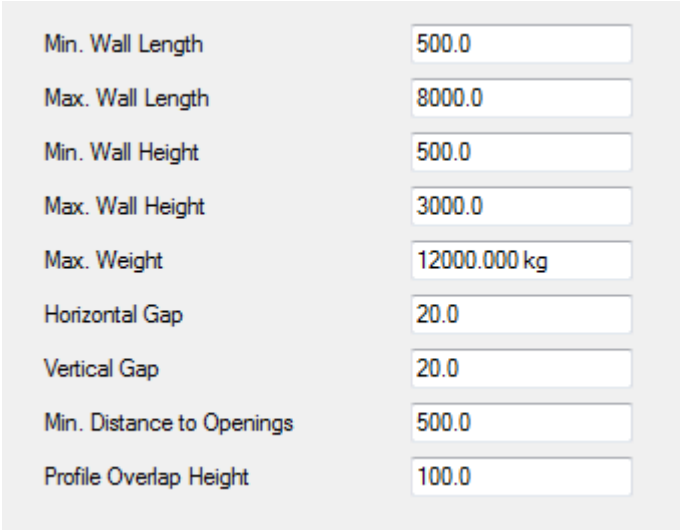
Once you open this dialog and click OK all necessary Revit families and templates will be loaded to the current project automatically and the warning message will not be bothering you anymore. The Configuration dialog is organized in a tree view, each node contains options for precast concrete assemblies.



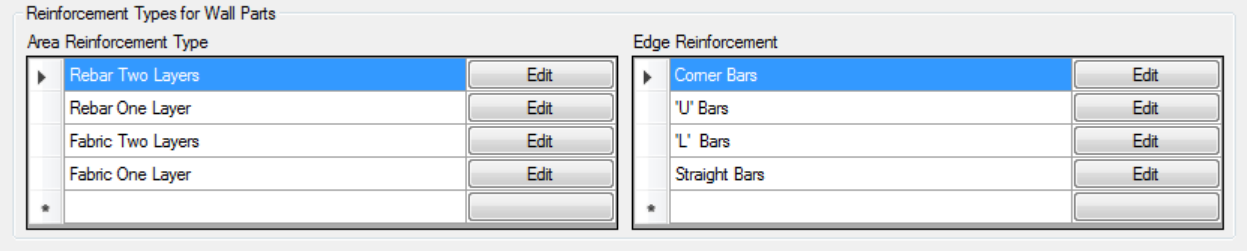
For example the wall node configures settings for solid structural walls. These settings include wall part information about lifters, bracings and connectors:



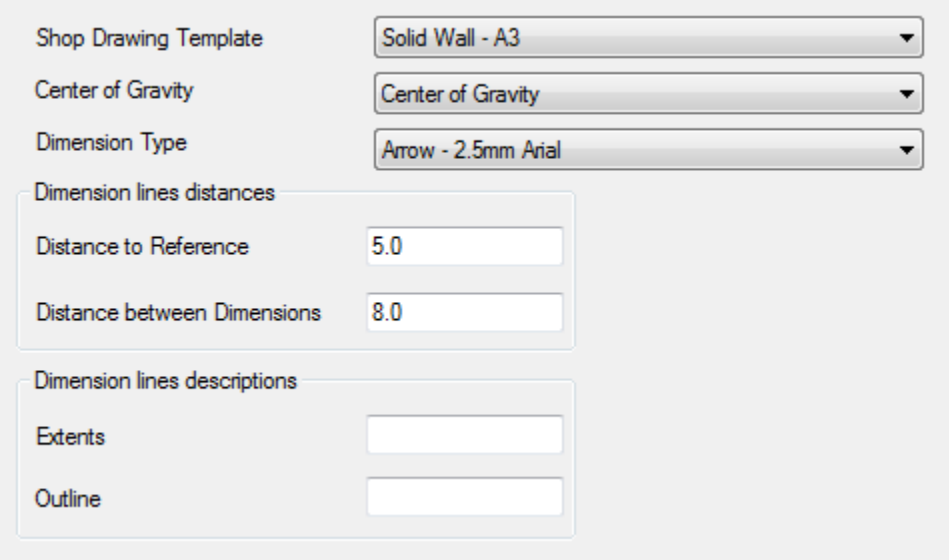
…segmentation rules:



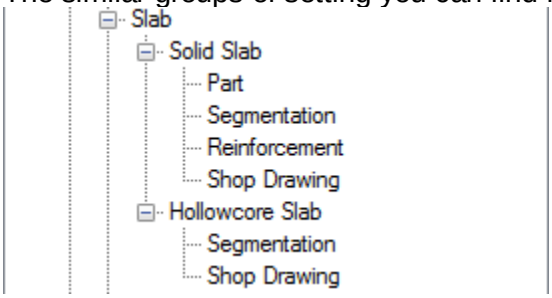
…reinforcement definition:



…and project shop drawing defaults:



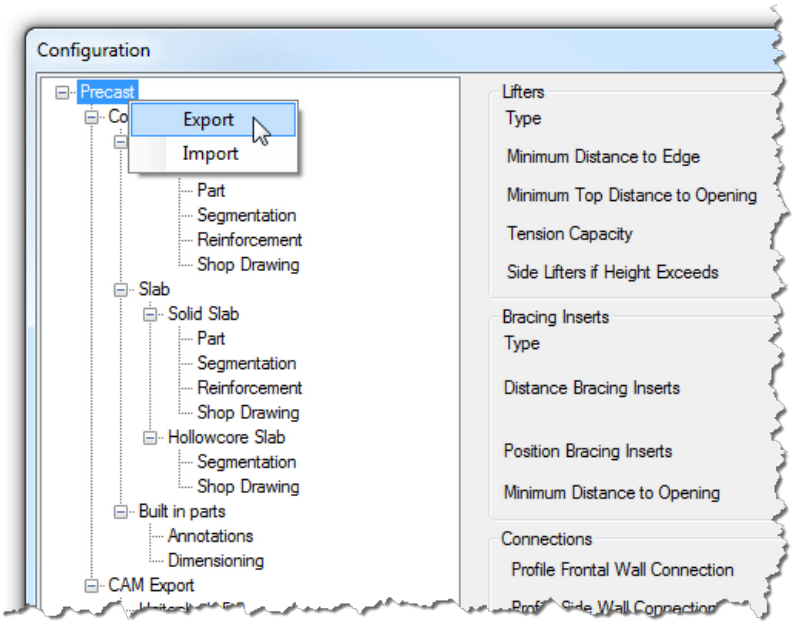
The similar groups of setting you can find for slabs (solid slabs and hollowcore slabs):



The Built in parts node configures settings for annotation and dimensioning in parts and the CAM Export node configures additional data specific to each CAM file type.

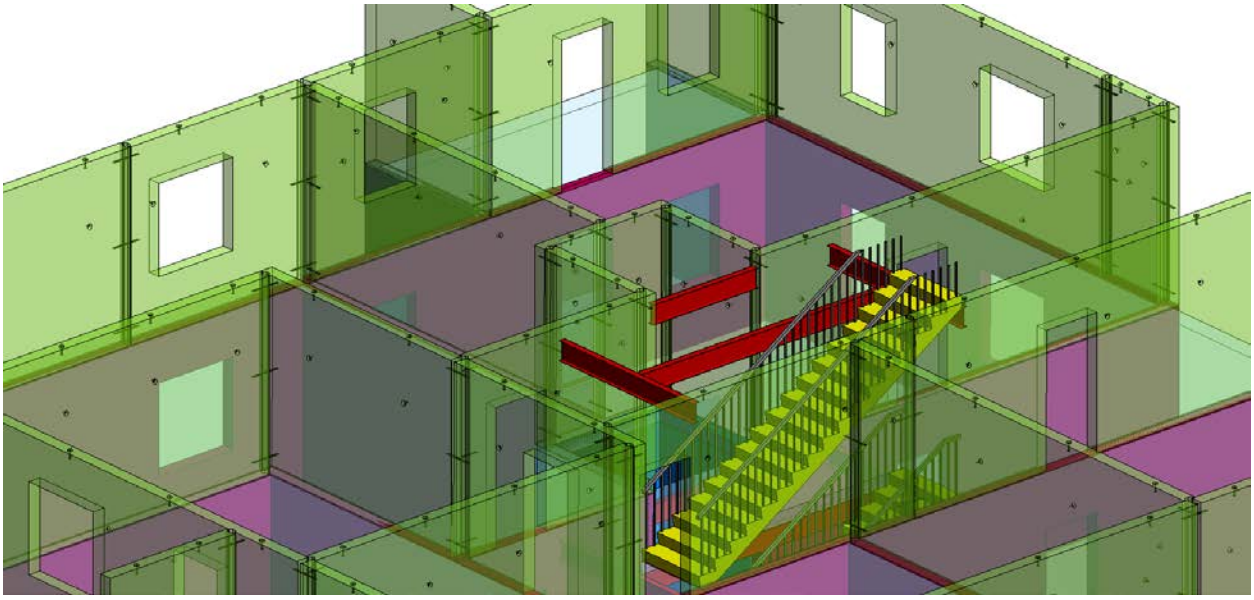
All these Precast settings are project related but you can maintain consistent configuration settings across similar projects by exporting them to an XML file which can later be imported into other projects. This gives you the ability to create different configurations for different manufacturers or sites.

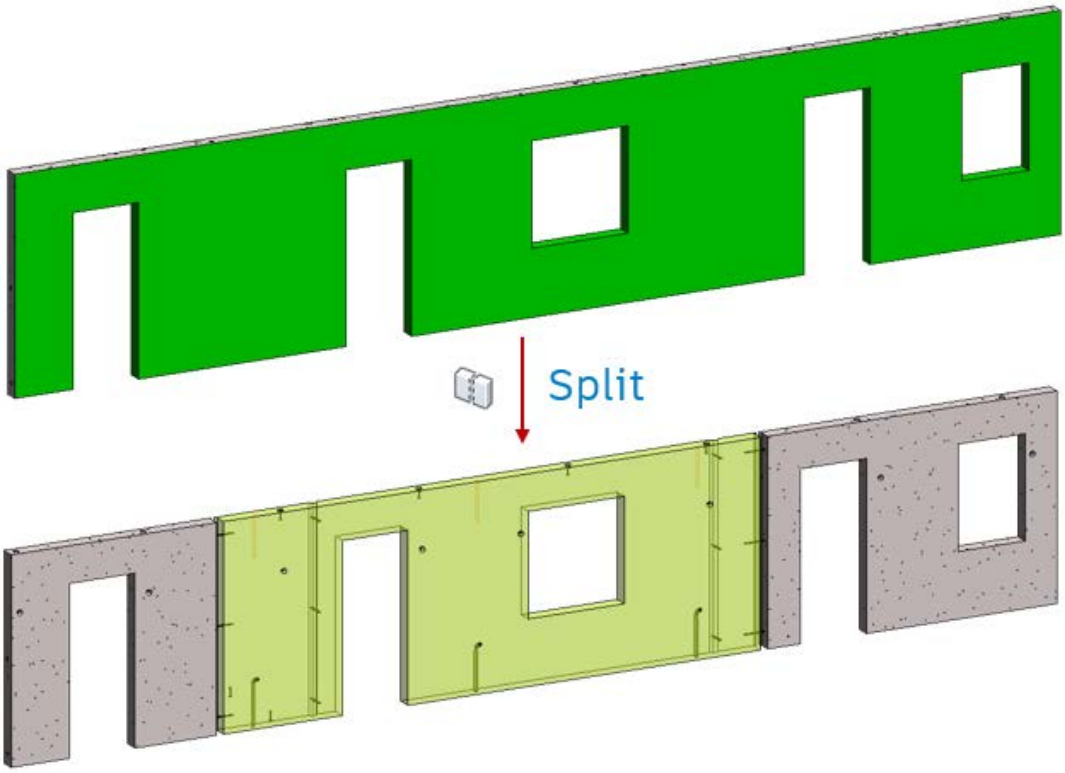
To export/import your current configuration file just right click in the tree view and select Export/Import from the context menu.

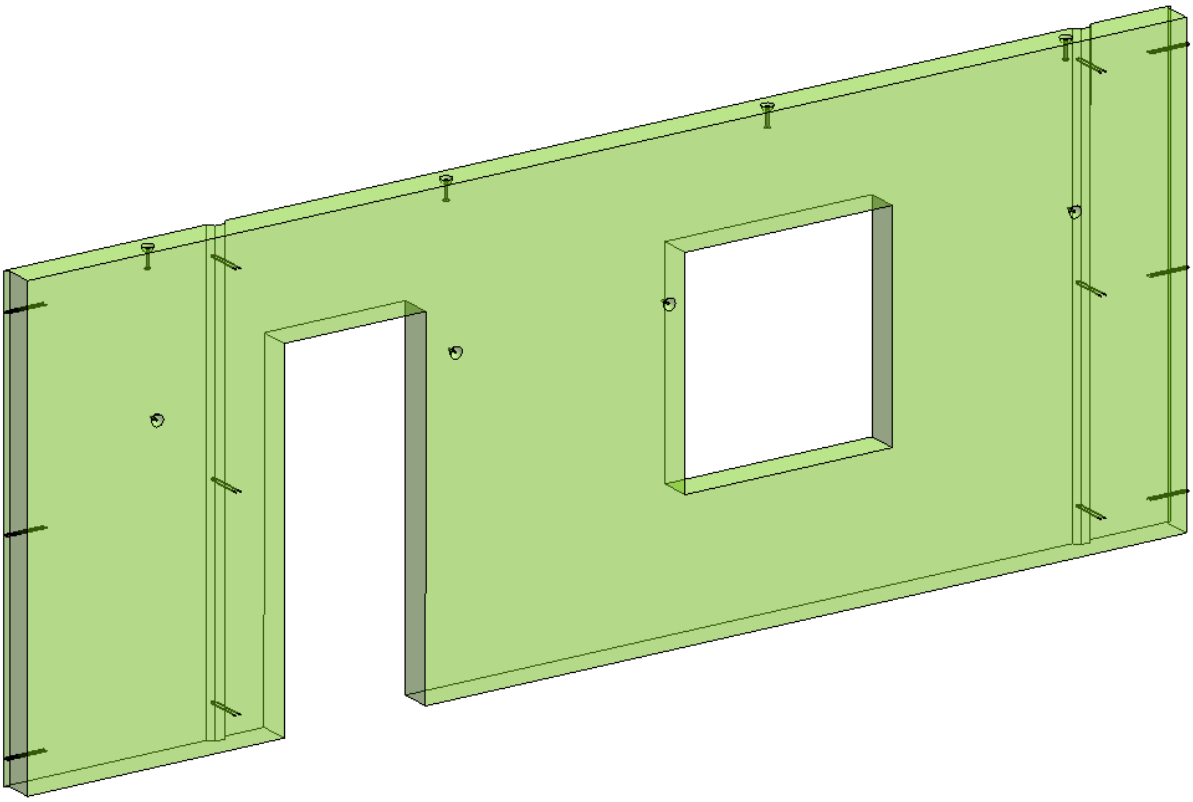

Configuration
Precast
Export
Import
Part
Segmentation
Reinforcement
Shop Drawing
Slab
Solid Slab
Part
Segmentation
Reinforcement
Shop Drawing
Hollowcore Slab
Segmentation
Shop Drawing
Built in parts
Annotations
Dimensioning
CAM Export
Lifters
Type
Minimum Distance to Edge
Minimum Top Distance to Opening
Tension Capacity
Side Lifters if Height Exceeds
Bracing Inserts
Type
Distance Bracing Inserts
Position Bracing Inserts
Minimum Distance to Opening
Connections
Profile Frontal Wall Connection

Split walls and floors according to user-defined rules

In order to execute a segmentation of walls select all the walls and press Split command. During this command the precast wall assemblies are created. Lifters, bushings and connection are created according to the rules from your Configuration.

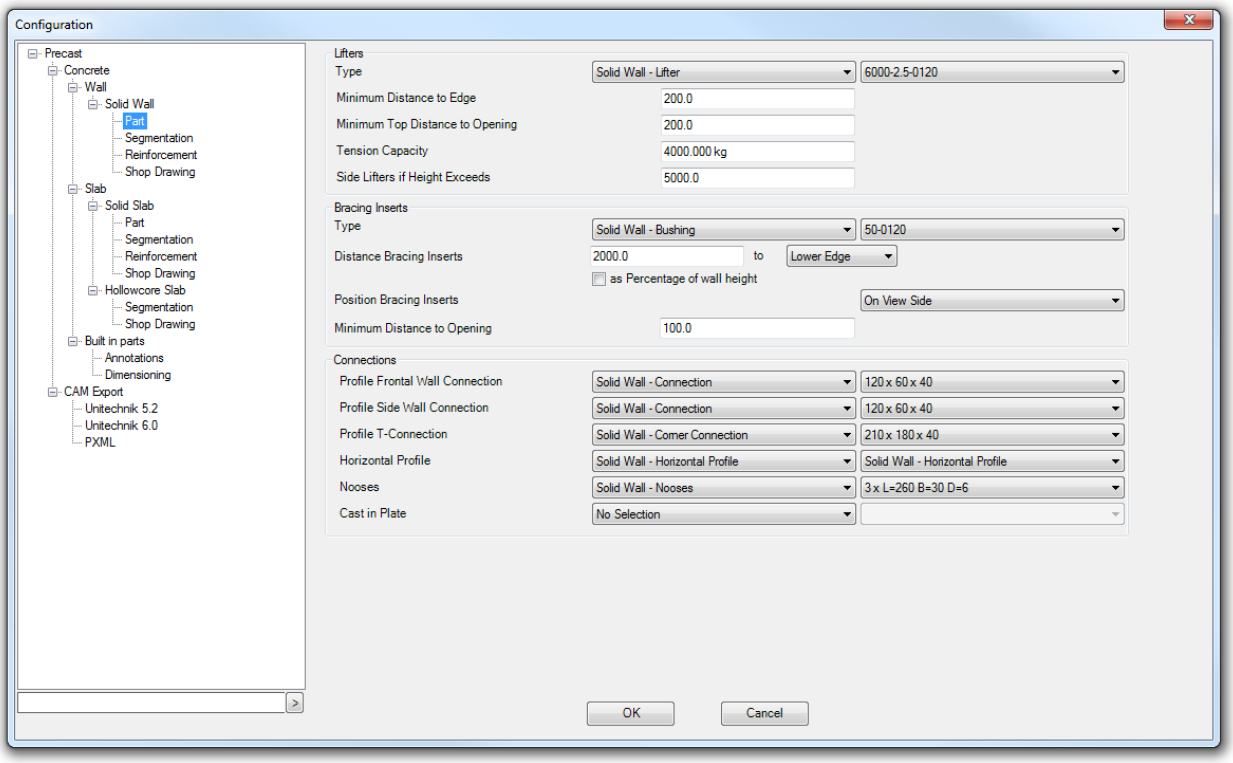




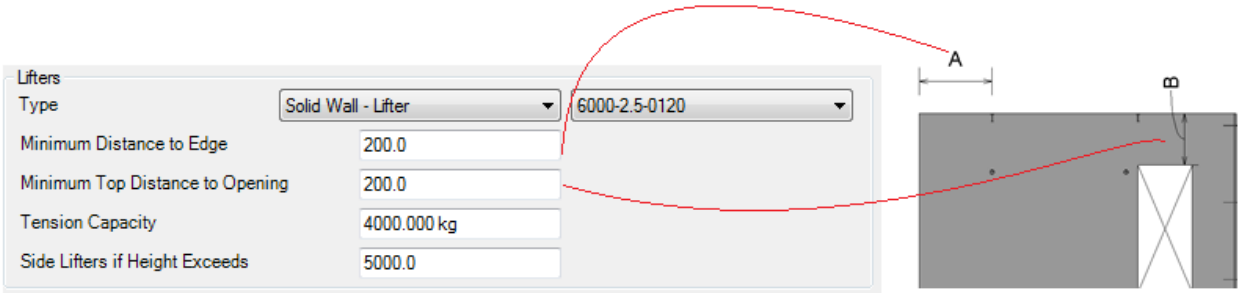


In order to change the result of the segmentation, you should go to Configuration and edit the desired fields according to your needs.

All the mounting parts (lifters, bushings, anchor plates) and connections can be defined in this section. For the desired element go to the Part tab.

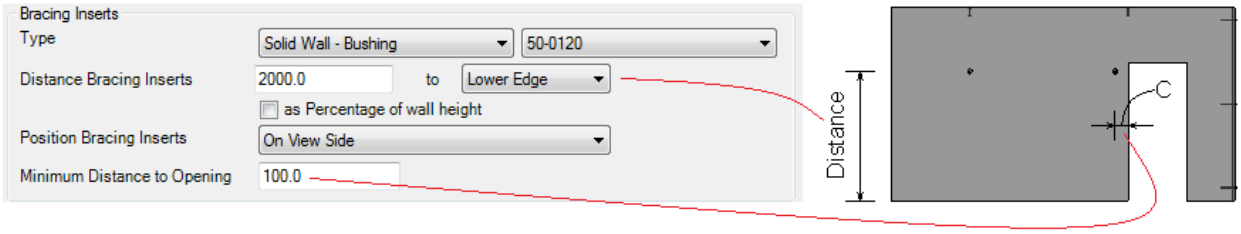


In the Lifters section you can set the lifter type, found inside the Lifter family, the minimal distance from the element edge (A) or an opening (B) and the maximum weight per lifter. For high walls, you can specify the height above which the software adds side lifters.



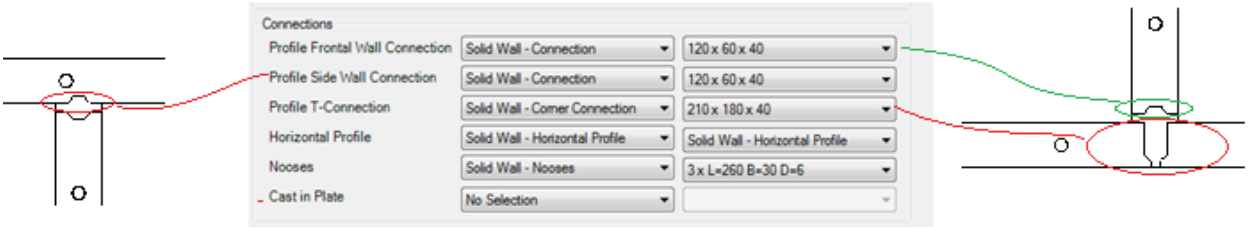
Lifters Section

In the Bracing Inserts section you can specify the position of bracing inserts, relative to the lower or upper edge of the wall or the minimum distance to opening (C).



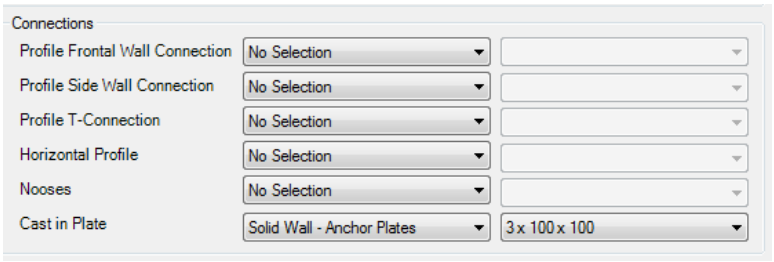
Bracing Inserts Section

In the Connections Inserts section you choose the profiles types that should be applied either where the segmentation is performed between two walls or at wall junctions.
The following configuration inside the Connections section is used when you want to create a connection using Nooses (Loops):



Lifters Section

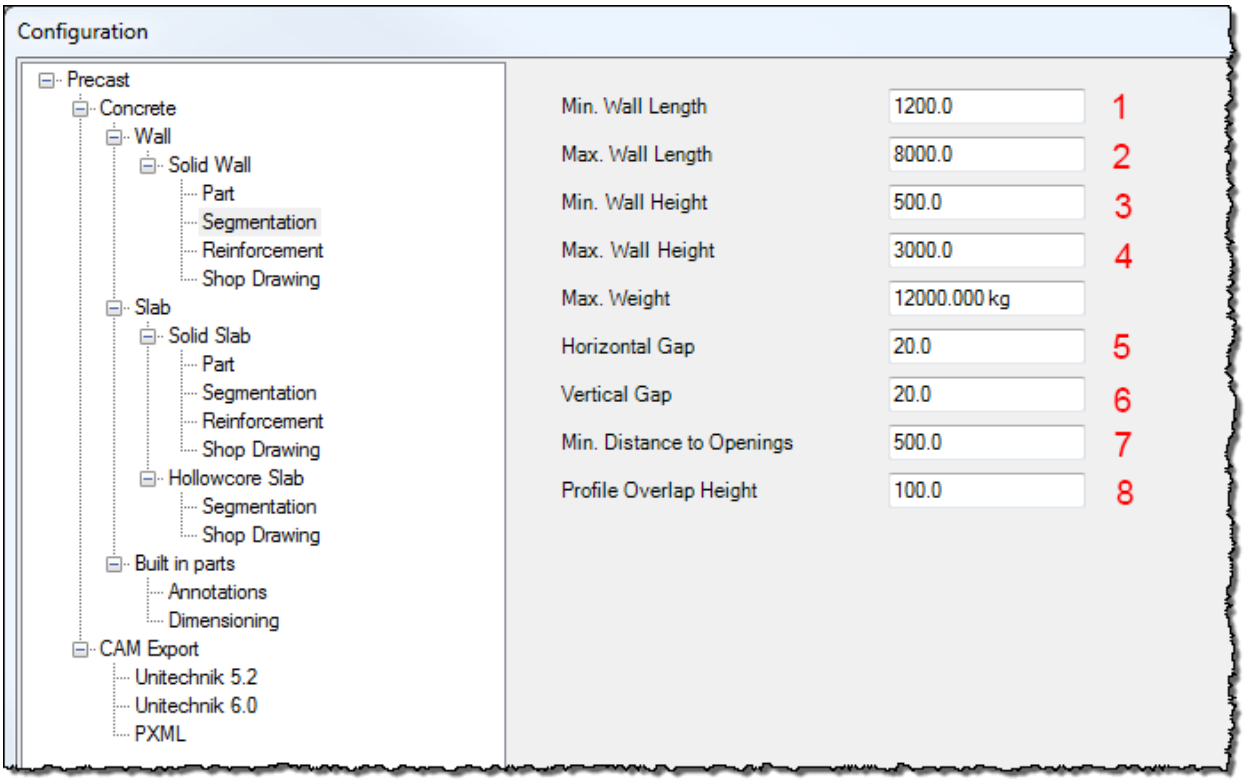
In order to create connections using anchor plates, the following configuration should be used:

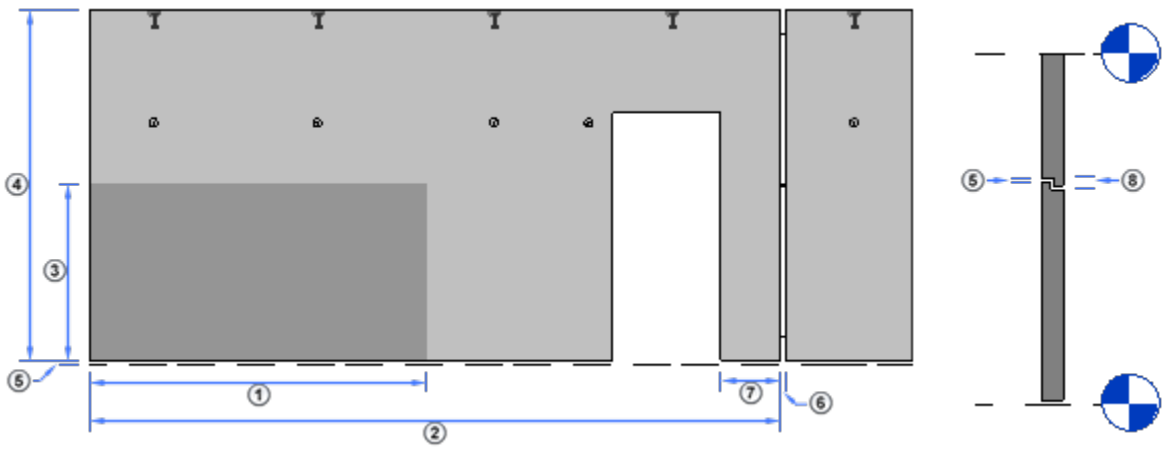


Connection with Anchor Plates Configuration

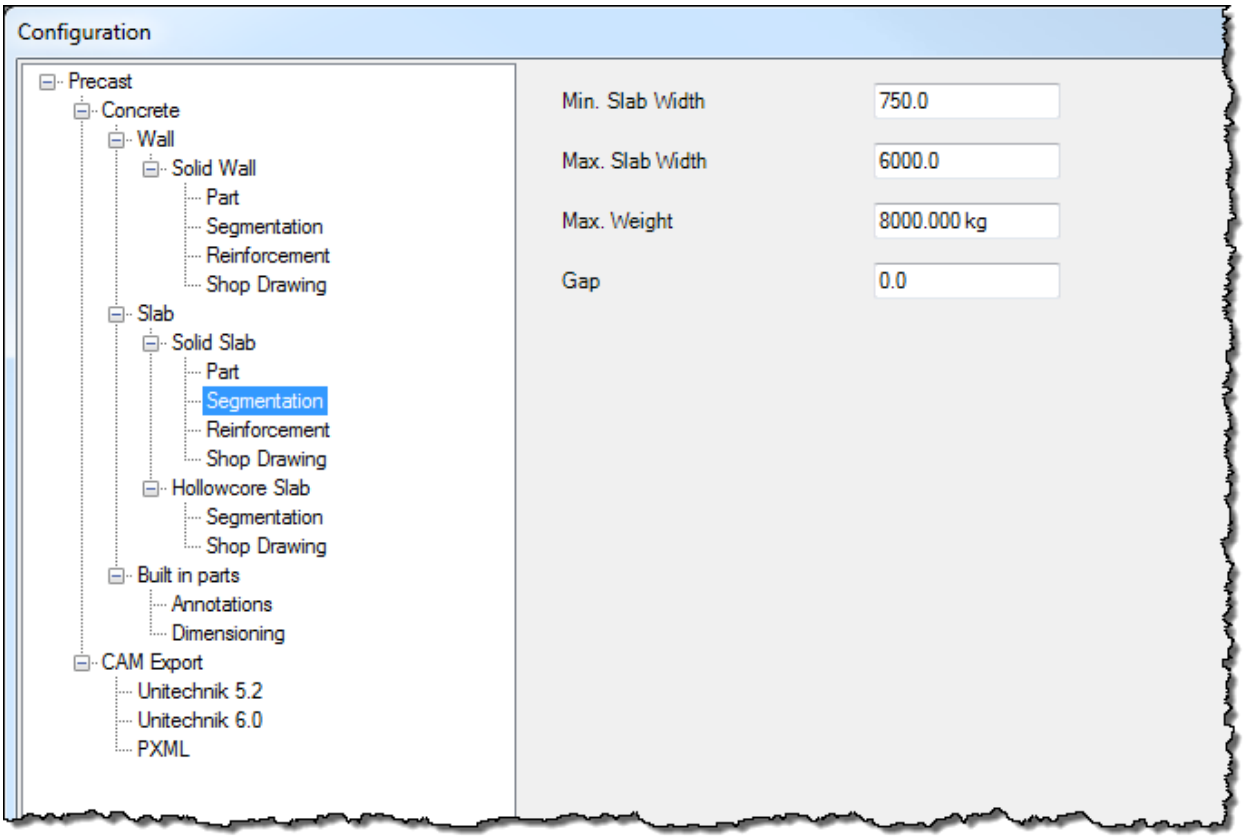
Connection with Anchor Plates

You can also change segmentation settings according to your needs on the Segmentation tab. You can adjust the following wall segmentation parameters for generating assemblies as needed.


Configuration
Precast
Concrete
Wall
Solid Wall
Part
Segmentation
Reinforcement
Shop Drawing
Slab
Solid Slab
Part
Segmentation
Reinforcement
Shop Drawing
Hollowcore Slab
Segmentation
Shop Drawing
Built in parts
Annotations
Dimensioning
CAM Export
Unitechnik 5.2
Unitechnik 6.0
PXML
Min. Wall Length 1200.0 1
Max. Wall Length 8000.0 2
Min. Wall Height 500.0 3
Max. Wall Height 3000.0 4
Max. Weight 12000.000 kg
Horizontal Gap 20.0 5
Vertical Gap 20.0 6
Min. Distance to Openings 500.0 7
Profile Overlap Height 100.0 8


4
3
5
1
2
7
6
5
8

In Revit a slab must be created as Floor: structural and one layer of the floor structure must be set to Structural Material (even if only one layer is defined). At least this layer must have a material defined and the material must have a defined density.



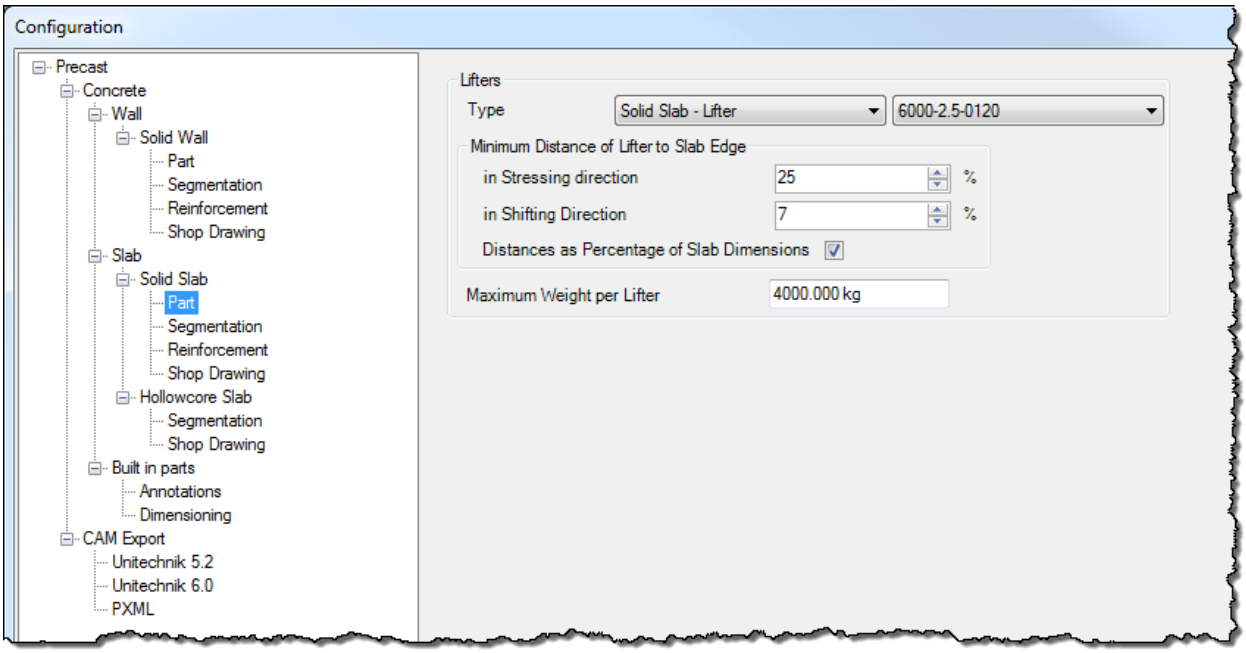
Segmentation parameters:

- Min. Slab Width: This parameter sets the minimum width for a solid slab assembly;
- Max. Slab Width: This parameter sets the maximum width for a solid slab assembly;
- Max Weight: This parameter sets the maximum weight for a solid slab assembly; This parameter forces the segmentation at the position where the weight of the assembly matched the maximum weight specified inside configuration. If the weight determined a width that is not in the interval defined by the minimum and maximum slab width, the user is warned that he should change the segmentation parameters.
- Gap: This parameter determines the gap (distance) between two adjacent assemblies.

If there is no material assigned the segmentation will be performed taking into account only the minimum and maximum width parameters.

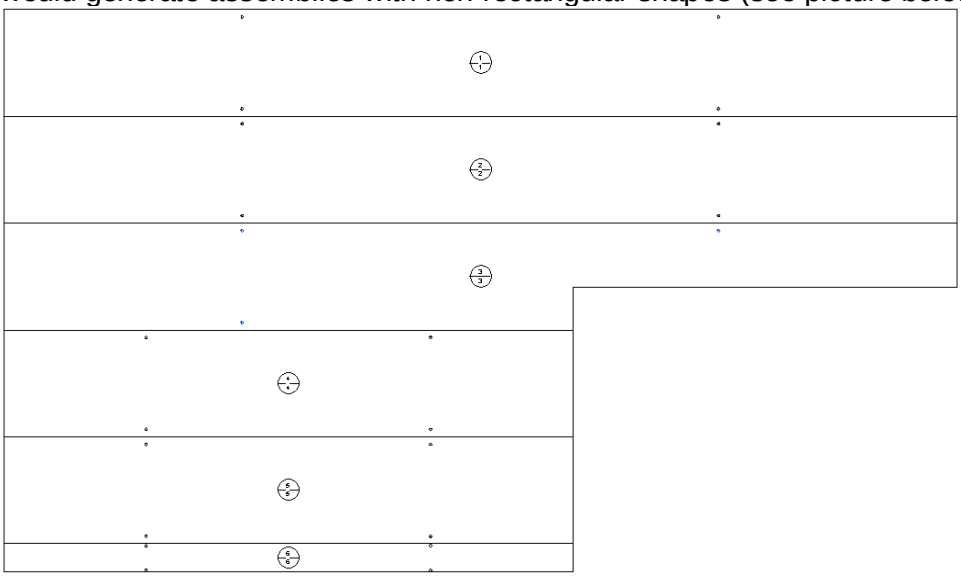
When the floor has material and if the width resulted considering the actual weight is between the minimum and maximum width, the governing parameter will be the width determined so that the assembly has the weight as close as possible to the parameter from Configuration.

During the split process 4 lifters are added at specified location inside Configuration. There are always only 4 lifters added; in the case of the weight being higher that the capacity of the 4 lifters the user is warned that he should change the type of lifters or manually add additional lifters so that the solid slab assembly can be transported and mounted on site.

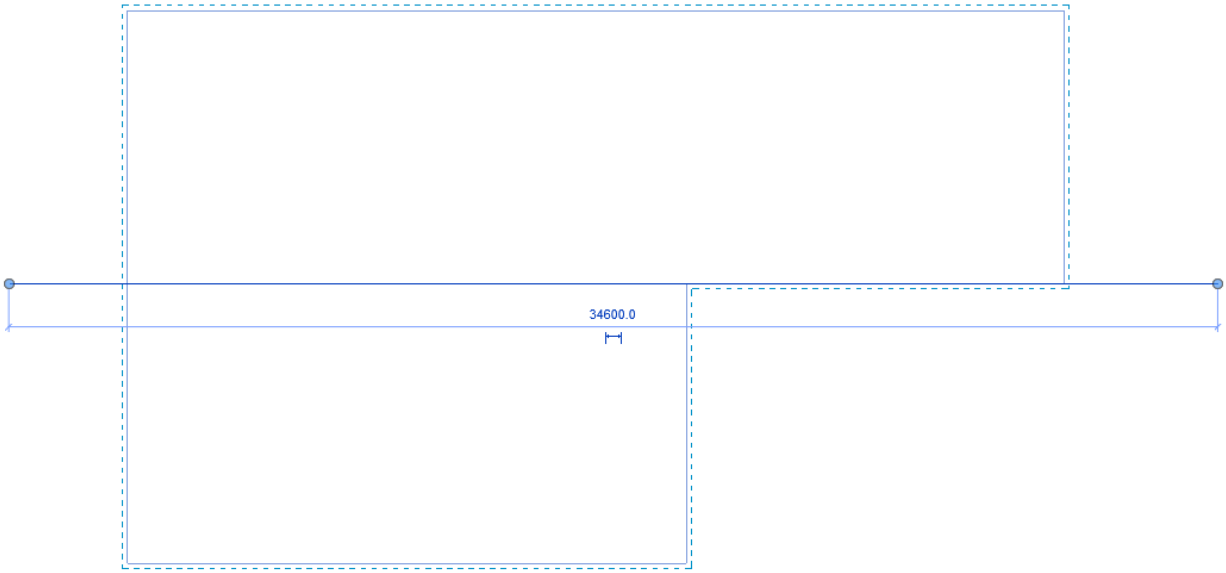


Manual division of slabs:

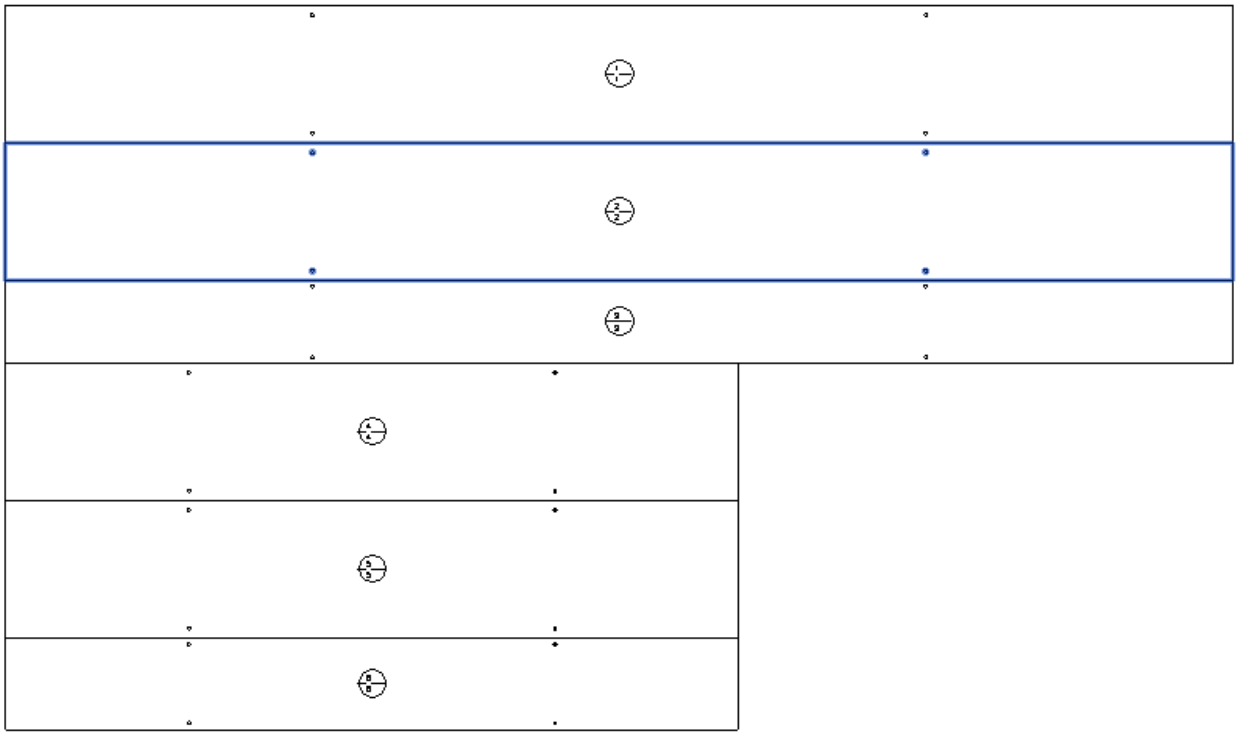
For some cases, there might be some situations when using the segmentation parameters would generate assemblies with non-rectangular shapes (see picture below).



In order to force the segmentation at a certain point, the user should transform the floor into parts and add division lines at the position where the segmentation should be performed.

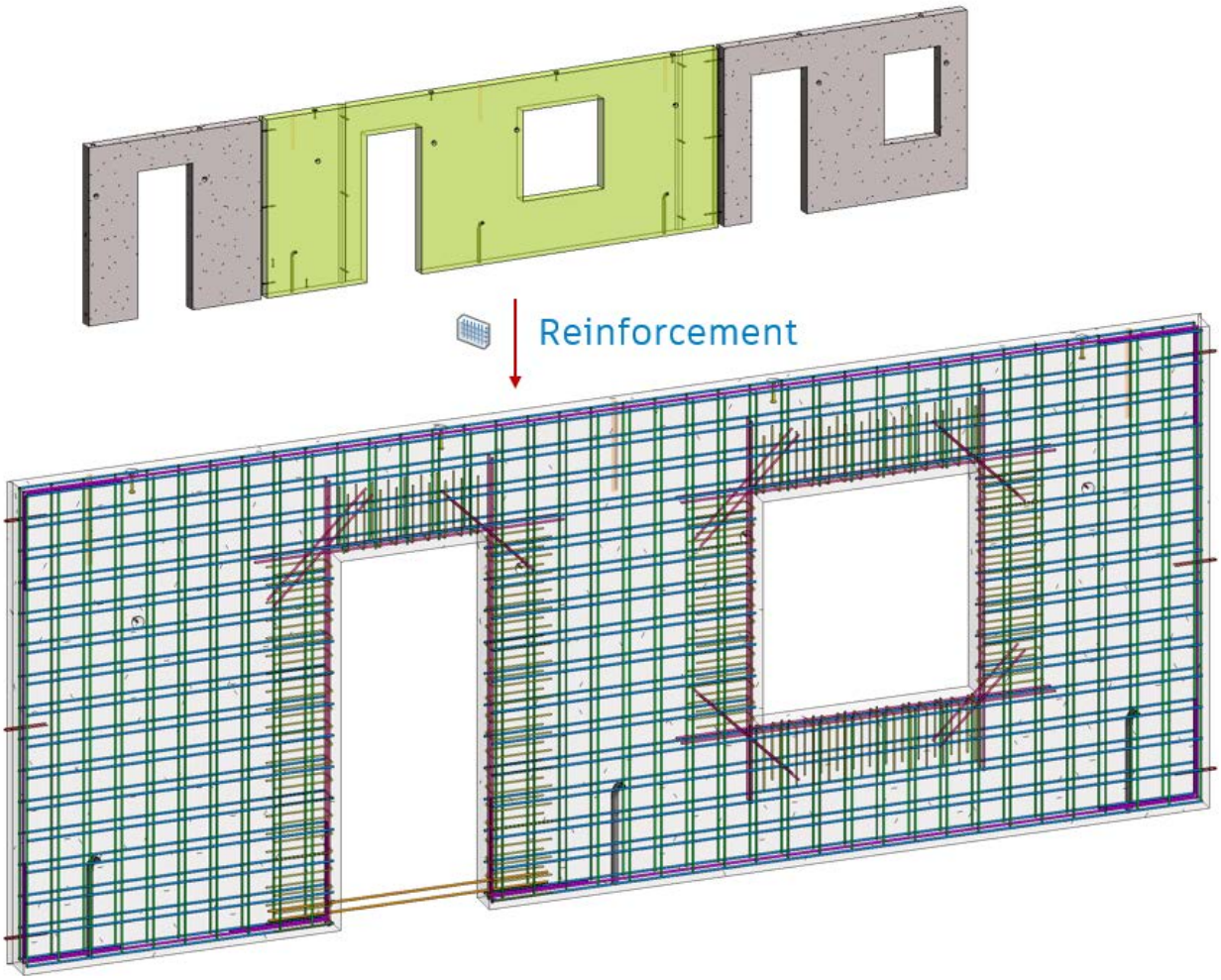


After the floor has been divided into relevant parts, the user can select and split them into producible assemblies. Once the Split command has been performed on one part, the user is asked which precast slab he would like to have and the original floor inherits the proprieties selected for the first part (e.g.. Solid Slab or Hollow Core Slab). For the other parts, the user will not be asked about the type of floor and will get the segmentation according to the segmentation parameters present in the Configuration.



Reinforce walls and floor according to user-defined rules

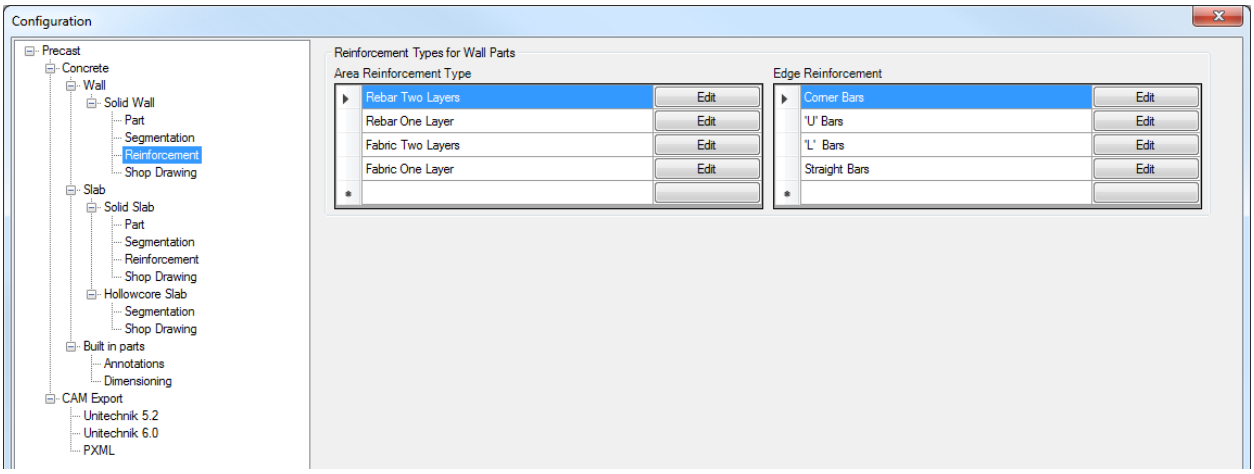
After the wall has been split into precast assemblies, the user can add reinforcement for the wall by using the *Reinforcement* command.



Using the Reinforcement section in the Configuration dialog, you can define any type of wall reinforcement to fit your requirements.

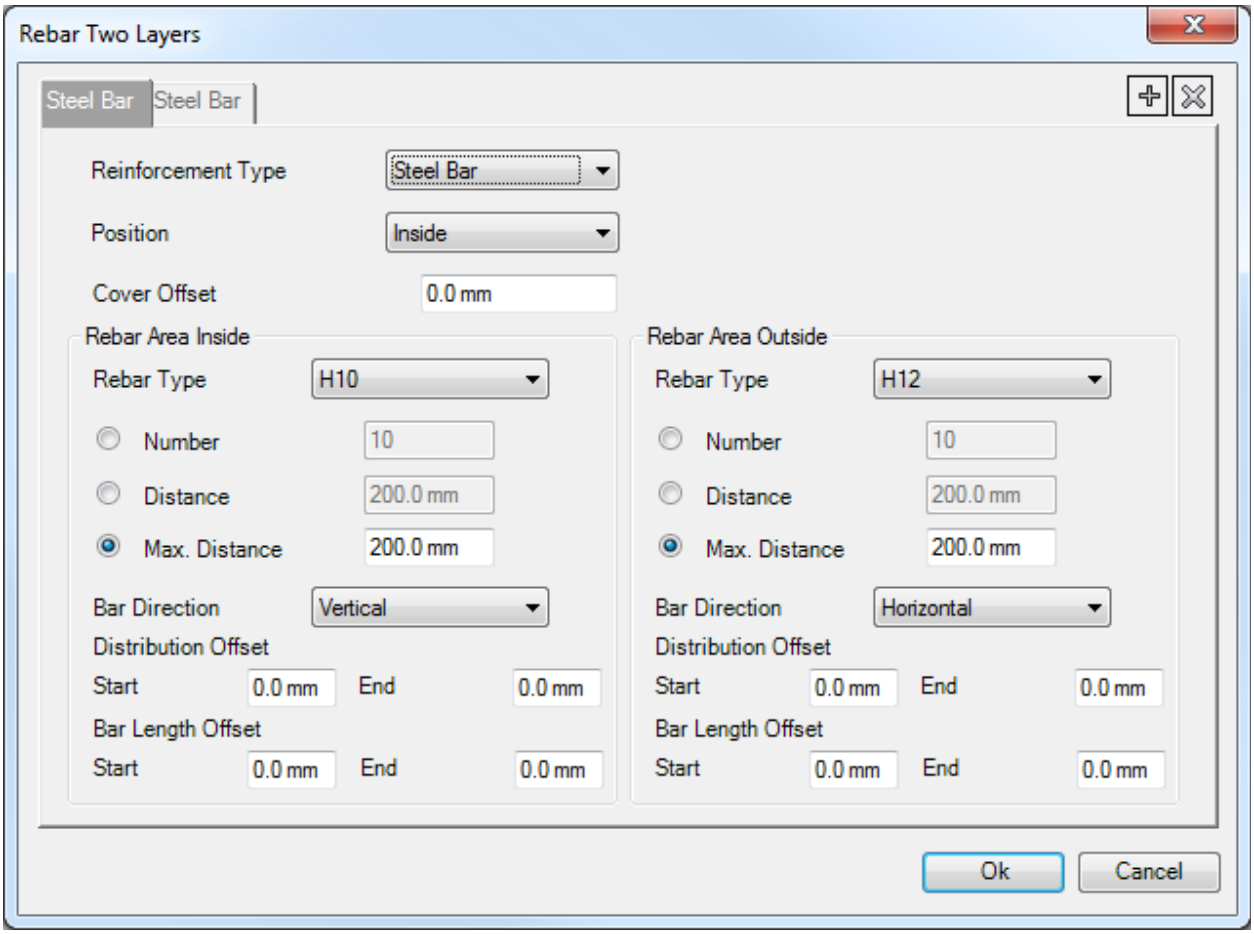
Area reinforcement component enables the user to add area reinforcement for a wall/slab according to pre-defined styles. In the he Reinforcement section in the Configuration dialog, you can define any type of wall reinforcement to fit your requirements.

To add new reinforcement style, go to ***Precast*** tab → ***Configuration*** panel →***Configuration*** → ***Wall*** → ***Solid Wall*** → ***Reinforcement***, type the desired name and click 'Edit' to customize it so that it fits your needs. After the creation of a new reinforcement style, this will immediately be available in the dialog for *Reinforcement* command.



Following the split of walls into precast assemblies, the user can add reinforcement for the wall by using the *Reinforcement* command. Reinforcement command can be applied on precast assemblies obtained either from manual or automatic segmentation.

Area Reinforcement Settings



Inside each area reinforcement type dialog box, you can define the following properties:

Number of layers: you can add at most 6 layers by clicking the "✙" sign and you can also remove them by clicking the "✖" in the top right corner of each layer.

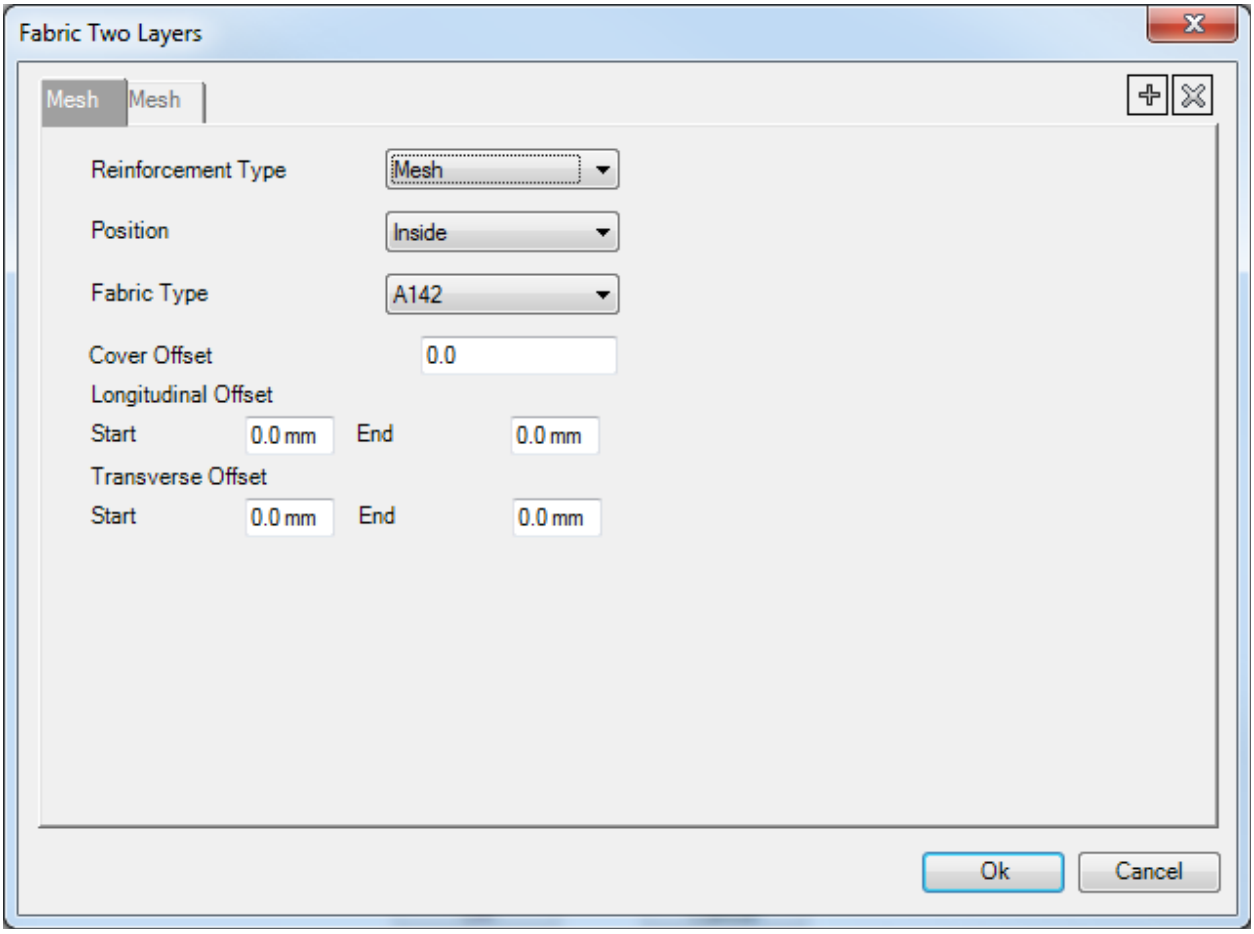
Reinforcement Type: you can choose from the following reinforcement types: Steel Bars or Meshes.

Steel Bar Reinforcement Type*:*

- Position – define the position of the reinforcement choosing one of the three options from the drop-down list: Inside, Center or Outside.
  - Inside - steel bars will be placed on the inside of the wall, where the inside is the opposing face to the viewing direction of the wall.
  - Center - steel bars will be placed at the center of the element determined by the cover boundaries;
  - Outside - steel bars will be placed on the inside of the wall where the inside is the opposing face to the viewing direction of the wall.
- Cover Offset – With this option the user can specify additional cover offset for the rebar area. This can be only positive and shift the rebar area toward the center of the wall.
- Rebar Area Inside / Outside
  In the current layer, you can define the rebar relative position. You can specify which reinforcement goes first (is bound to the cover – Rebar Area Inside). You can choose the bar direction for the two available sets of reinforcement (horizontal / vertical).
  The rebar set can be defined either by the number of bars, the distance between bars or the maximum distance between bars.
  - *Distribution Offset*: This parameter allows the user to adjust the offset in the direction of the rebar distribution, thus being able to increase or decrease the length of distribution
  - *Bar Length Offset*: This parameter allows the user to adjust the offset in the direction perpendicular to the direction of distribution, thus being able to increase or decrease the length of reinforcement.

The offset properties are relative to the slab boundary. The offset overwrites the actual rebar cover and gives you the possibility to extend/trim the reinforcement. Negative offset extends the reinforcement outside of wall boundaries.

Mesh Reinforcement Settings



- Reinforcement Type – The mesh type which will be used in the fabric area can be specified.
- Position – define the position of the reinforcement choosing one of the three options from the drop-down list: Inside, Center or Outside.
    - Inside - steel bars will be placed on the inside of the wall, where the inside is the opposing face to the viewing direction of the wall.
    - Center - steel bars will be placed at the center of the element determined by the cover boundaries;
    - Outside - steel bars will be placed on the inside of the wall where the inside is the opposing face to the viewing direction of the wall.
- Cover Offset – With this option the user can specify additional cover offset for the rebar area. This can be only positive and shift the rebar area toward the center of the wall.
- Rebar Area Inside / Outside

In the current layer, you can define the rebar relative position. You can specify which reinforcement goes first (is bound to the cover – Rebar Area Inside). You can choose the bar direction for the two available sets of reinforcement (horizontal / vertical).
The rebar set can be defined either by the number of bars, the distance between bars or the maximum distance between bars.

- *Transversal Offset*: This parameter allows the user to adjust the offset in the direction perpendicular to the direction of fabrics.
- *Longitudinal Offset*: This parameter allows the user to adjust the offset in the direction of the fabrics.

The offset properties are relative to the slab boundary. The offset overwrites the actual rebar cover and gives you the possibility to extend/trim the reinforcement. Negative offset extends the reinforcement outside of wall boundaries.

Area Reinforcement Types

To create new area reinforcement types, go to *Configuration → Wall → Solid Wall →* Reinforcement, and start typing in the blank row the desired name and edit so that it matches your needs. After new reinforcement styles are added, their names are automatically displayed when Reinforcement command is being used.

Apply Area Reinforcement Types

In order to apply a previously defined edge reinforcement you should select either the original wall (that has been previously been split), multiple/individual assembly or corresponding parts and press the *Reinforcement* command. To apply on the edge reinforcement to your selection, you should select *None* for the *Area Reinforcement Type.*

Apply the Existing Area Reinforcement

To edit the existing area reinforcement, select the element(s) for which you want to change the configuration of the area reinforcement, press the *Reinforcement* command and edit the current type or select a different type from the list.

When editing an area reinforcement for a precast element, if the modifications do not match other pre-defined styles, a new reinforcement style will be created and named User Defined Reinforcement. This means that this style is element specific (stored in the precast element) and is not defined in the Configuration.

If the "User Defined Reinforcement" style is edited and all the parameters are equal to the ones from "Configuration" style, it will be automatically deleted and will not be visible anymore in the area reinforcement type drop-down list.

If multiple parts/assemblies (from different entities) are selected and "Reinforcement" command is used, only the parameters for the first *User defined reinforcement* style will be available to be edited.

Manually Added Reinforcement

For each precast assembly that has been populated with reinforcement, the user can manually add extra reinforcement. This can be achieved by using the standard reinforcement tools provided

by Revit. If the user modifies the reinforcement for the precast assembly, the manually added reinforcement will not be modified and will retains its definition (position and parameters).

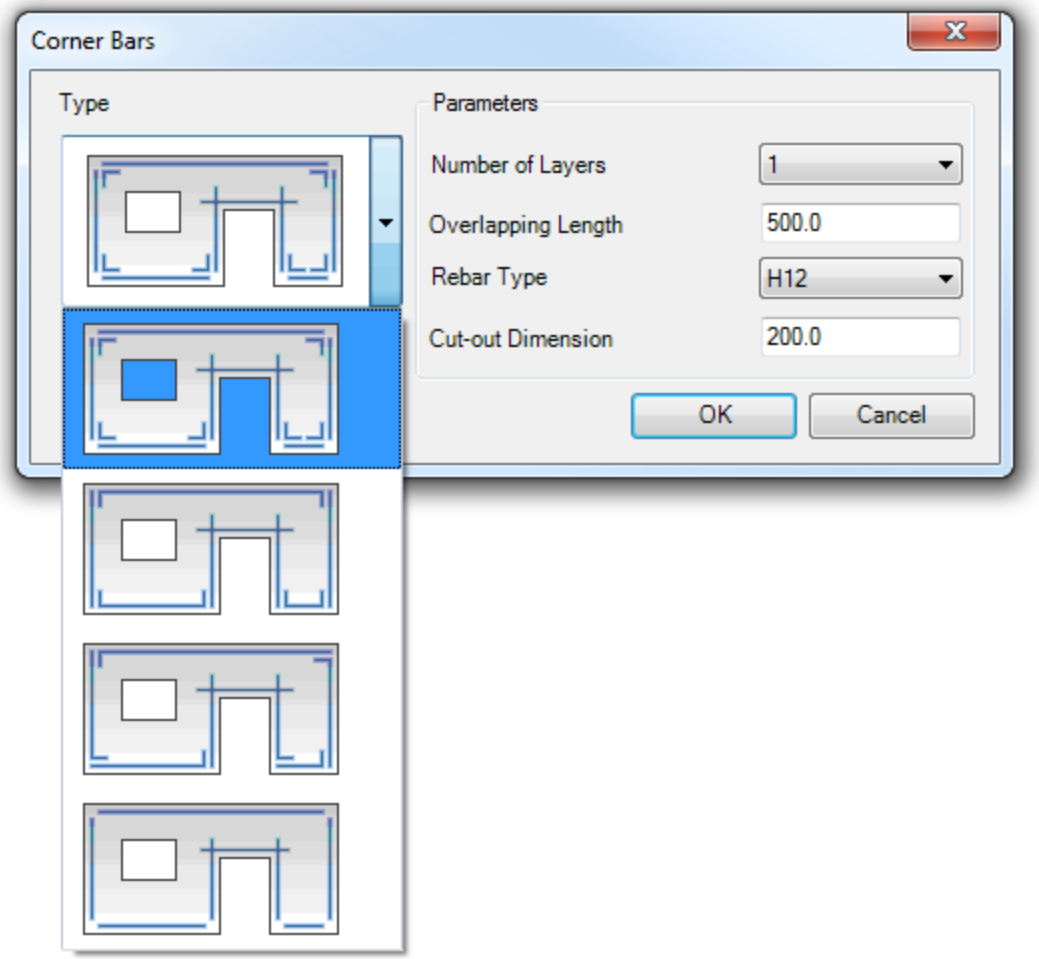
Manual adjustments can be performed on the area reinforcement but you must keep in mind that on the re-creation of area reinforcement those modifications will not remain

Area Reinforcement Re-creation

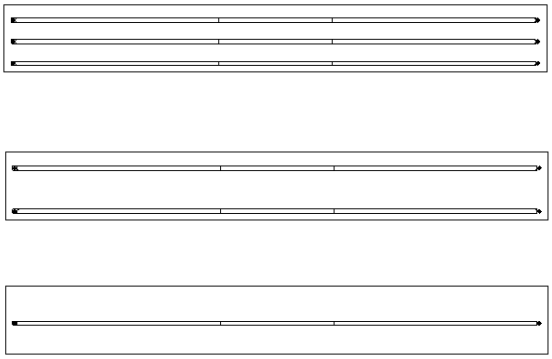
The area reinforcement is always re-created whenever the contour of the elements or openings are added to the wall. Therefore, if previous modifications have been performed to the area reinforcement these will not be retained. The re-generation of are reinforcement is performed even for small recessed like the void of lifters or bushings.

Edge Reinforcement Settings

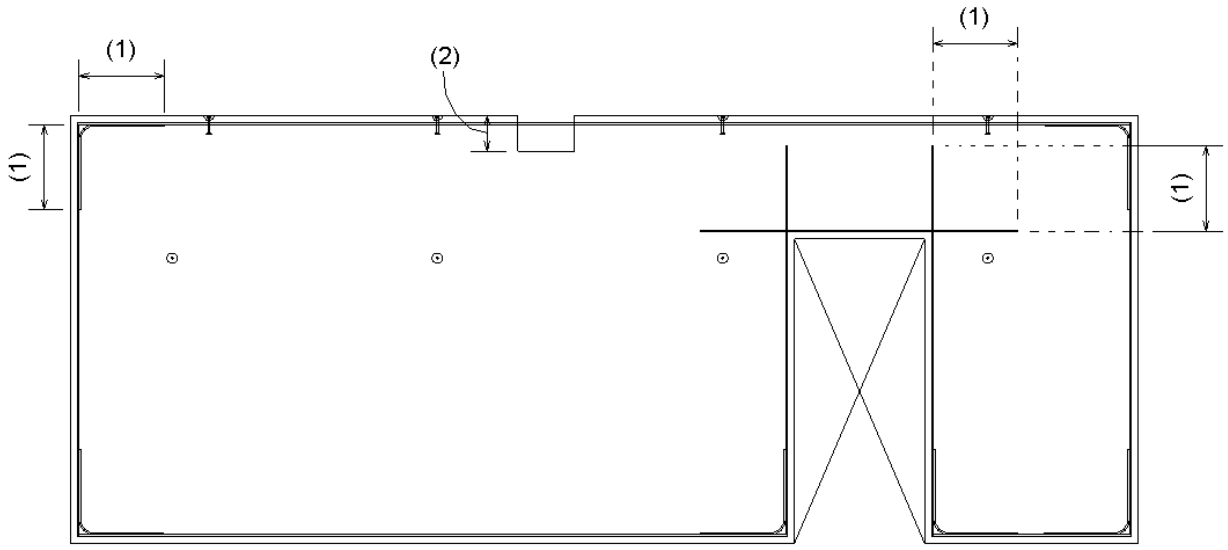
In the edge reinforcement dialog one can the following parameters:

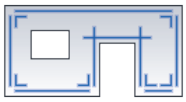
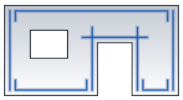
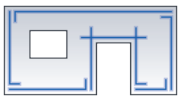
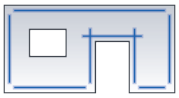


- **Number of layers:** This represents the number of layers of edge reinforcement inside the solid wall;
    - **1 layer:** is positioned at the middle of the wall;
    - **2 layers:** are positioned at the faces determined by the covers;
    - **3 layers:** are positioned at the faces determined by the covers and at the middle of the wall;



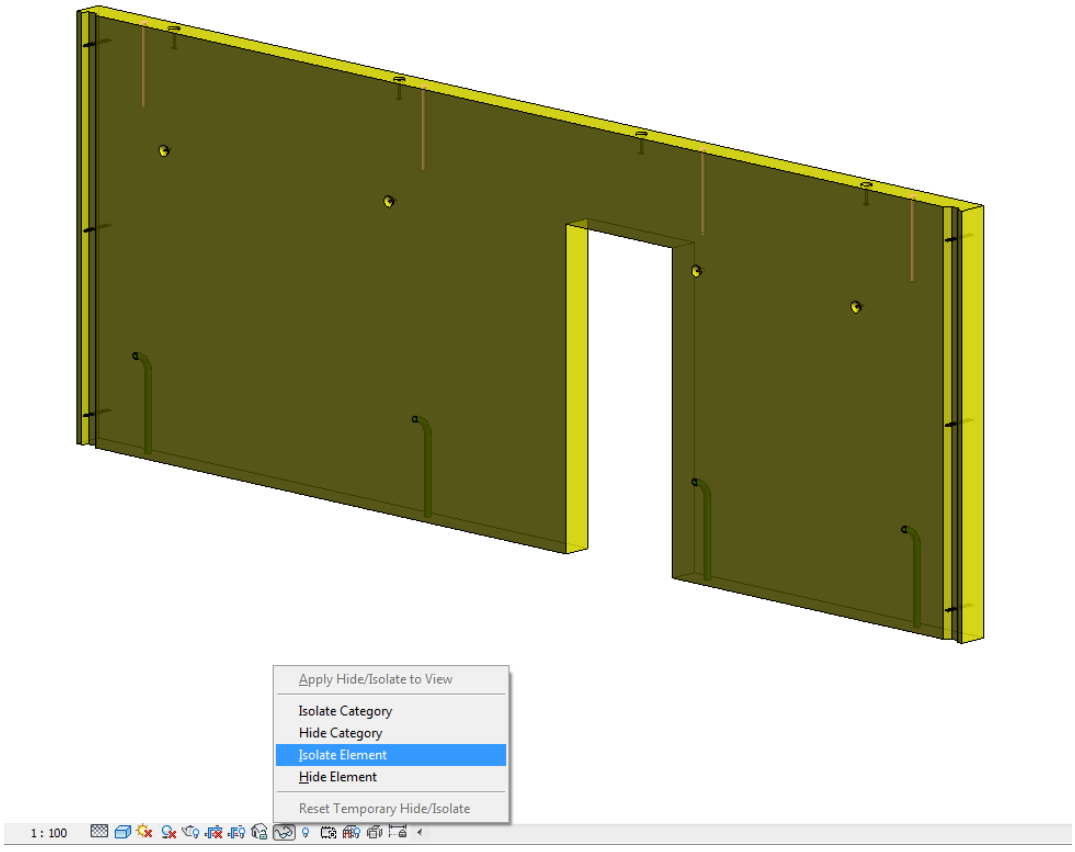
- **Overlapping Length (1):** is the length on which the bars are overlapping so that the efforts are transferred around the contour. The overlapping parameters is applied throughout the whole contour (there is no ability to specify independent overlapping lengths for different situations);
- **Rebar Type:** is the diameter used for the edge reinforcement;
- **Cut-out Dimension (2):** is a parameter that allows the user to specify the \limit for which the rebar will go through the cut-out or will treat it as a recess and add edge reinforcement accordingly.



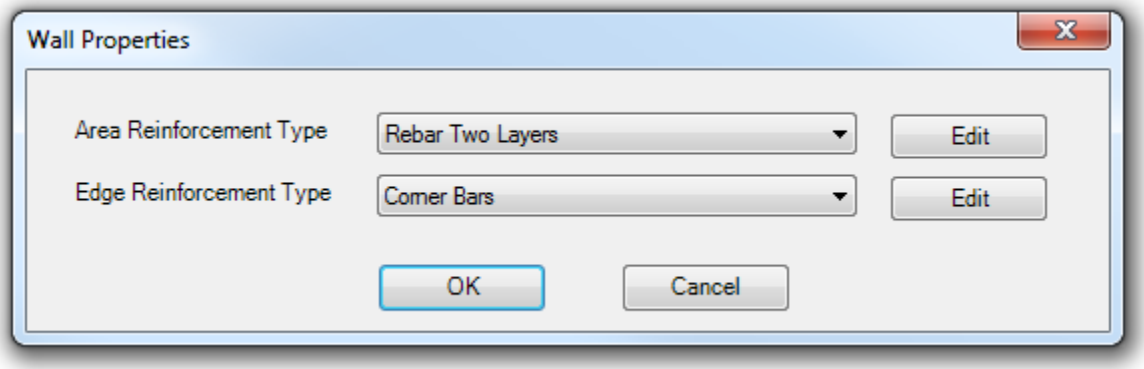
Edge reinforcement with corner and straight bars	
Edge reinforcement with 'U' and straight bars	
Edge reinforcement with 'L' and straight bars	
Edge reinforcement with only straight bars	

In order to add area and edge reinforcement to a wall assembly you have to perform the following steps:

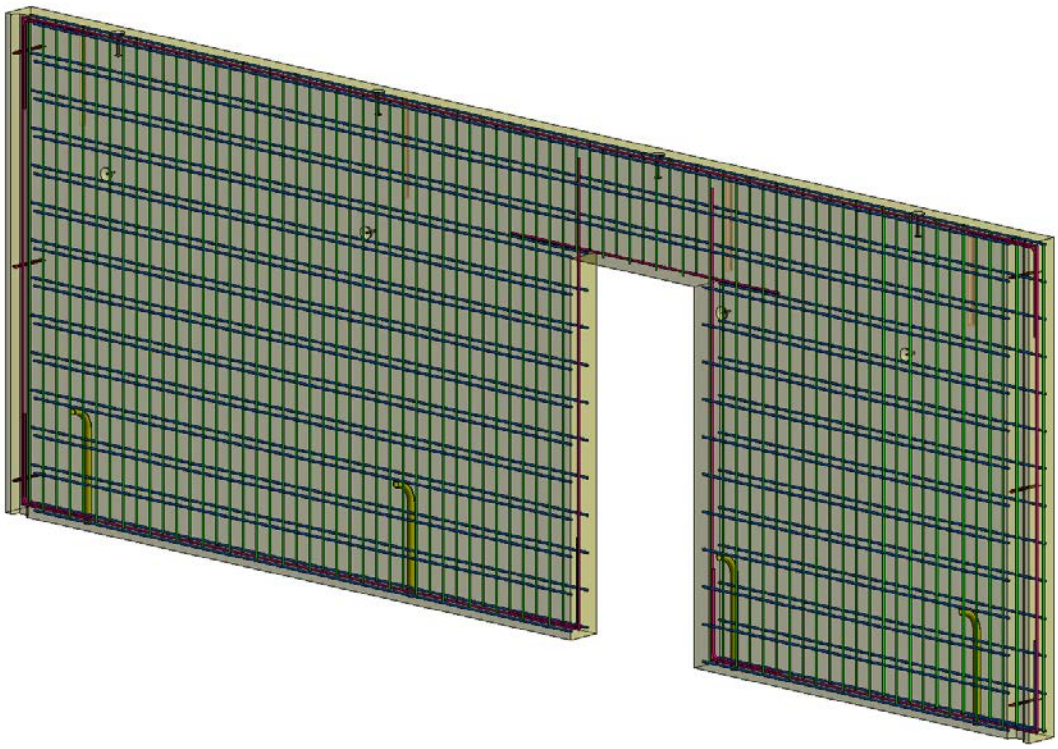
1) Select the a previously split wall assembly; Optional: Isolate the assembly using the Temporary Hide/Isolate button;

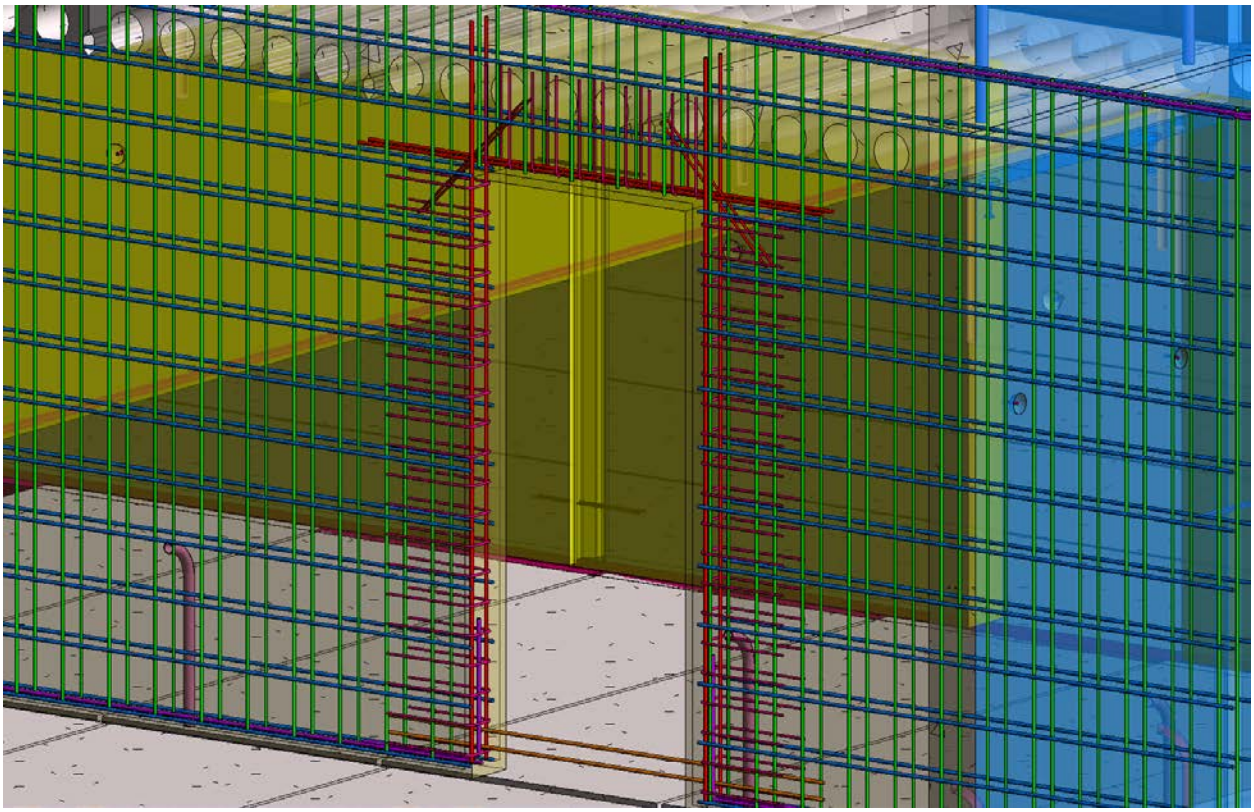


2) Press the *Reinforcement* command. The following dialog appears:



3) Click Ok button. The assembly should be reinforced with area and edge reinforcement.

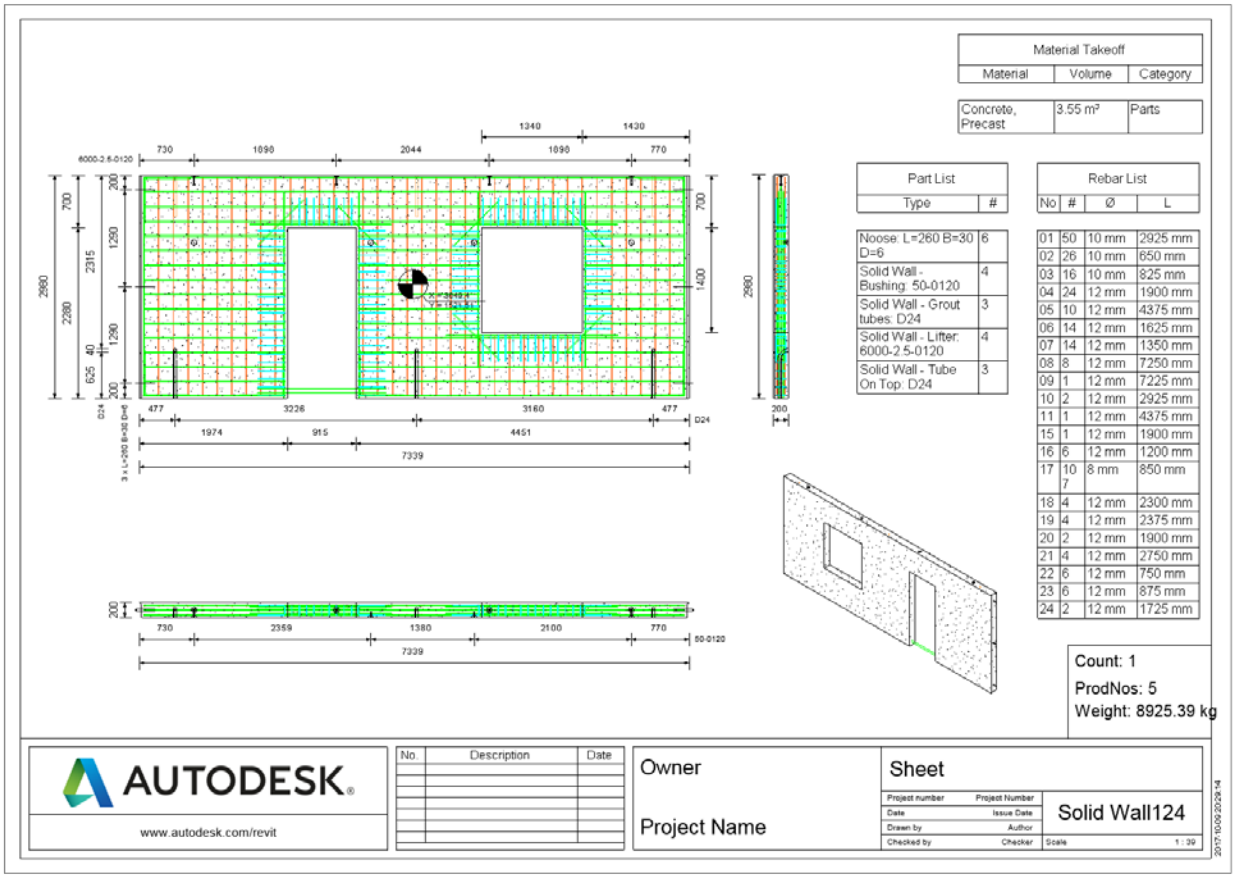




Shop Drawings and CAM files generation

To create shop drawings select the elements for which you want to create shop drawings and use the ***Shop Drawings*** command.

For each wall or slab assembly you can automatically create a shop drawing.



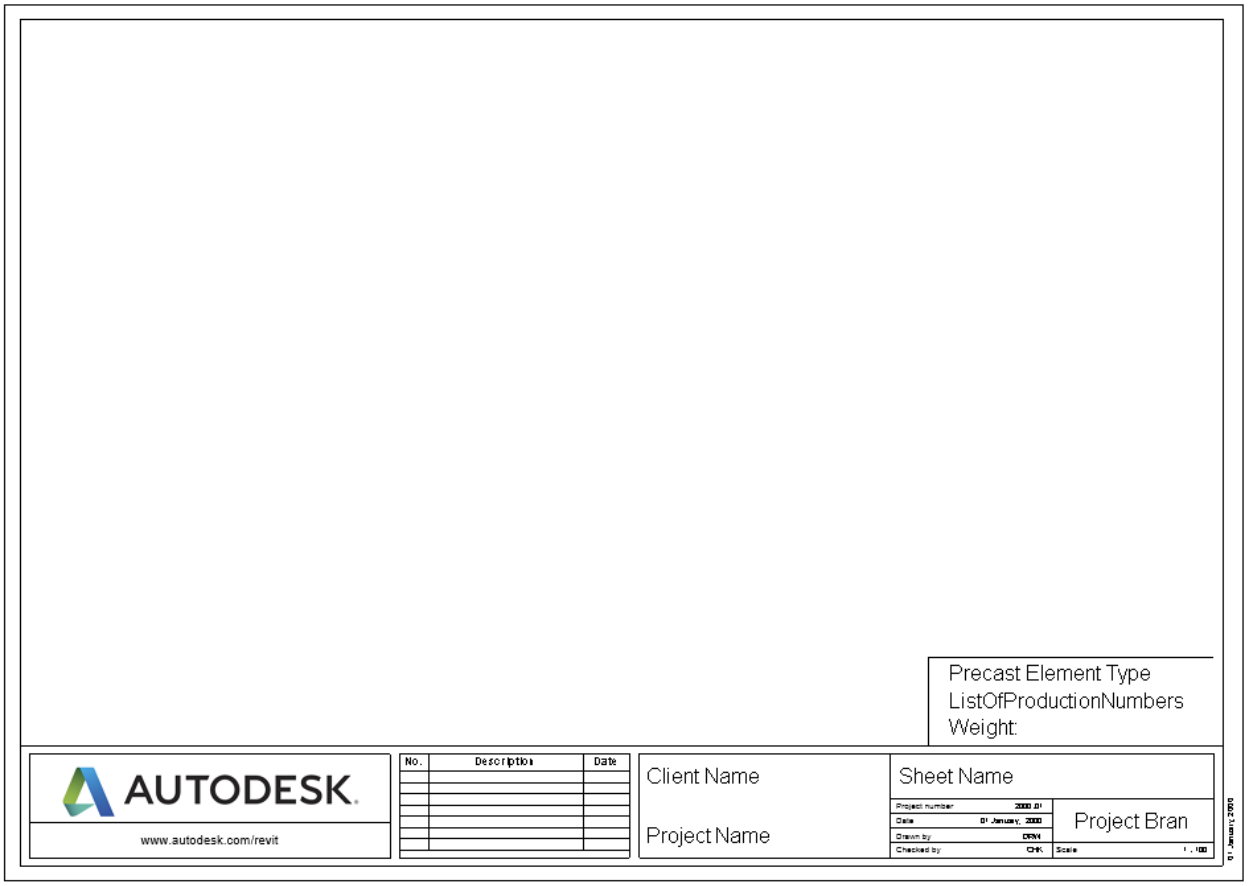
The settings for Shop Drawings can be found in the Configuration under the Shop Drawings section.
For each wall or slab assembly you can automatically create a shop drawing. Therefore, you need to define and set a title block in the project data.

Title block

You can define the views and where they should be placed on the title block, by adding instances of a viewpoint family on the title block definition in the title block family. To define the views position on the sheet, you need to use the “Viewport.rfa” family located in the “Families”/en folder.

To create your own title block follow these steps:

1. Open the title block family you want to use as a template or create a new one.



2. Modify the title block as needed, adding data fields, company logo etc.
3. The following Shared Parameters with additional element information are available for labels on the title block:
    - PosNoSheet – Position number of the element
    - ListOfProductionNumbers – List of the production numbers for the element on the sheet
    - NumberElementSheet – The number of elements that must be produced
    - ThicknessSheet – Thickness of the element
    - WeightSheet – Weight of the element
    - VolumeSheet – Concrete volume of the element

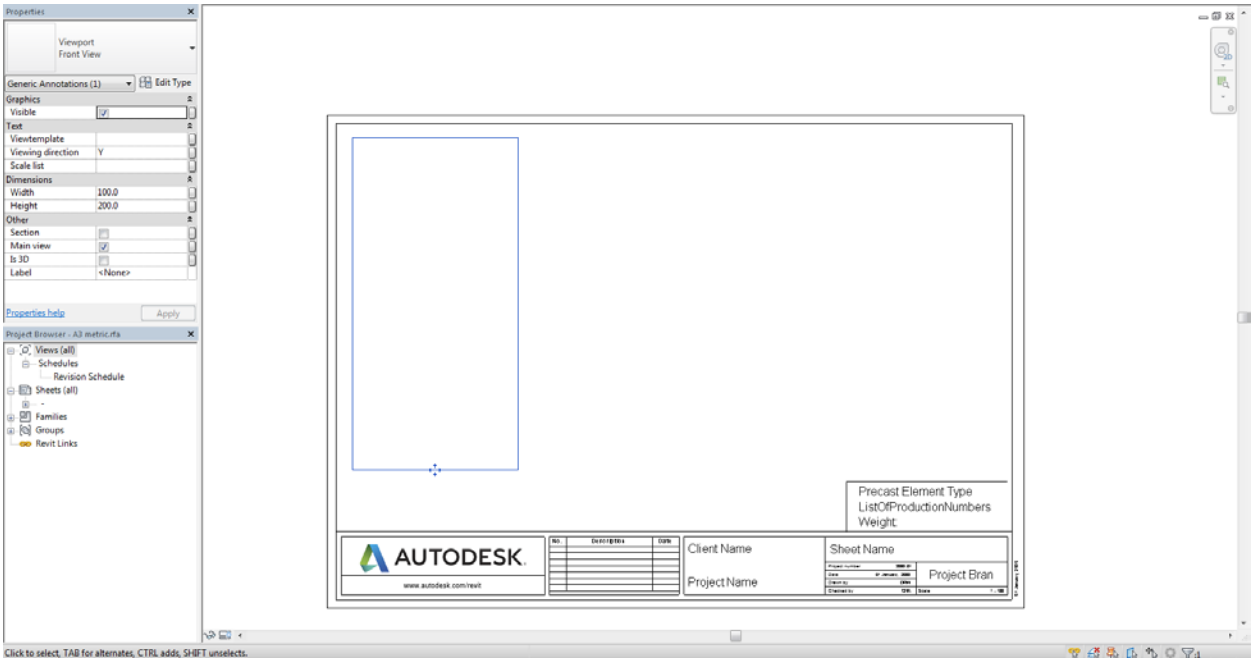
    To insert the labels:

    - Create and place a label on the title block: *Create* → *Text* → *Label*.
    - In the *Edit Label* dialog click the *Add Parameters* button.
    - In the *Parameter Properties* dialog click the *Select* button and select the parameter you need. Close the dialog.

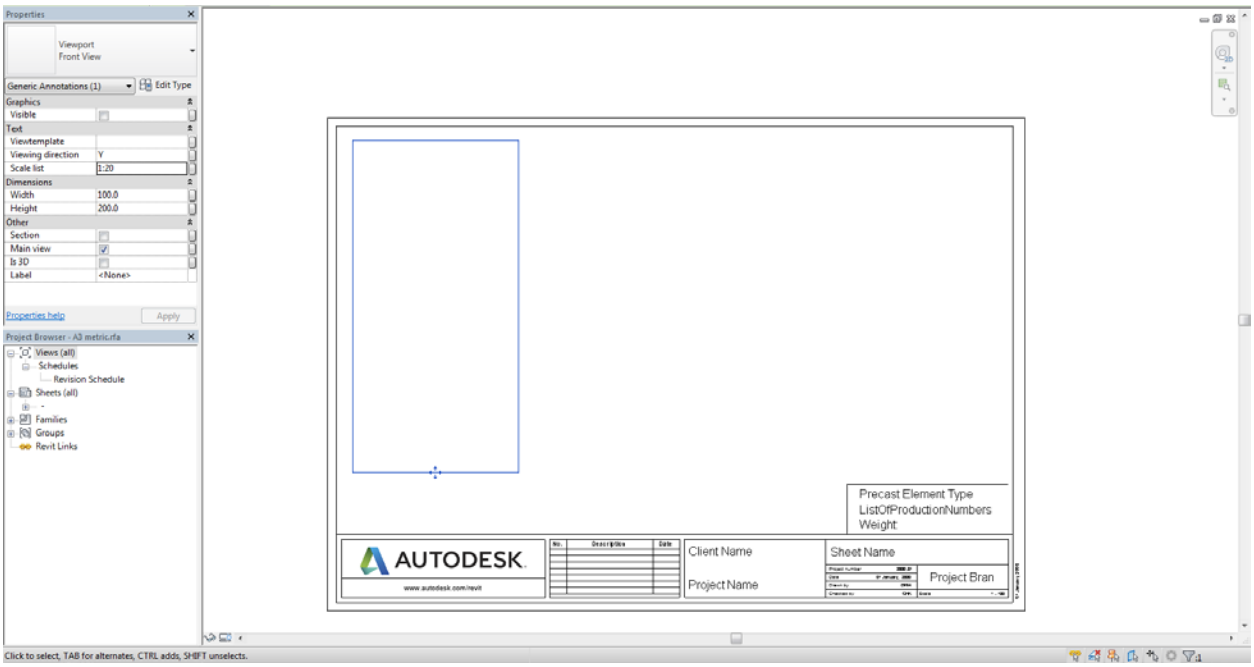
- Add the selected parameter to the label using the *Add parameter(s) to label* button (green arrow) and edit the prefix and suffix.
- Repeat as needed

4. Load the "Viewport.rfa" and "Schedule.rfa" families: *Insert* → *Load from Library* → *Load Family*.
   The family is located in the Structural Precast Revit 2018 Program Data Folder, "Title Blocks" folder (e.g. "C:\ProgramData\Autodesk\Structural Precast for Revit 2018\Families\en\Title Blocks").

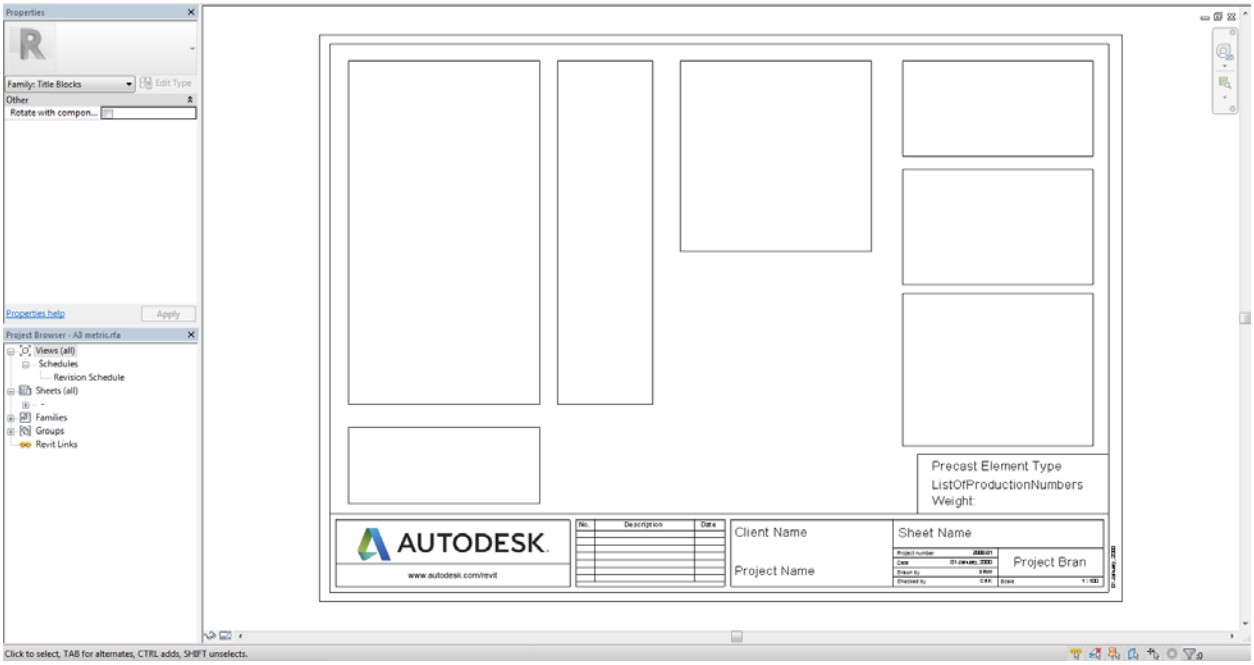
5. Insert Viewpoints: *Create* → *Detail* → *Symbol*; before placing the view port, select the view port type from the properties. You can use the following view ports: Top View, Side View, Front View, 3D View, Material Takeoff, and Part List.



6. Select the inserted view port and modify the Width and Height parameters with the correct dimensions. Uncheck the *Visible* parameter of the view port. Here you can also specify the desired scale list and view template.



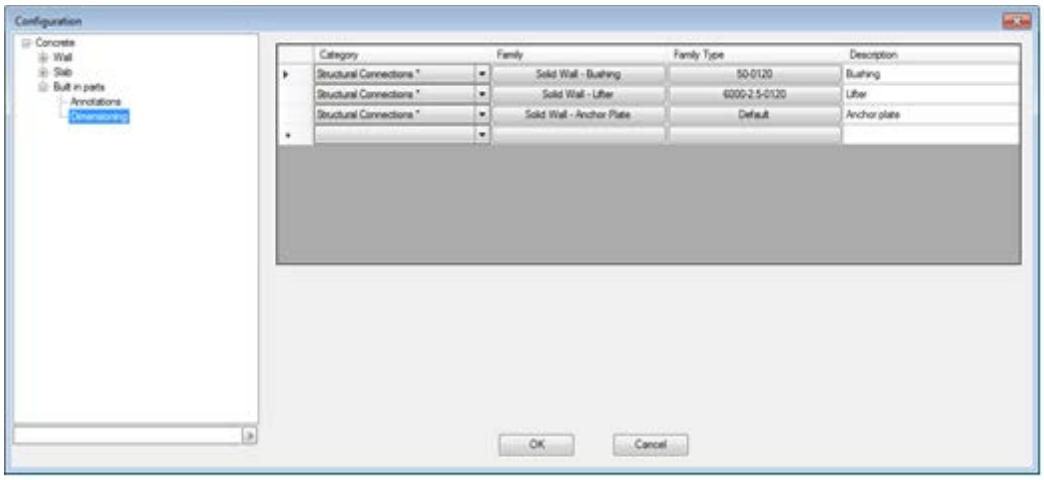
7. Repeat for all views that should be placed on the shop drawings:



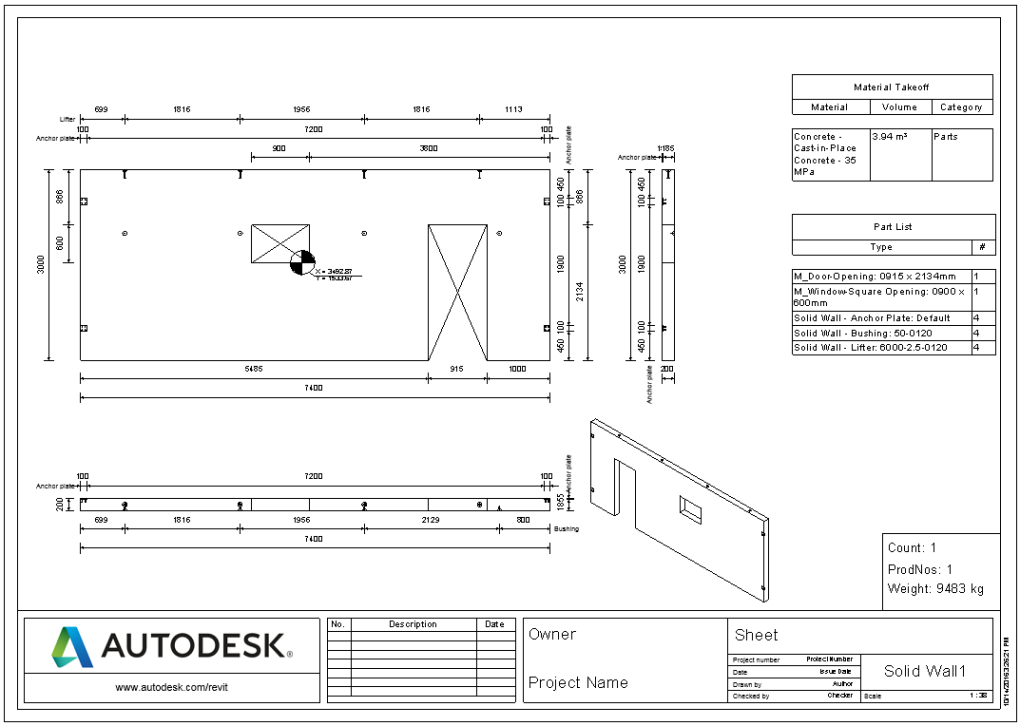
8. Save the title block.

Dimensions Description

Inside *Configuration*, under *Built in Parts* there is the *Dimensioning* tab where the user can configure the description of the dimension line for a certain family type. Each family type has a unique dimension line assigned to it. If there is no description added, the name of the type family will be added as description.



Families manually added to the assembly will also be automatically dimensioned with related description (see example below). In the example below the anchor plates have been added manually.



Manually Created Assemblies

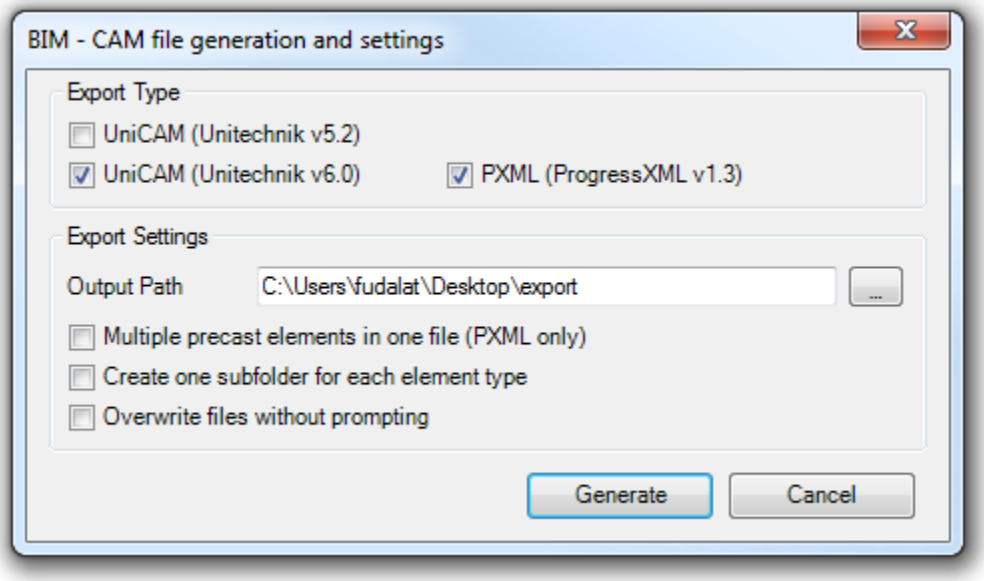
Precast for Revit has the ability to create shop drawings for manually created assemblies for planar elements. The main condition for the shop drawings to be created for a planar assembly is that its associated part is contained inside the assembly.

Workflow example:

Step 1. Create a planar element (slab or wall).
Step 2. Create part(s) for the planar element.
Step 3. Create the assembly containing the part and associated elements (mounting parts and reinforcement)
Step 4. Select the assembly and use the *Shop Drawings* command. The command will generate a shop drawing starting from the template configured from solid wall for manually created wall assemblies and from solid slab for manually created assemblies for slabs.

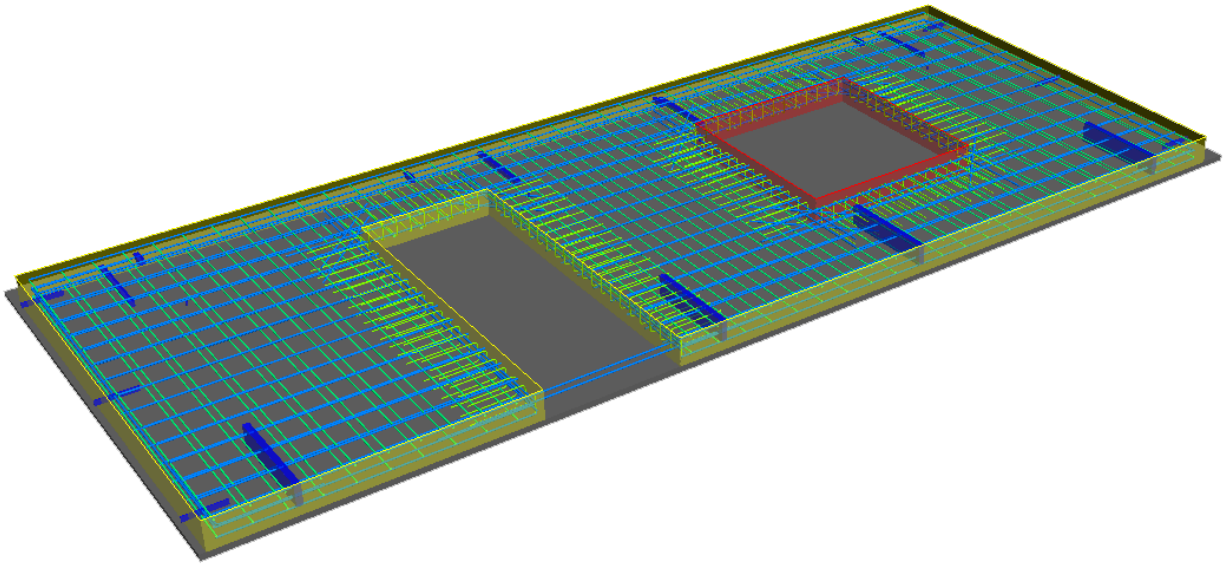
CAM files generation

In order to create CAM files you need to select all the assemblies and start the CAM Export command.



In this dialog you can select the export type (Unitechnik V5.2, Unitechnik V6.0 or ProgressXML v1.3), the output path for the files and different option for when exporting. In order to generate the files you have to press the *Generate* button.

After this you can select the assembly and export it to the desired format. If needed this can be viewed in the specific viewer.



APPENDIXES

Appendix 1 – Dimensioning Mounting Parts

1. Overview

For a mounting part dimension lines are created in a view, if

1. The family of the mounting part defines special lines, which are invisible and are used as dimensioning points. (The lines can also be created in a sub-family, but only if the sub-family is “shared”).
2. The family of the mounting part defines no lines for dimensioning, but contains a solid with references at its extents. In this case the extents are dimensioned in all views. Or if
3. The family of the mounting part has neither lines nor a solid, but it creates a hole into the hosting element. In this case the mounting part is dimensioned together with the contour of the main element

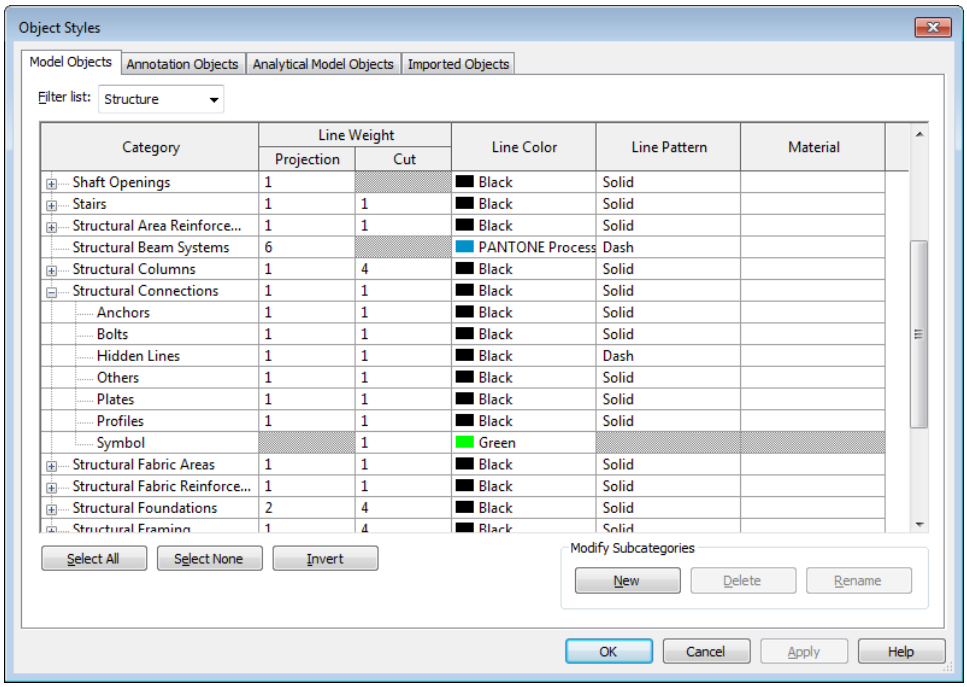
2. Define lines for the dimensioning

The lines used as references for the dimensioning on the shop drawing are standard model lines with a special sub-category defined for them. This subcategory or rather its name also defines in which views (seen from the mounting part) these lines are used to create dimensioning:

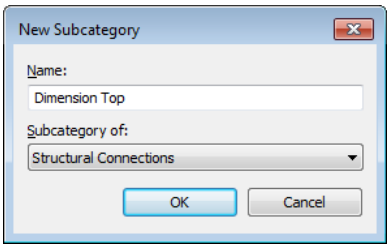
- *Dimension_Top* → View from top
- *Dimension_Front* → View from the front
- *Dimension_Side* → View from the side

2.1. Defining sub-categories

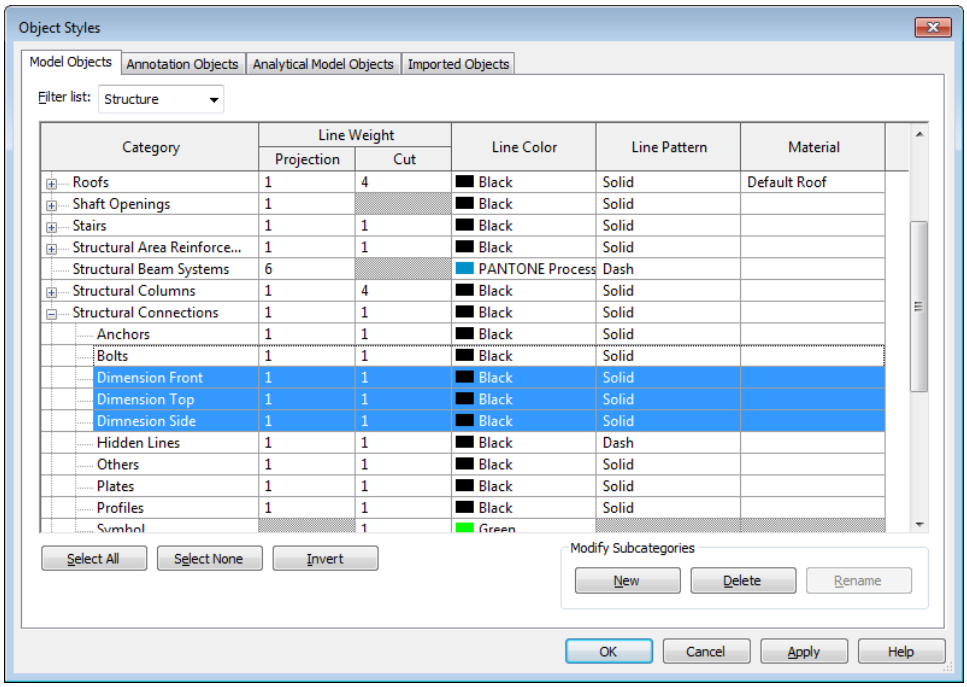
First one or more sub-categories with the desired names must be created: Manage → Object Styles:



The window contains a section "*Modify Subcategories*". Click on "New", and enter the desired name for the sub-category (see above for rules naming convention).



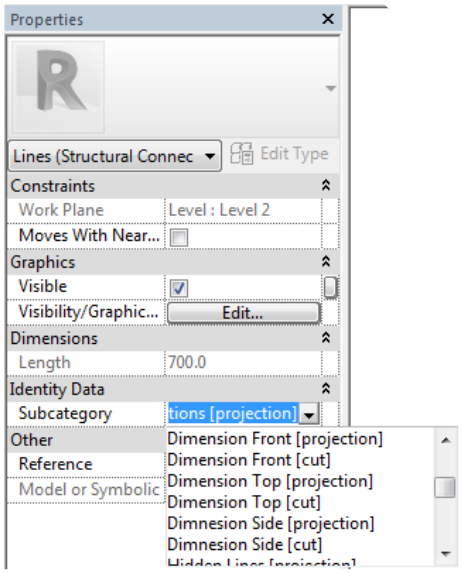
Repeat the steps in order to obtain the all necessary sub-categories.



Close Dialog *Object Styles* with OK.

Create a family that you will later on use in your project.

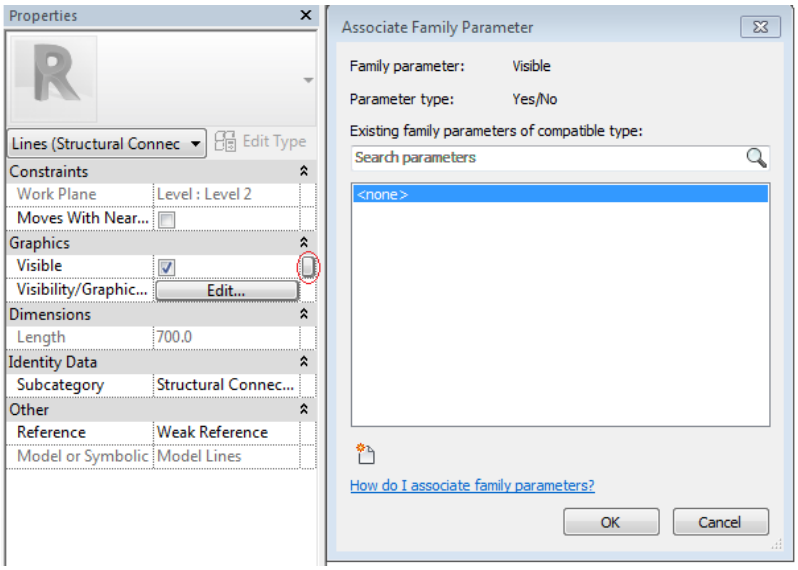
Now the lines can be assigned to this new sub-categories (i.e. create a model line in a *Structural Connection* family and select it).



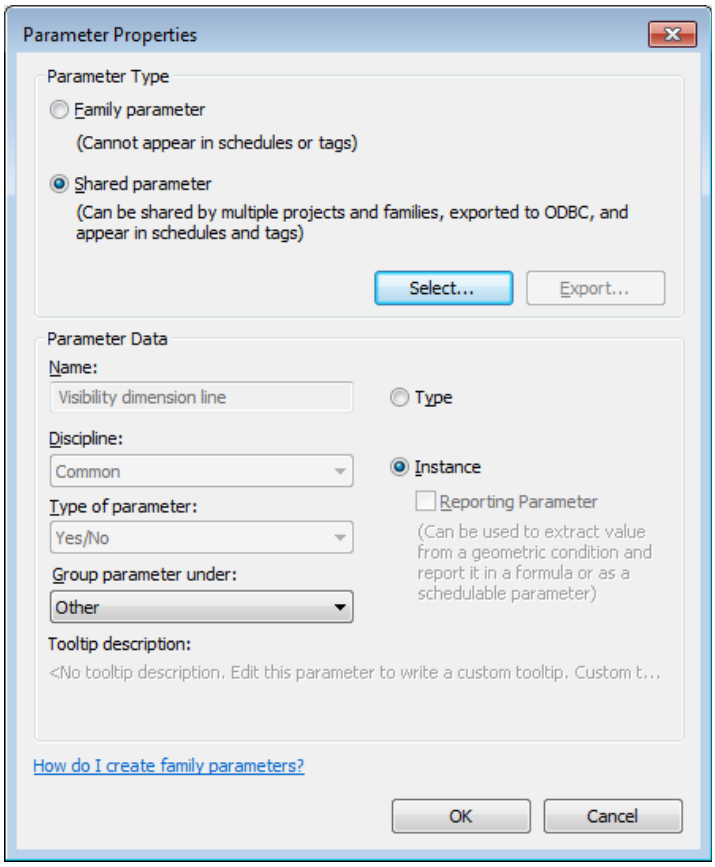
2.2. Define parameter for visibility

The line should be invisible normally so it does not change the appearance of the model. But to create the shop drawing, the line must become visible shortly (due to Revit limitation: cannot get

reference of invisible line). To achieve this, the line needs a shared parameter attached to its "Visible" property. In the properties of the line click the button next to the parameter "Visible":

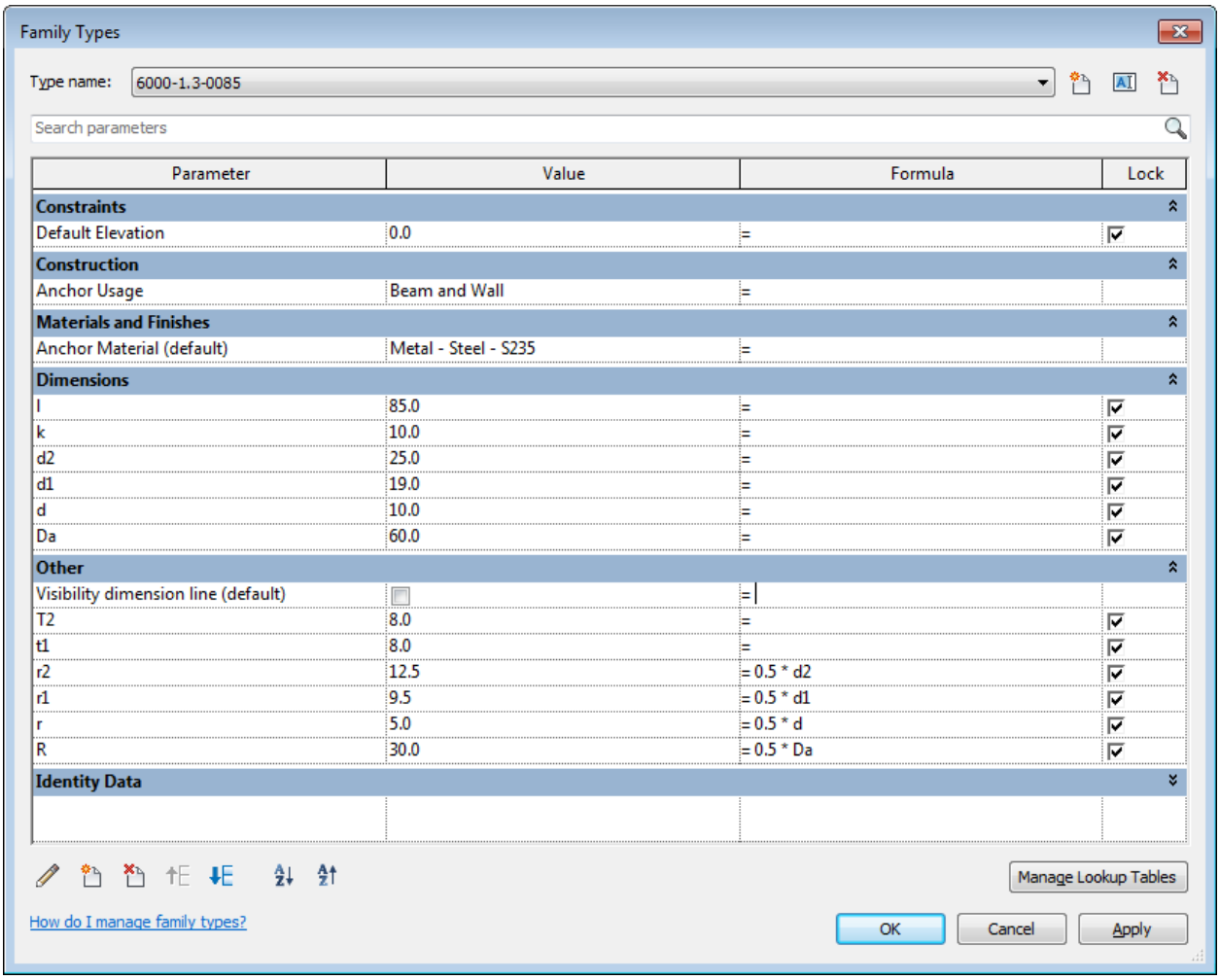


Select the parameter "Visibility dimension line" (if it is already in the list). If this parameter is missing, click on "Add Parameter" and add the parameter from the shared parameter file as an instance parameter.



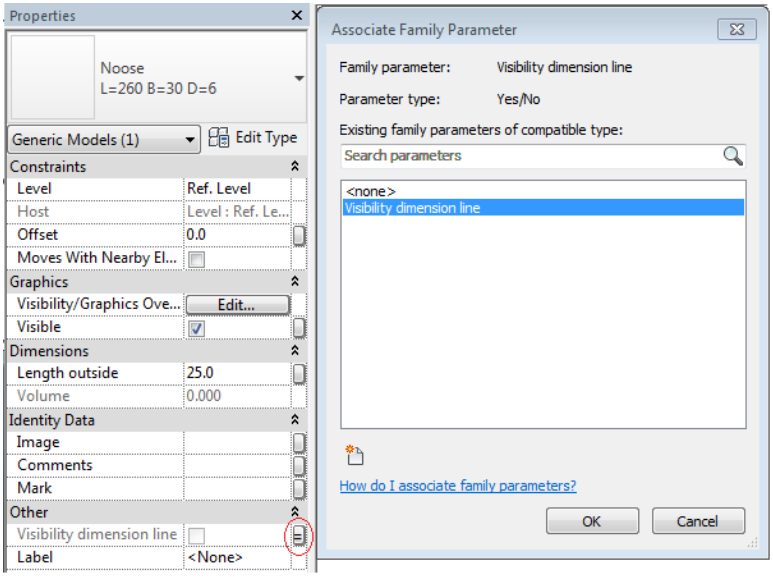
Close the dialog with OK, select the parameter in the parameter selection dialog and close this one also with OK. Repeat as necessary for all dimension lines.

Now this parameter must be set to not visible, i.e. “false”: Modify → Properties → Family Types. Below you there is an example for the Lifter family containing all the parameters, including the one for the visibility of the dimension line.



In the family types dialog uncheck the check box of the parameter “Visibility dimension line *(default)”* in the column “Value”, i.e. set the default value to not visible. Close the Dialog with OK.

If the dimension line was defined in a sub-family, this parameter must be passed through up to the main family (i.e. the family that is loaded into the Revit project). In the parent family select the sub family instance and click on the button next to the Parameter “Visibility dimension line” and add the parameter the same way it was described above. Below is shown an example for the loops inside the Nooses families used for connecting walls.



A dimensioning line in a view on the shop drawing is dimensioned only if the plane of the view is parallel to the plane of the mounting part in which the dimension line should be used (based on the name of the dimension lines sub category). If that is the case, it is dimensioned as follows:

- The line is in a plane which is parallel to the plane of the mounting part in which it should be dimensioned:
  This line is dimensioned in direction of the line, if the line is parallel to a main axis of the view on the shop drawing.
  Every line is used no more than once and will be dimensioned either horizontal or vertical.
  If the line is not parallel to one of the main axes, it is not dimensioned.
- The line is in a plane orthogonal to the plane of the mounting part in which it should be dimensioned:
  This line will always be dimensioned vertically and horizontally.
  This line can always be dimensioned, even if the line is not parallel to any axes of the view.

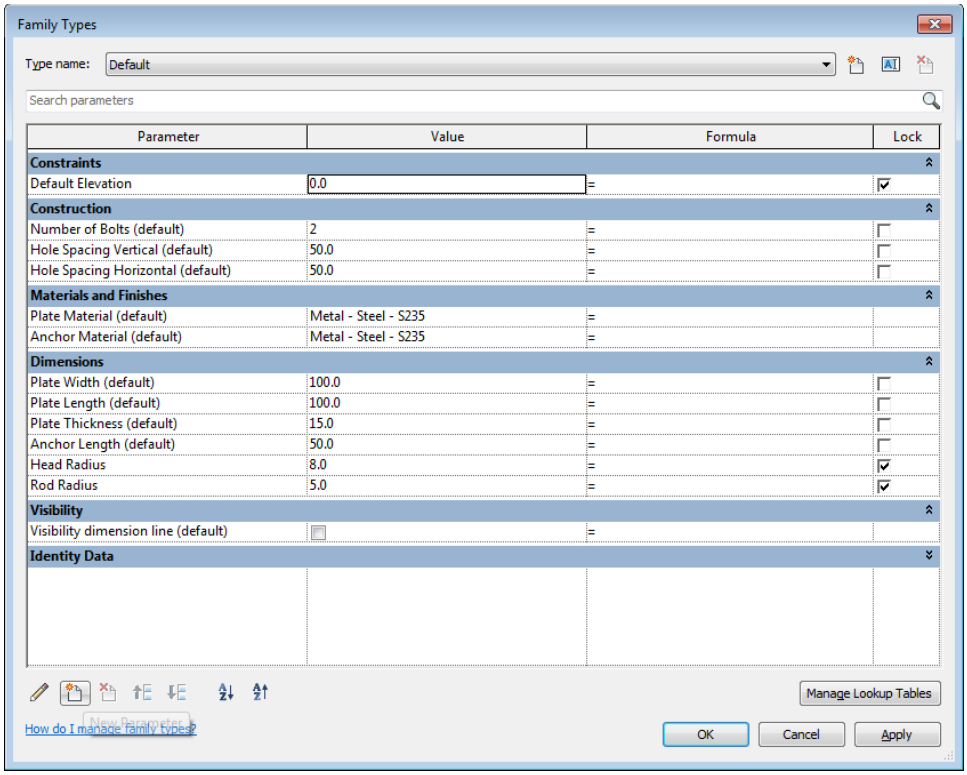
3. Dimensioning the extents of a mounting parts

In all views of a shop drawing, the extents are dimensioned, if a mounting part does not define dimensioning lines. The dimensioning of the extents does only work if the mounting part defines a valid solid. Bodies that are defined in the family as “Void” do not define a valid solid and as such cannot get there extents dimensioned. If a valid solid is there, the extents are dimensioned in all views (this will not work reliably for round edges or non-planar faces, since there might not be a reference at the extents of the element.

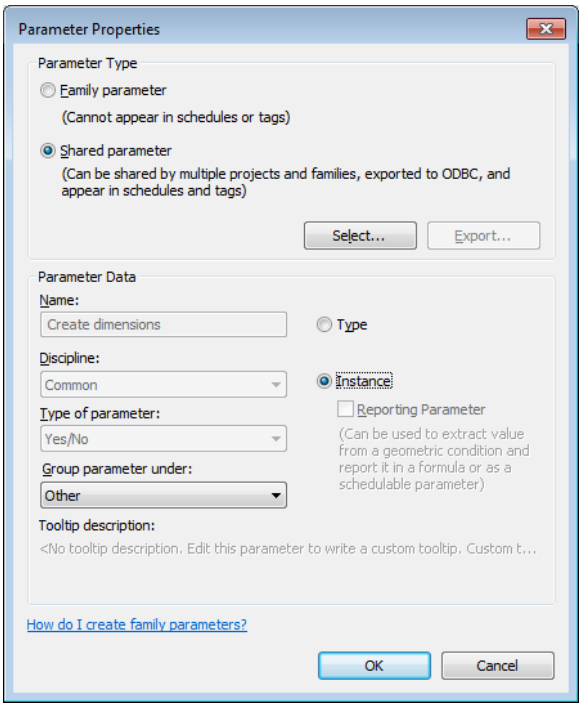
But if even one dimensioning line is defined in the mounting part, the extents will not be dimensioned at all (define dimension lines at the extents of the element if those are needed). If the mounting part cuts a piece out of an element, this hole is not dimensioned together with the main parts contour – unless the mounting part does not contain a valid solid at all.

As a result, mounting parts, which have dimensioning lines defined and do not contain a valid solid will be dimensioned twice: once with the dimensioning lines and once with the main elements extents. If a mounting part without dimensioning lines contains arc segments, the dimensioning can be quite wrong, since arc segments do not contain useful reference points.

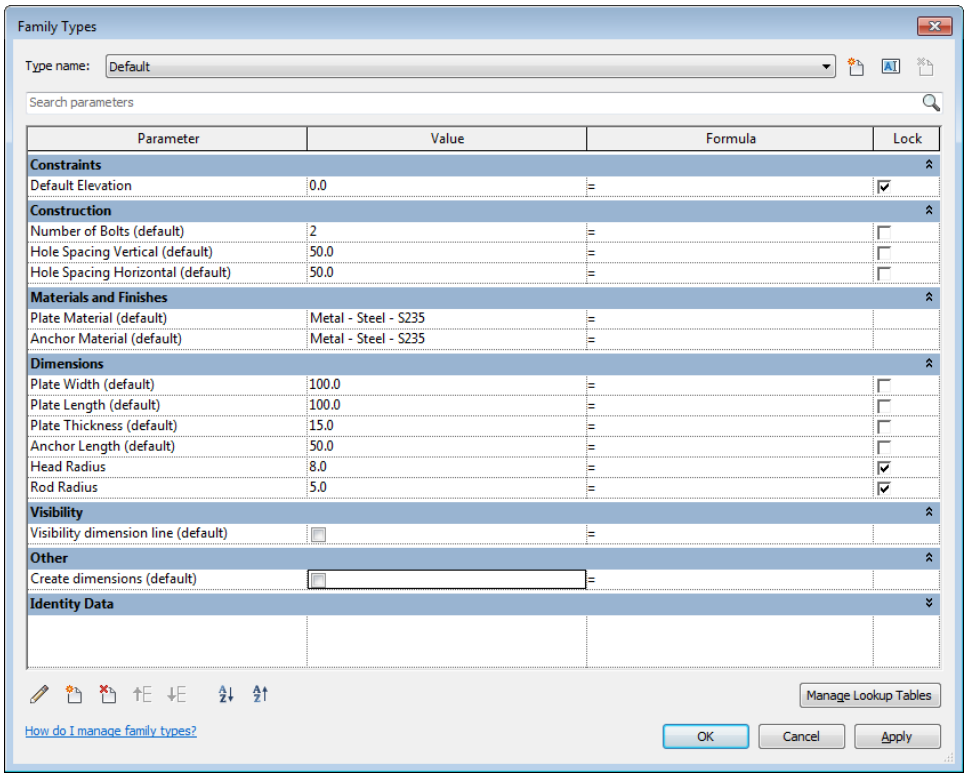
If a mounting part should not have its extents dimensioned, it can use the shared parameter "Create dimensions" as an instance parameter and set this to false. Modify Tab→ Properties Panel→ Family Types → New Parameter



Select "*Shared Parameter,* click on "Select", chose the shared parameter *"Create dimensions"*, select the option "Instance", and close the dialog with OK.



Now remove the check mark for the parameter in the column “Value”:



Close the dialog with OK. The Parameter is optional and is only needed, if no dimensions should be created. If the parameter is not defined, dimension lines are created.

4. Dimensioning mounting parts with the host element contour

Mounting parts without dimensioning lines and solids are dimensioned with the host elements contour, if the mounting part creates a hole. This is just a makeshift solution, based on the fact that Revit does hide void geometry from the API. All corner points of the resulting host elements contour are dimensioned. Exception: Recesses which are not going through the element and are placed at the edge of the element are not dimensioned.

5. Dimensioning equal mounting parts on one dimension line

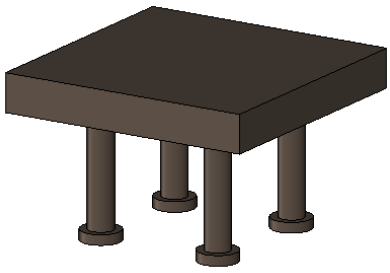
The dimensions of all mounting parts of the same family are dimensioned on one line, if they are placed on the same side of the host. Controlling for this is the center of the element. All mounting parts with the same family name on top of the center of the host are dimensioned on top of the element. Mounting parts below the center are dimensioned below the element. The same goes for the vertical dimension lines. Dimension points on the extents of the host are added to the dimension line, additionally to the dimension points of the mounting parts (one point left & right or top & bottom)

Workflow example:

The objective is to have the center of anchor plates dimensioned in the view from top in both directions.

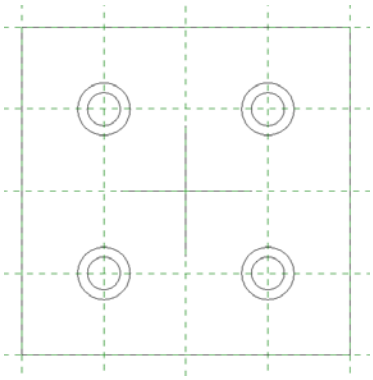
Step 1. **Open/Crete your mounting part family**

Open the anchor plate family, which can be found at the following location: *Libraries\US Metric\Structural Connections\Mounting Parts\Fastening Plates\M_Plate with Anchors.rfa:*



Step 2. **Dimensioning using model lines**

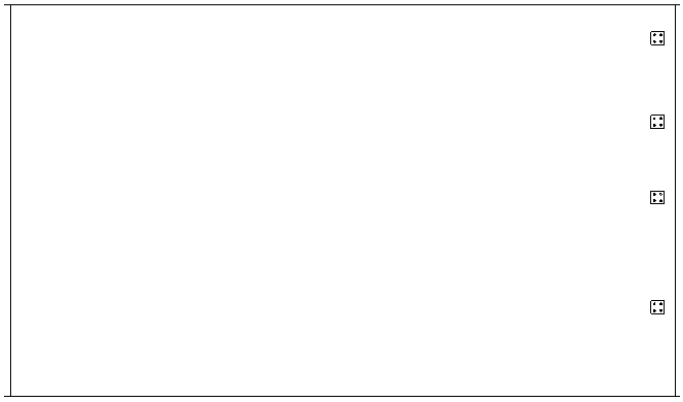
Switch to the Floor plans Ref. Level and create one horizontal and one vertical model line, intersecting in the center of the anchor plate:



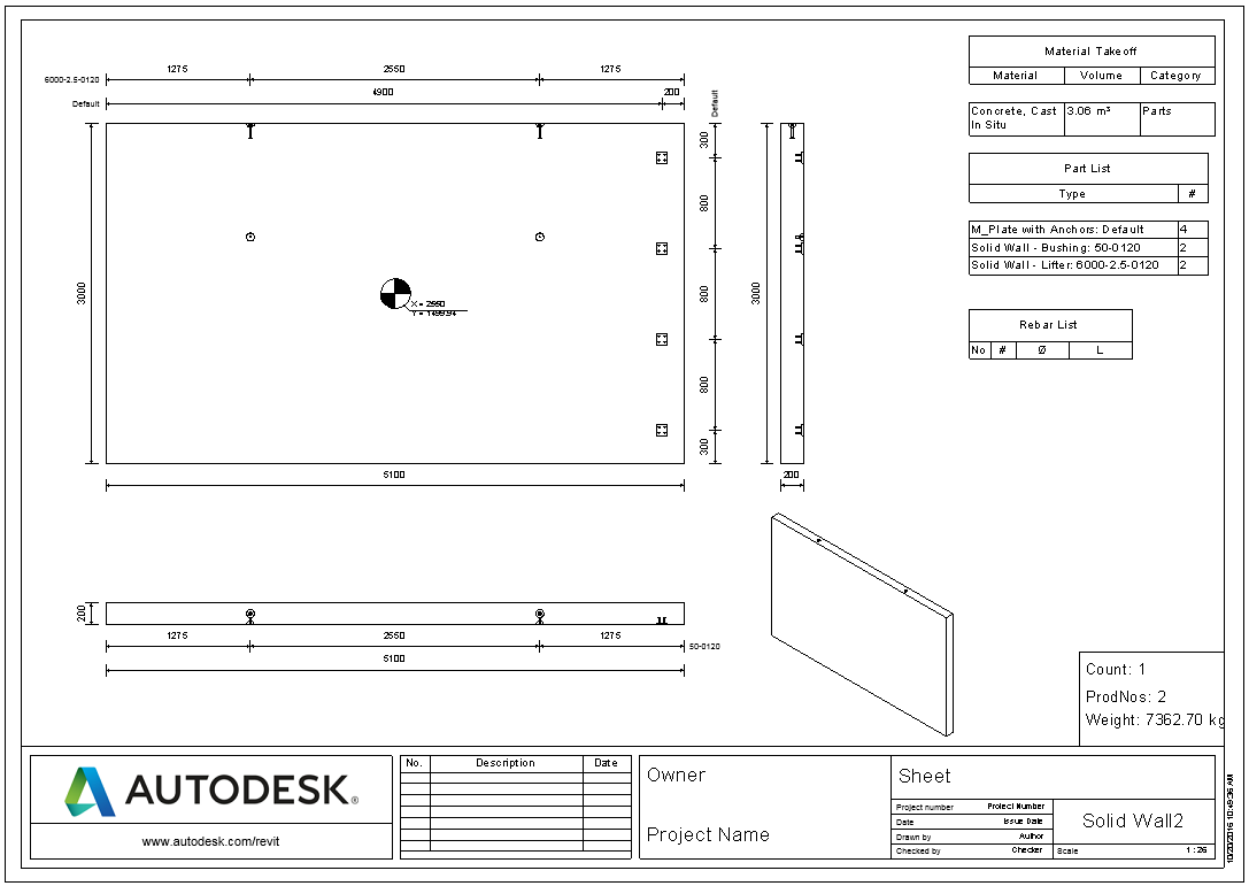
Define the sub category "*Dimension_Top*" and set it as the sub category for those two lines (see chapter 1). Now define the instance parameter "Visibility dimension line" and set the lines to invisible (see chapter **Error! Reference source not found.**).

Step 3. **Load the family into a project**

Load this family into a project and place it on a wall.

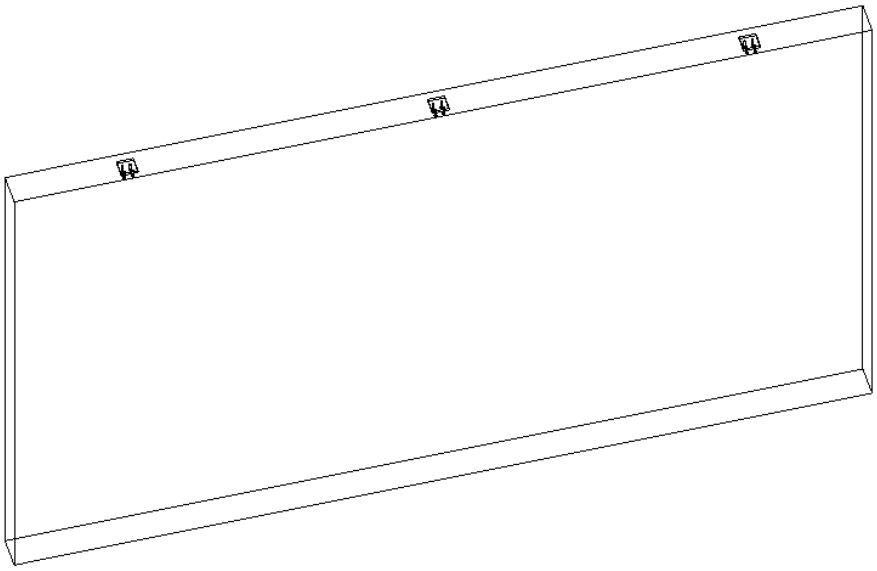


Split the wall, use the *Mounting Parts* command to add the anchor plates into the assembly and create a shop drawing for it. The center of the anchor plates is dimensioned in the view from top (on one line, since they are of the same family):

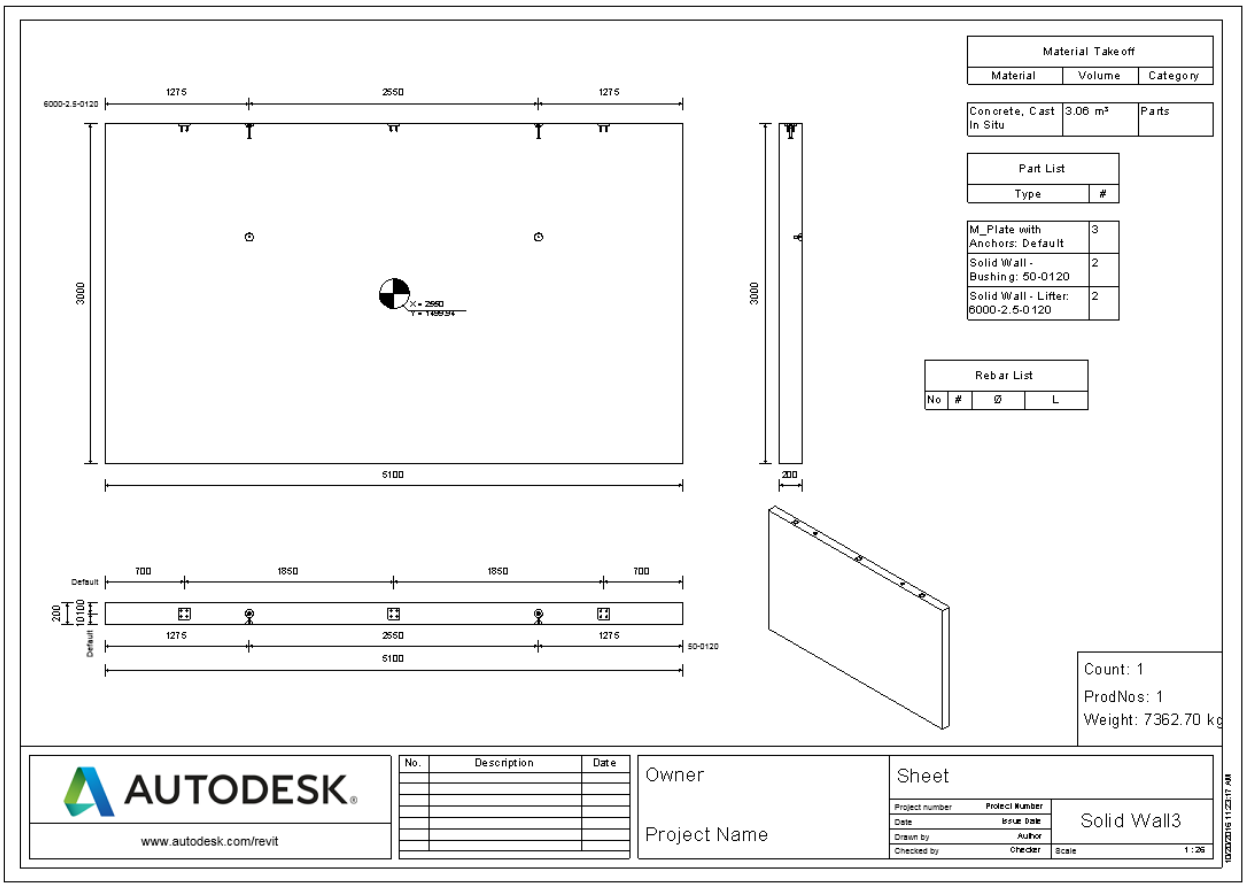


Based on the name of the sub category, the dimension lines should be created only when we are looking from the top (as seen in the family) at the anchor plates. For this wall that's only in the frontal view of the wall.

If now the plates are added to the top face of the wall:



Follow the same procedures as above. The families are now dimensioned only in the top view of the drawing:

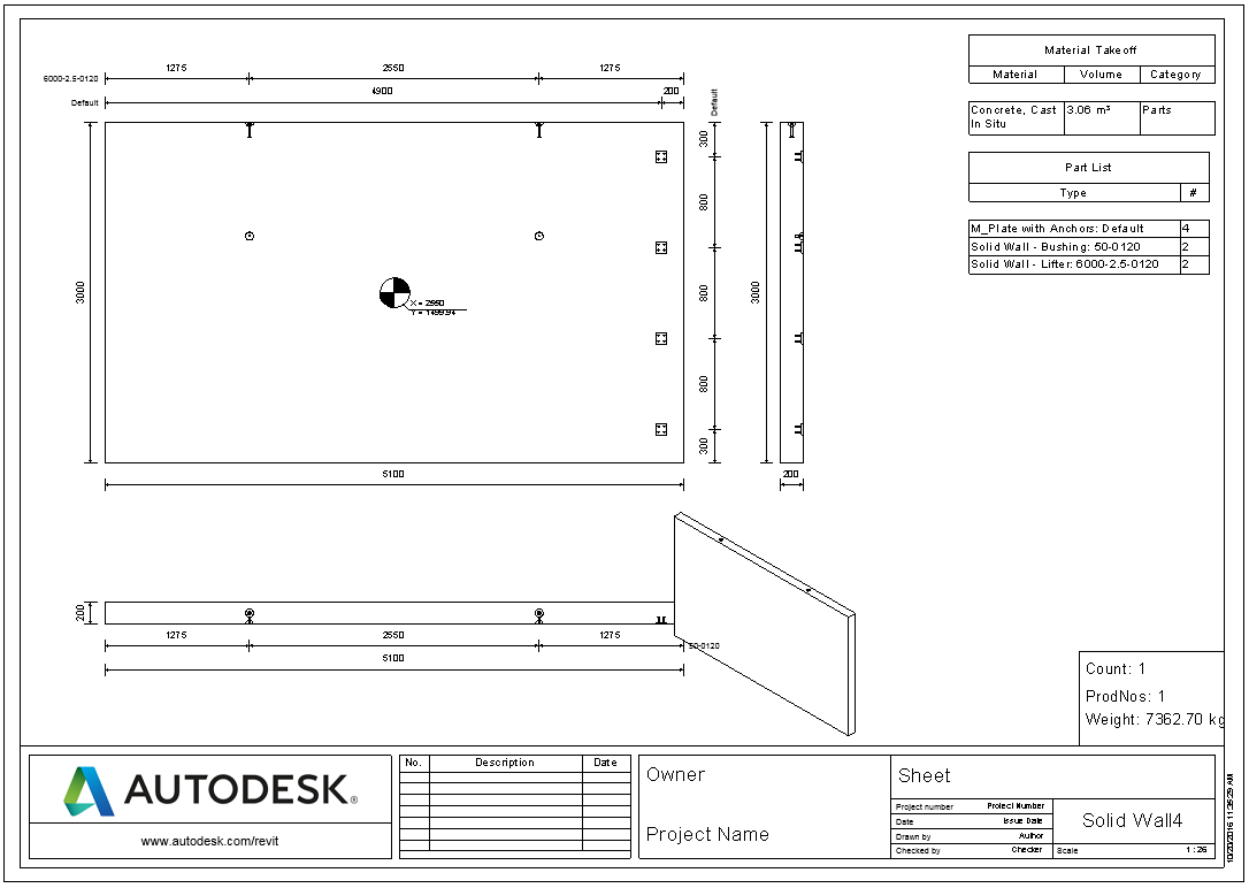


Step 1. **Dimensioning using only one dimensioning line**

Instead of two dimensioning lines intersecting in the center, we now create just one line, which is orthogonal to the plane of the anchor plate and is placed in the center of the family:

As before we need to set the sub category (see chapter 1) and the visibility parameter (see chapter **Error! Reference source not found.**).

When this family is used as before, it is dimensioned the same way as in the first example:



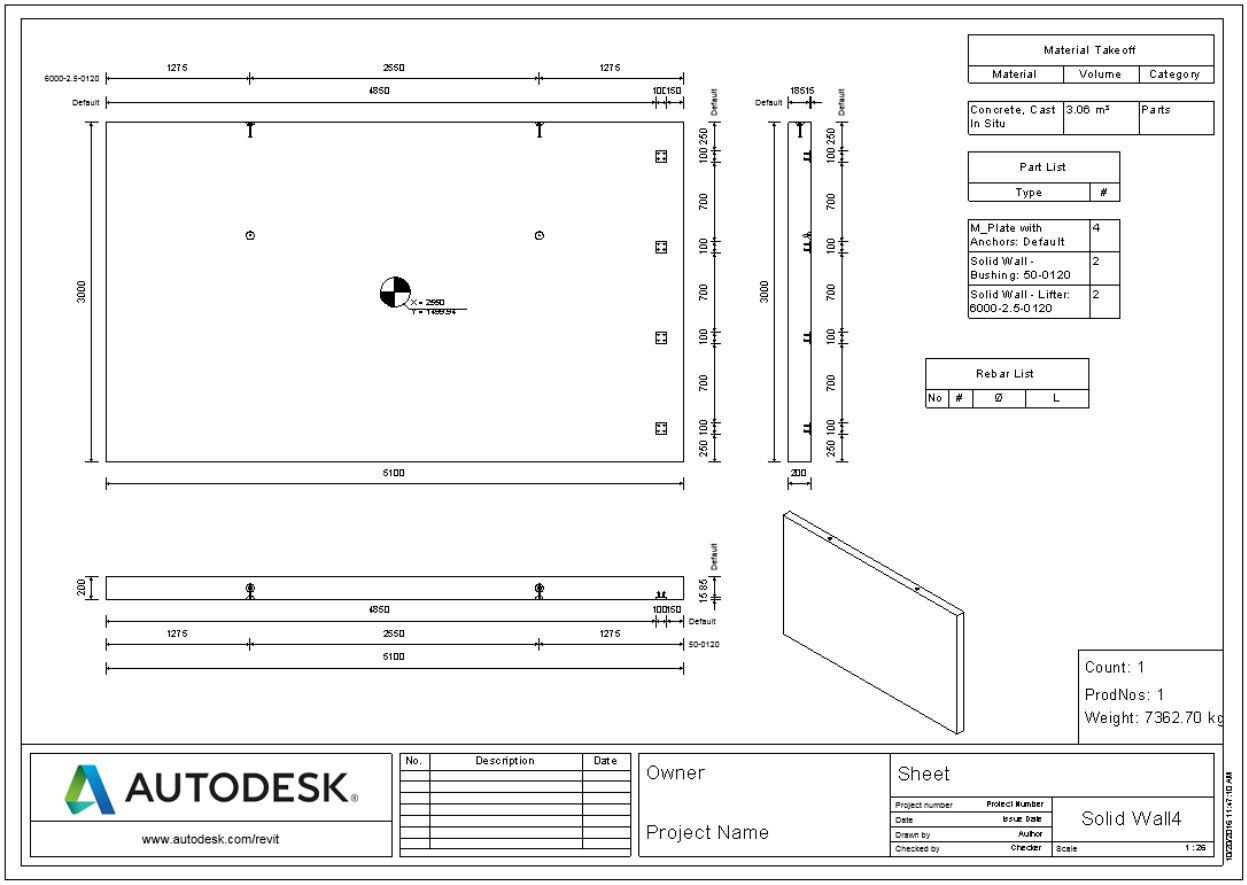
But there are two important differences:

- In the first example it would have been possible to create only one line, getting dimensions only vertical or only horizontal; in the second example always both directions are dimensioned.

- If the anchor plate is added rotated in the first example, it cannot be dimensioned at all; in the second example the center of the plate is always dimensioned.

Step 2. **Dimensioning using the contour of the family**

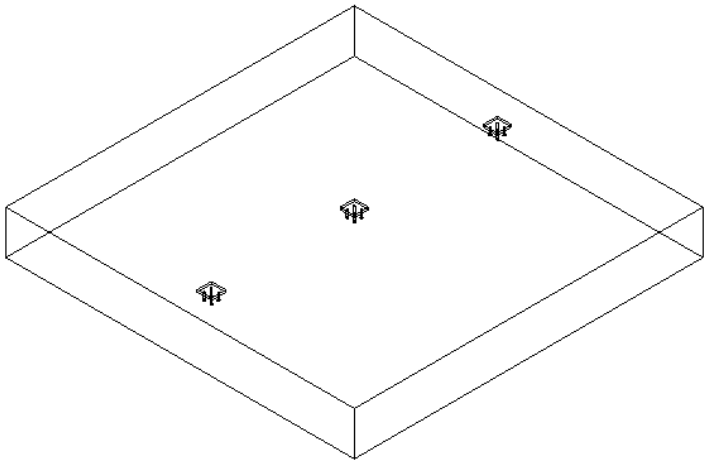
Remove the dimension lines and load the family into the same project, and create new shop drawings. Now the extents of the anchor plates are dimensioned in all views (using one dimension, since it's always the same family).



In top and front view only the plate without the anchors is dimensioned, because in these views there are only round faces of the anchors visible thus giving no references to dimension.

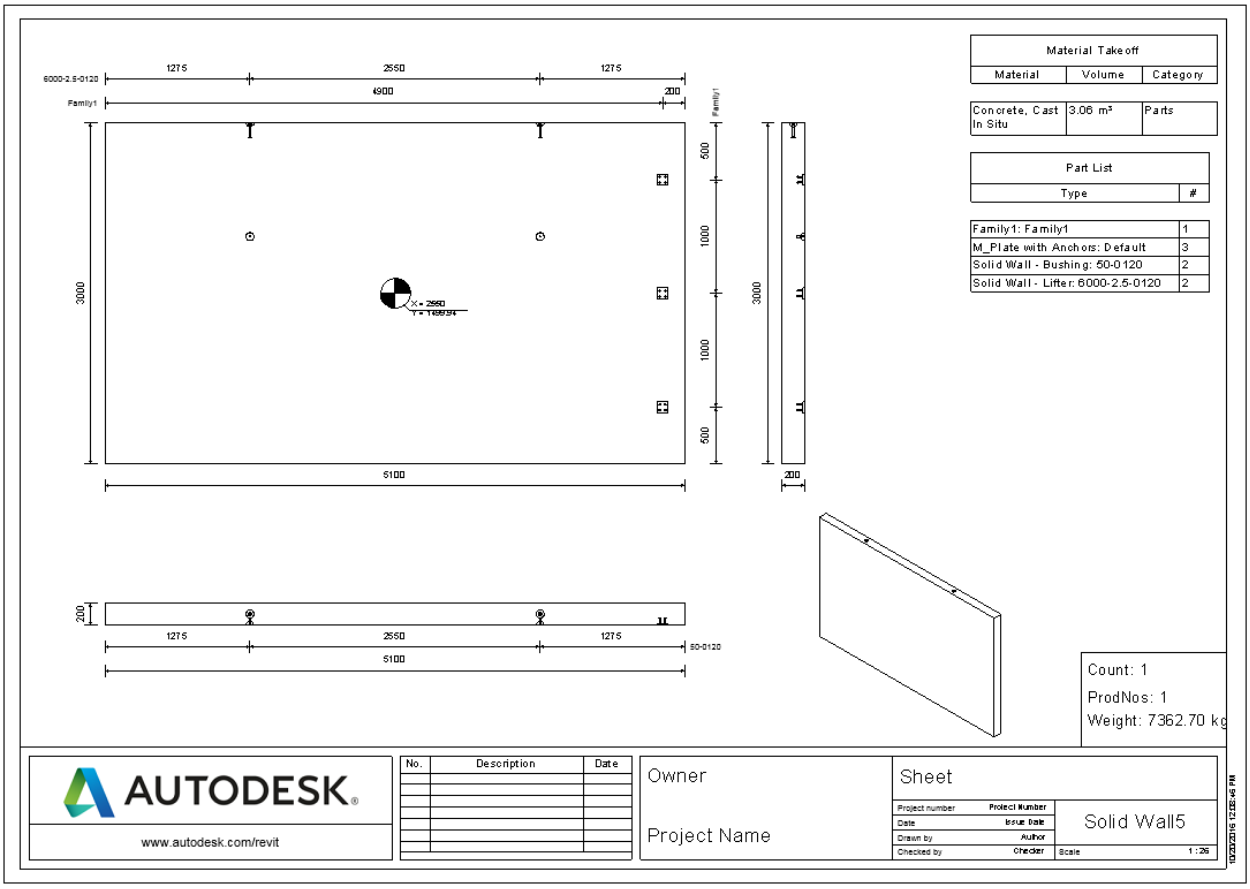
Step 3. **Dimensioning using sub-families**

Create new Family → "Metric Generic Model face based" (C:\ProgramData\Autodesk\RVT 2017\Family Templates\English). Load the anchor plates with dimensioning lines into the family and place it three times on the top face of the families' body:



The visibility parameter must be passed through into the new family (see chapter **Error! Reference source not found.**). Now the new family is placed in the wall.

Split the wall and create a shop drawing. The anchor plates are only dimensioned in the front view:


Material Takeoff
Material | Volume | Category
Concrete, Cast In Situ | 3.06 m³ | Parts
Part List
Type | #
Family1: Family1 | 1
M_Plate with Anchors: Default | 3
Solid Wall - Bushing: 50-0120 | 2
Solid Wall - Lifter: 6000-2.5-0120 | 2
Count: 1
ProdNos: 1
Weight: 7362.70 kg
AUTODESK
www.autodesk.com/revit
No. | Description | Date
Owner
Project Name
Sheet
Solid Wall5
Scale 1 : 26