Getting Started Guide

CONTENTS

INTRODUCTION

This starting guide is a brief introduction to working with Project Bran for Revit, describing the basic methodology and is not meant to replace formal training.

The examples presented in this guide are generic for worldwide use and do not conform to local or specific company standards.

The chapters described in this guide will help you learn how to define families for walls and slabs, use those families to divide walls and slabs into producible elements, create assemblies and configure parts. The chapters also include instructions on how to use and understand the commands in the Project Bran user interface and how to create shop drawings for both walls and slabs.

ABOUT PROJECT BRAN

Project Bran is a leading edge precast concrete construction application. It provides a simple, user-friendly working environment for creating 3D structural models from which drawings are created.

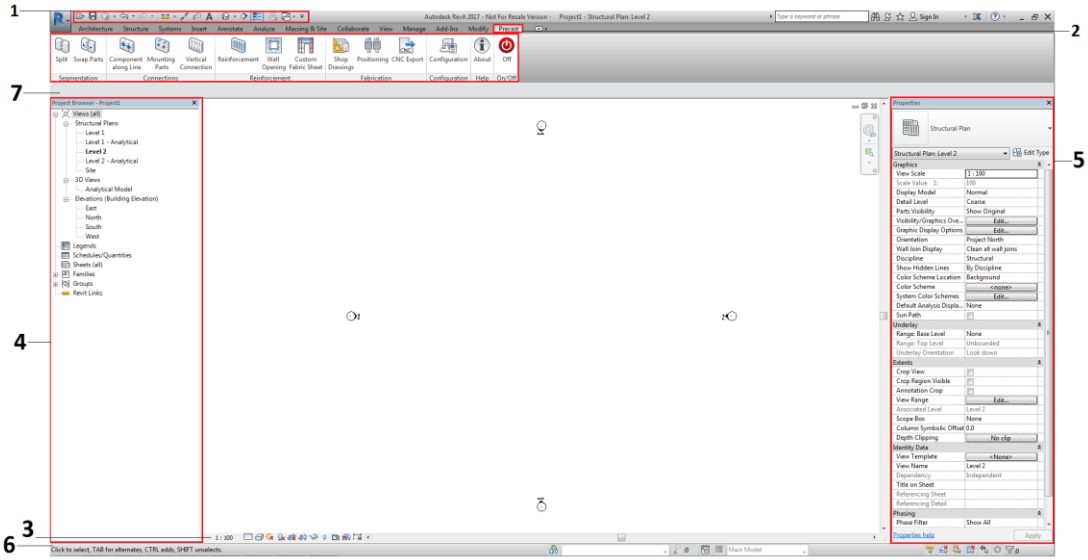
The Revit model forms the basis of the 3D construction. Complex assemblies are created using Project Bran commands (such as split or opening reinforcement) with all the required features, reinforcement and connections.

Once a model is complete, Project Bran creates drawings using a large selection of view production tools, dimensions, locations, symbols and automatic layout functions.

PROJECT BRAN USER INTERFACE

GENERAL

The Precast tab will be added alongside the other existing Autodesk Revit® tabs after installation. Project Bran is fully integrated in Autodesk Revit® and it uses the usual Revit interface and tools along with the Precast tab specific tools.



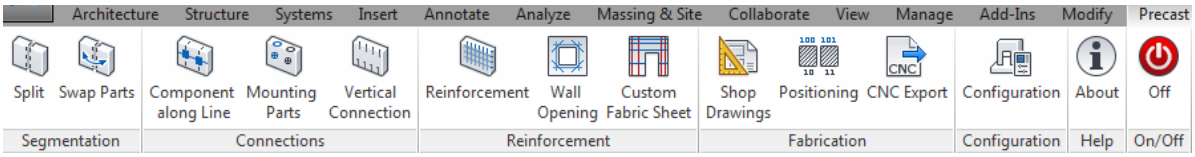
1. Quick Access Toolbar

The Quick Access Toolbar provides fast access to the most frequently used tools. The set of available tools can be extended.

To add a Project Bran ribbon button to the Quick Access Toolbar, right-click the ribbon button and select *Add to Quick Access Toolbar*. It can be placed above or below the ribbon.

2. The Precast Ribbon Tab

On the Precast ribbon tab, the buttons are grouped on one row and include large buttons for the available functionalities.



3. View Control Bar

The View Control Bar is located at the bottom of every view and provides quick access to functions that affect the current view.

4. Project Browser

The Project Browser is a tree folder containing all views, legends, schedules, sheets, families, groups and links.

5. Properties Palette

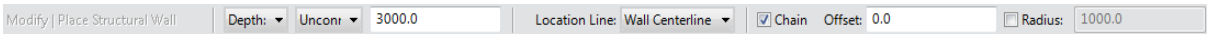
The Properties palette is a modeless dialog where you can view and modify the parameters that define the properties of elements in Revit.

6. Status Bar

The status bar is located along the bottom of the application window. When you are using a tool, the left side of the status bar provides tips or hints on what to do. When you are highlighting an element or component, the status bar displays the name of the family and type.

7. Options Bar

The Options Bar is located below the ribbon. Its content changes depending on the current tool or selected element.



Useful tips for working with Project Bran:

- To cancel a command, press the Esc key.
- The current command and prompts are displayed in the command line window at the bottom of the screen.
- When you hover the cursor over a ribbon button, a tooltip appears.
- The Undo command on the Quick access toolbar cancels one or several commands.

PROJECT BRAN SPECIFIC TOOLS

Split: Precast tab → Segmentation panel →  (Split). Use this command to divide walls and slabs into producible panels by selecting one or several walls and/or slabs.

Swap Parts: Precast tab → Segmentation panel →  (Swap Parts). With this command you can exchange the position of two part elements within the original element. The entire original element will be segmented after the new dividing rules.

Component along Line: Precast tab → Connections panel →  (Component along Line). With this command you can insert a component along a defined model line. Whenever the model line intersects an edge of a part, a selected component will be inserted and added automatically to an existing assembly.

Mounting Parts: Precast tab → Connections panel →  (Mounting Parts). This command automatically searches within an assembly to find any user mounting parts (lifters, loops, bushing etc.) which should be assigned to an assembly. In the same time it also adds parameters regarding the face or the layer in which the mounting part is added.

Vertical Connection: Precast tab → Connections panel →  (Vertical Connection). Using this command you can add connections (e.g. grout tubes) to join the top and/or bottom parts of a wall.

Reinforcement: Precast tab → Reinforcement panel →  (Reinforcement). This command inserts the defined reinforcements for all selected assemblies or parts from the same element type.

Wall Opening: Precast tab → Reinforcement panel →  (Wall Opening). If you open a project where openings are already inserted without the Precast Tools, you can select the openings and start this command. The same applies for newly produced openings. The command adds reinforcement for the opening based on your configuration and also edits the already placed opening reinforcement.

Custom Fabric Sheet: Precast tab → Reinforcement panel → (Custom Fabric Sheet). Using this command you can add custom fabric sheets to walls and slabs.

Shop Drawings: Precast tab → Fabrication panel → (Shop Drawings). To create shop drawings select the elements and use this command. For each selected position a shop drawing is created, using a template selected from the Configuration. The views on the shop drawing are scaled to fit the available space.

Positioning: Precast tab → Fabrication panel → (Positioning).This command helps you renumber the existing assemblies based on your preconfigured starting number.

CNC Export: Precast tab → Fabrication panel → (CNC Export). Use this command to create machine data files for production. A Unitechnik format production file is created for each selected assembly.

Configuration: Precast tab → Configuration panel → (Configuration). Using this command you can configure settings for walls, slabs and built in parts. These 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.

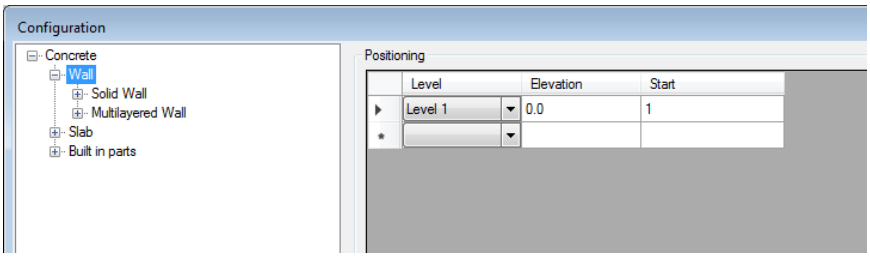
About: Precast tab → About panel → (About). This button provides additional version, copyright, and licensing information regarding the currently installed version of Project Bran.

On/Off: Precast tab → On/Off panel → (On/Off). This button Enables or Disables Project Bran in Revit. Turn Project Bran off to replenish hardware resources and improve Revit performance.

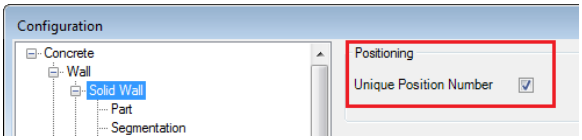
WALLS

SETTING UP PROJECT BRAN

1. **Positioning**: For each floor you can assign a different starting number for the walls.

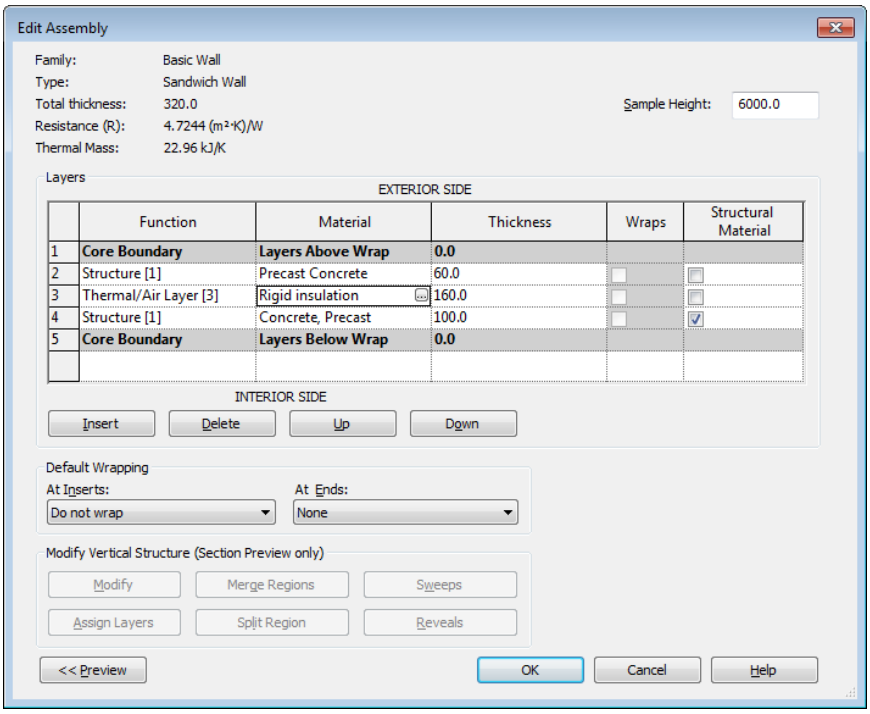


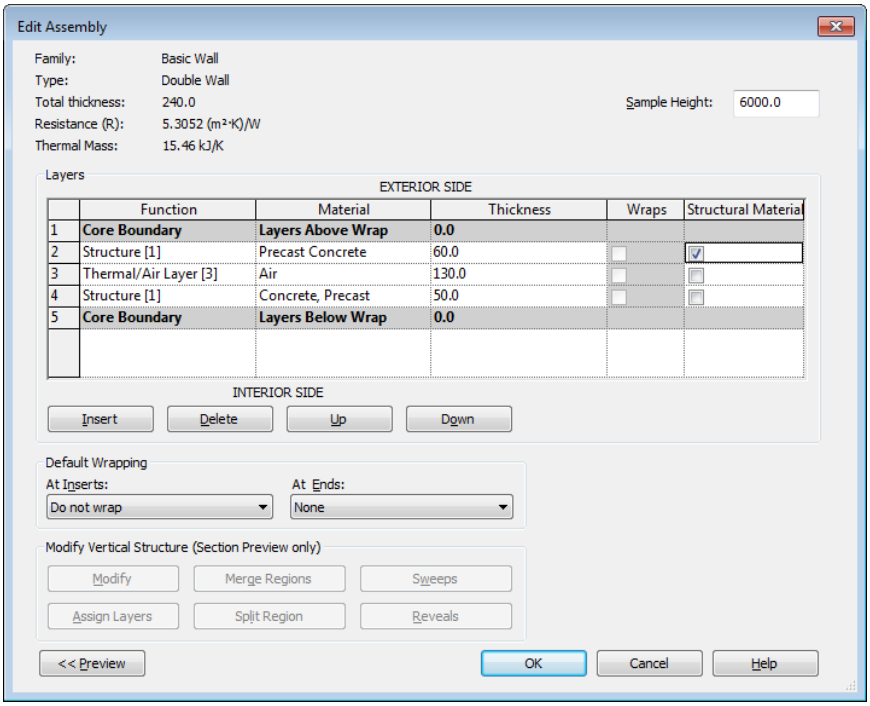
2. **The Unique Position Number Box:** The first thing you can specify is the unique position number for each type of precast element. This is taken into account when the software assigns position numbers. This option is useful when you want to have different numbers for the same assembly. The unique position number option is available when selecting any specific precast element inside the Configuration dialog:



3. **Wall Creation**: There are two conditions to create solid precast wall panels out of walls inside Revit:
   - The wall has to be a structural one.
   - At least one layer of the wall must be set to Structural Material (even if only one layer is defined). This layer must have a material defined and the material must have a density assigned.

4. **Sandwich / Double Wall Creation**: To create a sandwich wall or a double wall, perform the following steps:
   1. Create a structural wall.
   2. In the element Properties click Edit Type → Duplicate → type a name (e.g. Sandwich / Double Wall) → Edit the Structure parameter:
   3. Define your layers:
      A sandwich / double type wall needs to have three layers. In the Edit Assembly dialog, click Insert to add layers. The first and last layers are Structure layers. For the middle layer, select the Thermal/Air Layer function.
   4. Define the material and thickness of each layer.
   5. For the third layer check the Structural Material checkbox.
   6. Click OK.





Note: For double walls, the first layer is always assigned to the structural layer.

5. **Cover for Walls**: Before using any of the Project Bran tools, you need to define the cover properties for each element.

Note: If the user does not use the precast template (C:\ProgramData\Autodesk\Project BRAN 2017\Families\en), whenever pressing any command Project Bran will load all the needed families so that it can populate the configuration with default values.

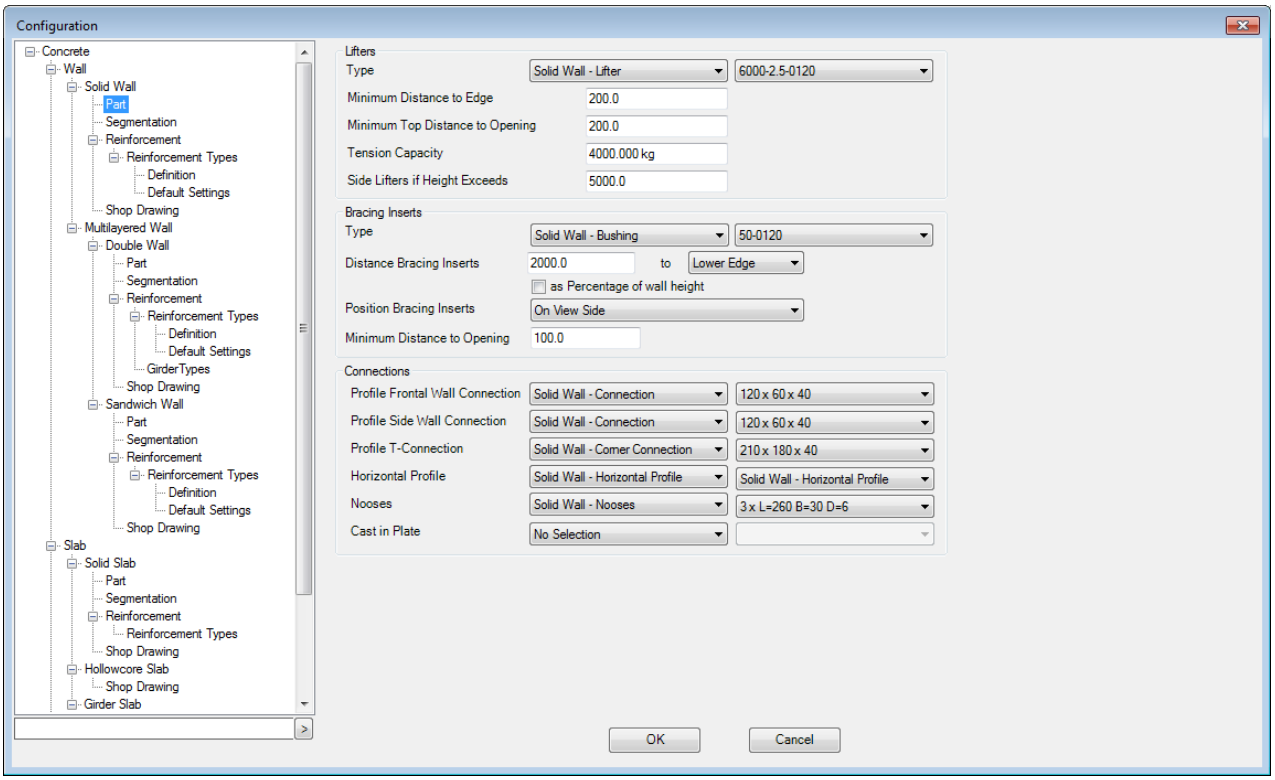
TEMPLATES AND FAMILIES

File locations: When installing Revit Project Bran, you can set a path for the template and family files. The default value is C:\ProgramData\Autodesk\Project BRAN 2017\Families\ and, according to the language, there are the “en” folder for English, the “de” folder for German and the “fr” folder for French.
All templates for the walls and slabs are stored in folders in the Families folder. These are:

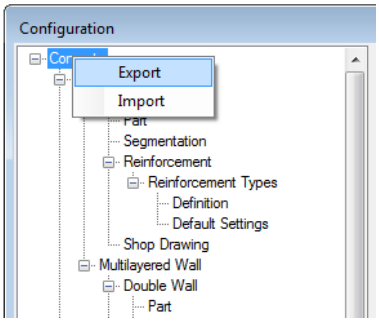
- The annotations are in the Annotations sub folder of each element.
- The connection components, lifters and mounting parts are in the Mounting Parts sub folder of each element.
- The profiles for the connections are in the Profiles sub folder of each element.
- The symbols for the hollow core slabs are in the Symbols sub folder of each element.
- The rebar families needed for reinforcing the walls are stored in the Revit folder.
- The examples for title blocks are in the Title Blocks sub folder of each element.

CONFIGURATION

When using the Configuration command, a dialog opens allowing you to change different settings:



Open an external XML file with different settings using the Open button: To create such an external configuration file select your desired category from the tree on the left side of the Configuration window and right-click to choose the Export option. You can only import or export the entire tree.



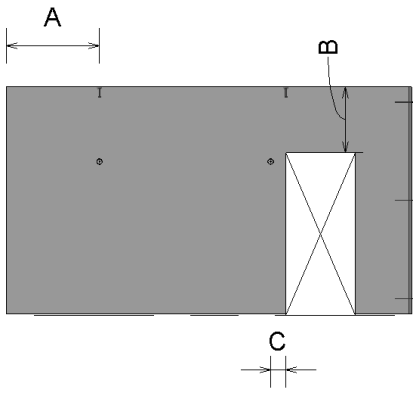
Therefore, you can create different configurations for different factories or other situations. You can load these configurations into projects individually.

PART

Inside the Part branch, you can set several properties.

Lifters section

In this 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.



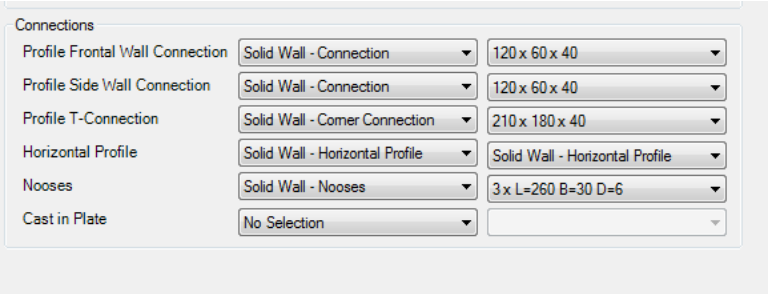
Bracing Inserts section

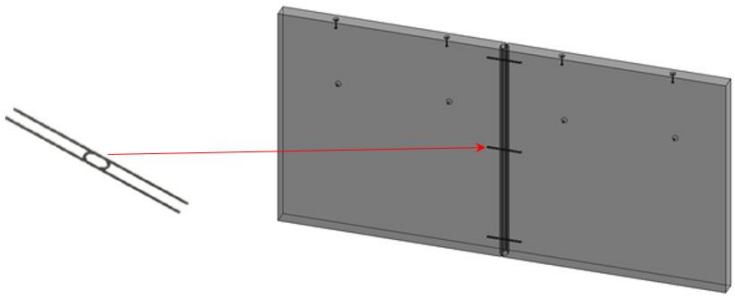
In this section you can specify the position of bracing inserts, relative to the lower or upper edge of the wall or the minimum distance to opening (C). You can specify this either as a distance or as a percentage of the wall height. Bracing inserts can be added either on a specific side of the wall or on both sides of the wall. The bracing inserts are usually added in pairs (2, 4…) and aligned with the position of the lifters (when possible). Parameters set in this section are taken into account during segmentation. The Bracing Inserts section is only available for precast walls.

Connections section

In this section you can choose the profiles types that should be applied either where the segmentation is performed between two walls or at wall junctions. This section is only available for the Solid Wall and the Sandwich Wall types.

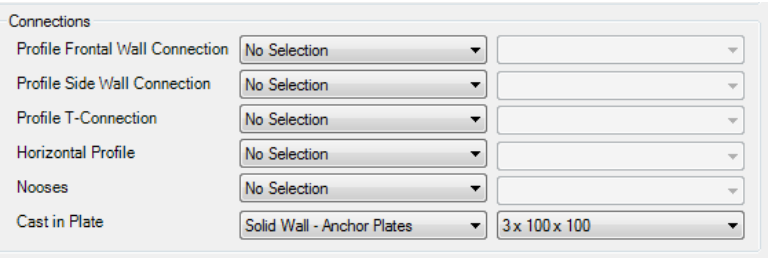
The following configuration inside the Connections section is used when you want to create a connection using Nooses (Loops):





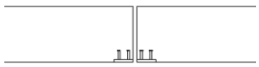
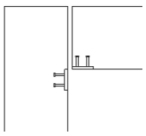
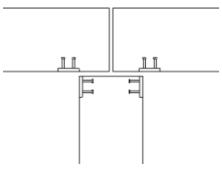
For the connection with loops, the program creates a profile and the loops according to the predefined families. In the family it is defined how many loops should be created, depending on the height of the wall.

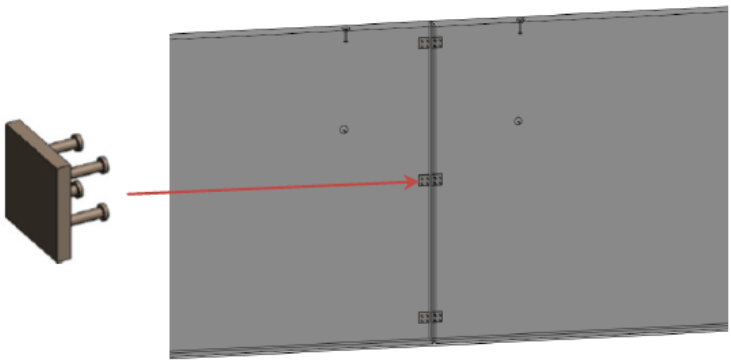
In order to create connections using anchor plates, the following configuration should be used:



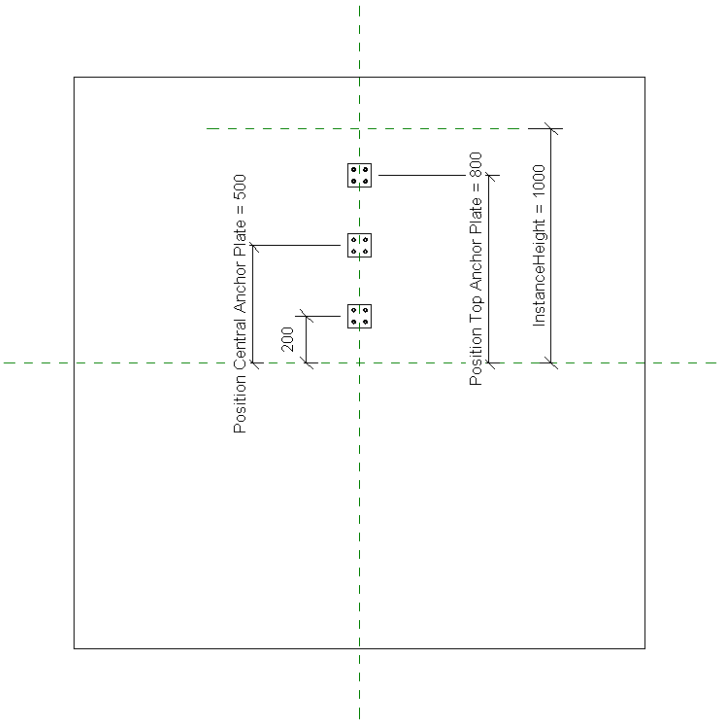
For the connection with anchor plates, the program inserts the plates according to the predefined families. Depending on the panel height, two or more plates are created. The rules can be defined in the family.

The program automatically performs the following connections:

- Straight connection: 
- Connection at 90°: 
- T-connection: 

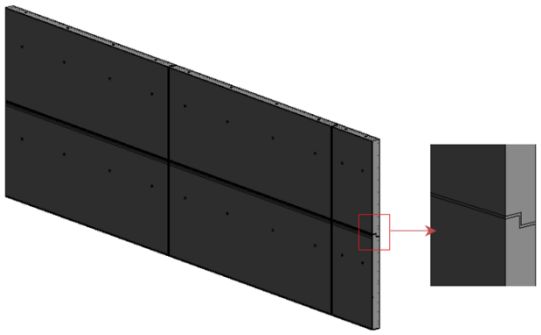


The default families provide three loops/anchor plates distributed equally throughout the height of the wall.



You can use these provided families to customize the number of anchor plates/nooses needed for the connection. In addition, you can define your own family, using the same principle as found in the families that come with Project Bran.

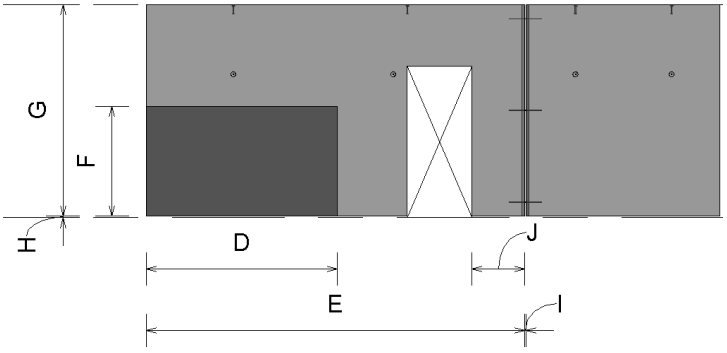
Horizontal Profile families can be set, in order to perform the segmentation. If the wall is higher than the maximal wall height the software will automatically divide the wall into layers, horizontally:



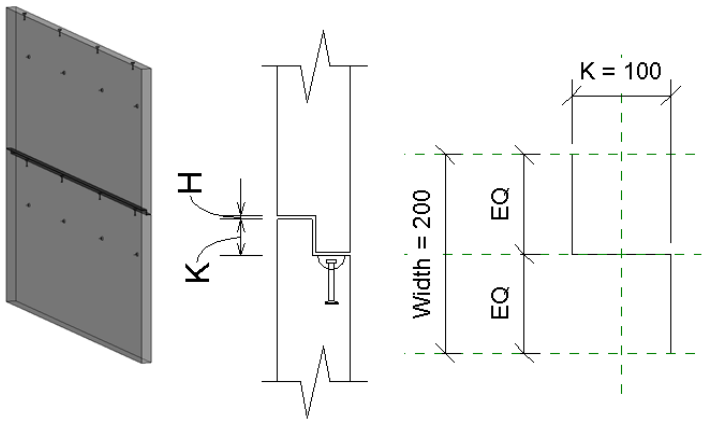
SEGMENTATION

The Segmentation section is available for the Solid Wall, Double Wall and Sandwich Wall, where you can set different appropriate parameters for each type of these elements.

The most important parameters for dividing the walls are the minimum (D) and maximum wall length (E) and height (F and G) as well as the maximum panel weight or the minimum distance to opening (J). The dividing will always be within the limits of the parameters. The horizontal gap (H) is the gap between the panel and the bottom of the wall, or the distance between two adjacent vertical walls. The vertical gap (I) represents the gap between two vertical adjacent wall panels.



The Profile Overlap Height (K) parameter in the Segmentation tab represents the height on which two walls on top of one another are overlapping. In order to obtain the maximum height per element, you should synchronize this parameter with the height of the profile in the actual division profile family which can be set in the Part tab for the Solid Wall.

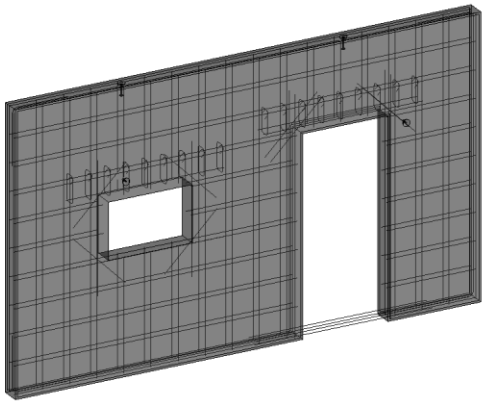


REINFORCEMENT

In the Reinforcement section, you can define reinforcement types and use them for different situations, specified in the Default Settings.

REINFORCEMENT TYPES - DEFINITION

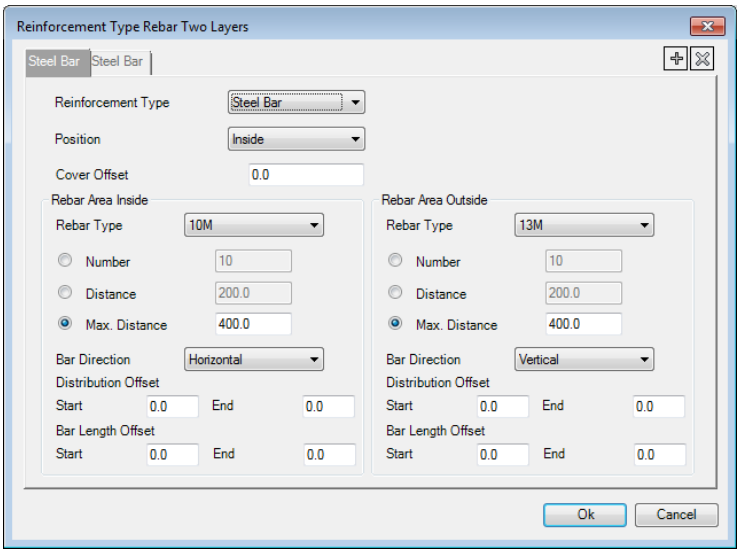
In this tab you can define as many reinforcement types as needed for area or opening reinforcements.



Using the Reinforcement section in the Configuration dialog, you can define any type of wall reinforcement to fit your requirements. Depending on your configuration, Project Bran automatically provides a dialog populated with Area and/or Edge Reinforcement to the wall, according to your predefined rules.

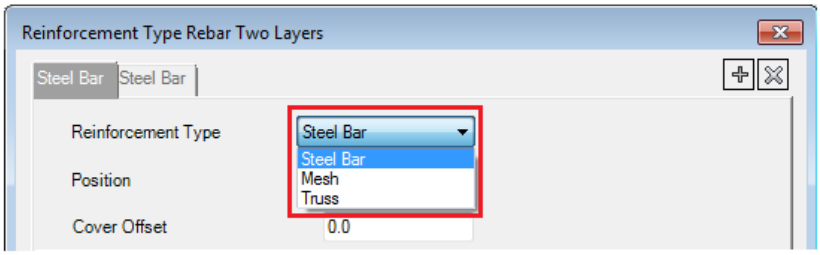
To access the reinforcement type configuration dialog, click on the Edit button located after each reinforcement type name.

Set up the Area Reinforcement



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

- Number of layers: you can add as many layers as you need by clicking the "+" sign and you can also remove them by clicking the "X" in the top right corner of each layer.
- Reinforcement Type: you can choose from the following reinforcement types: Steel Bars and Meshes or Trusses.

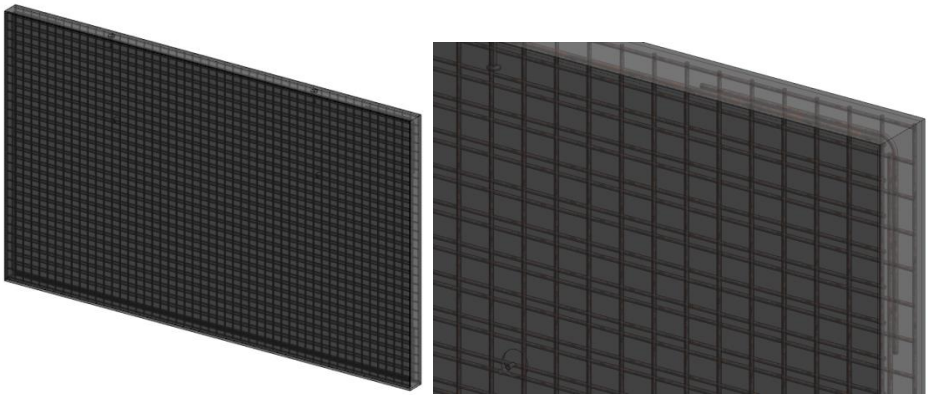


Note: Truss layers are used to assign reinforcement only to Double Walls.

Steel Bar Reinforcement Type*:*

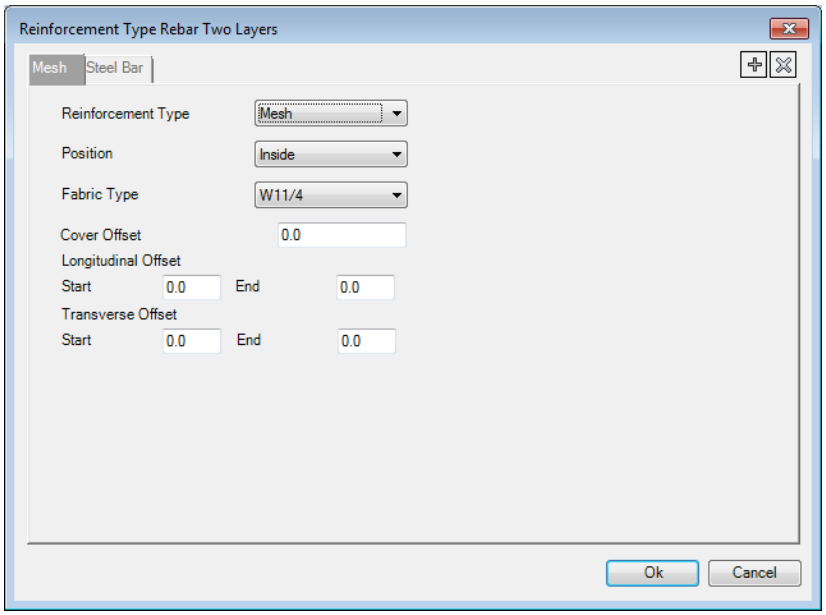
- Position – defines the position of the reinforcement choosing one of the three options from the drop-down list: Outside, Inside or Center.
- Cover offset – defines the additional offset value from the cover. Positive values offset the reinforcement inside the precast element.
- Rebar Area Inside / 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 Outside). You can choose the bar direction for the two available sets of reinforcement (vertical/horizontal).
  - The rebar set can be defined either by the number of bars, the distance between bars or the maximum distance between bars.

- *Distribution and Bar Length Offset*: The offset properties are relative to the wall boundary. The offset overwrites the actual rebar cover and gives you the possibility to extend the reinforcement outside the wall boundary using negative values. The distribution offset affect the length and number of bas in the distribution, whereas the length offset increases or decreases the length of bars in the distribution.



Mesh Reinforcement Type:

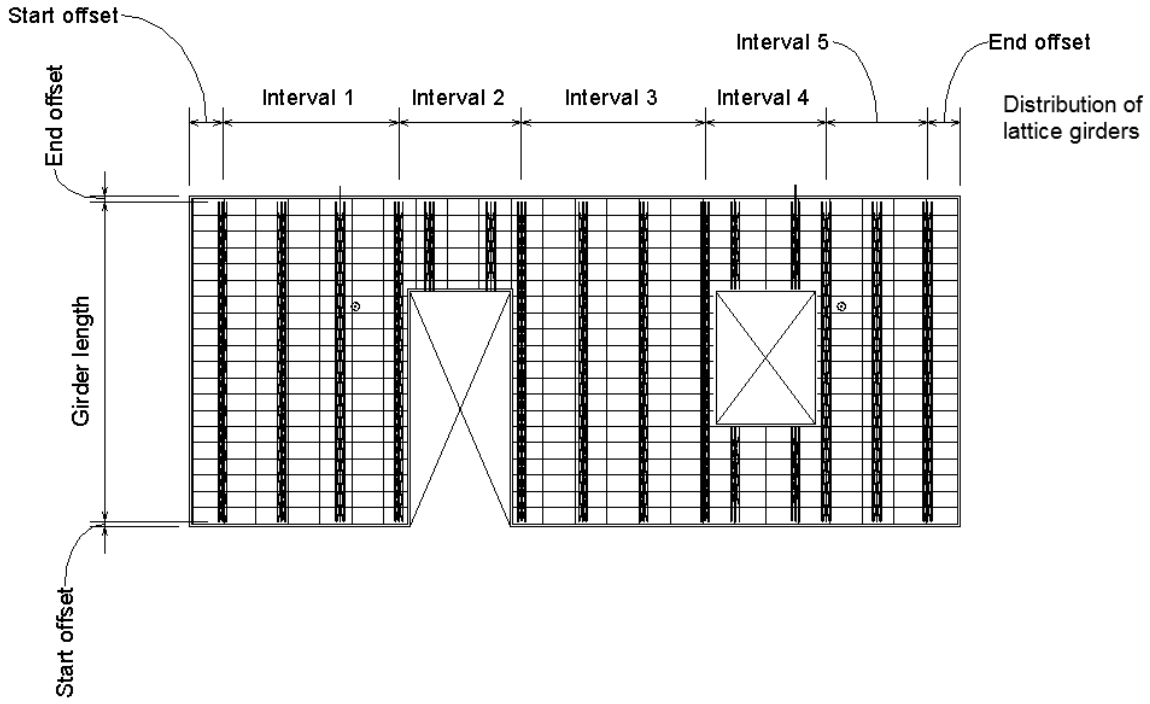
- Position – define the position of the reinforcement choosing one of the three options from the drop-down list: Outside, Inside or Center.



- Fabric Type – you can define a type with one layer (in the center of the wall) or two layers (one on the inside and one on the outside of the wall). Sidewise an additional cover offset can be defined. If the offset is zero, the outside bars are placed at cover.
- Longitudinal and Transverse Offset allow you to customize your fabric area. By specifying negative values you can obtain fabric reinforcement outside the actual wall boundaries.

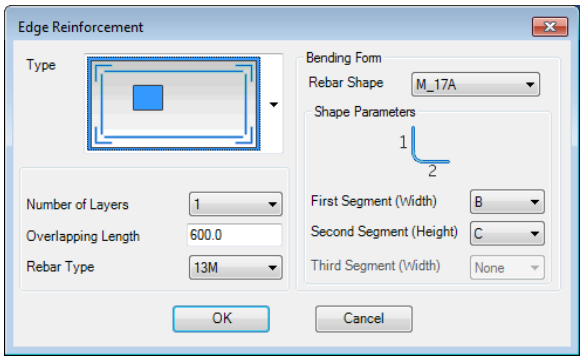
Truss Reinforcement Type*:*

- Position – define the position of the reinforcement choosing one of the three options from the drop-down list: Outside, Inside or Center.
- Truss Type – specify the type of family you want to use for the Lattice Girders. The distribution of Lattice Girders can be defined either by specifying their number, the distance between them or the maximum distance between them. The distribution of lattice girders is defined either between the edge of the wall and an opening or between two consecutive openings. You can also choose the direction for how lattice girders are distributed along the wall (horizontal/vertical):



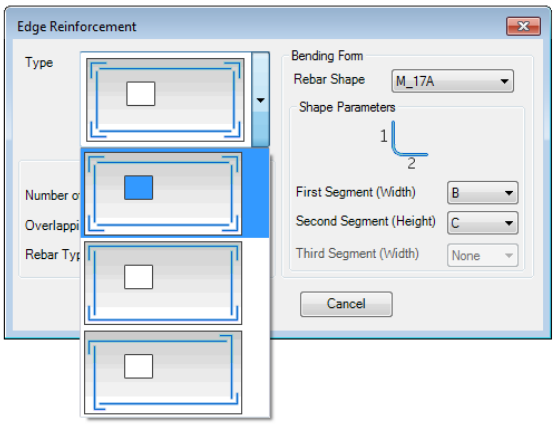
- Offset- this parameter allows you to offset the lattice girders inside the double wall.

Edge Reinforcement Definition



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

- Type: select the desired type from the drop-down list.



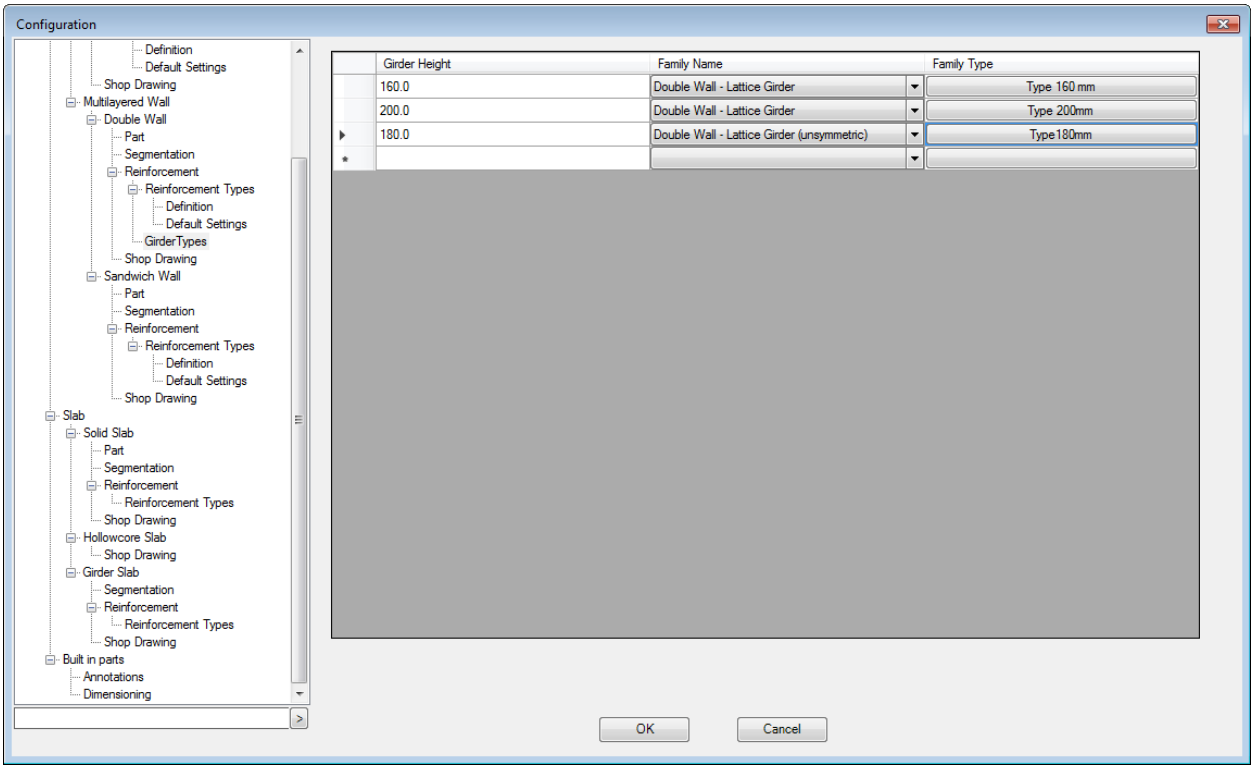
- Number of Layers: you can select one, two or three layers from the drop-down list.
- Overlap: set the overlapping distance between the two segments that join at a corner.
- Rebar Type**:** choose the desired rebar type from the drop-down list.
- Bending Form – Rebar Shape: choose the desired rebar shape from the drop-down list.
- Shape Parameters: choose the width of the first segment and the height of the second segment from the drop-down lists.

Girder Types Tab – Double Walls

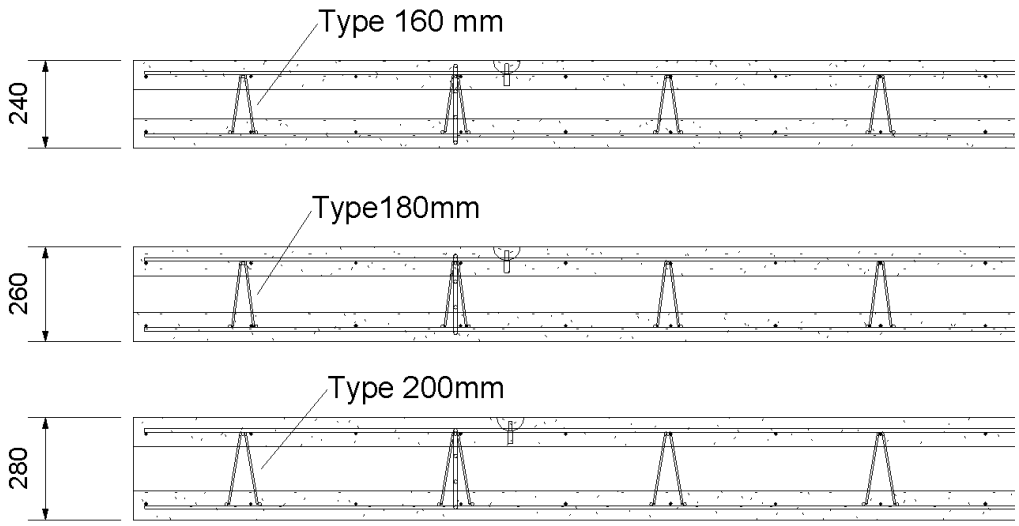
Besides the Lattice Girder family that comes with Project Bran, you can also define your own line-based family that you can use during placing lattice girders.

In the Configuration dialog, under Wall → Double Wall → Reinforcement → Girder Types, you can specify the family name and type for a specific, selected, girder height. The girder height is calculated, by using the following formula:

`Girder height = wall thickness – cover inside – cover outside – diameter horizontal bars inside – diameter horizontal bars outside.`



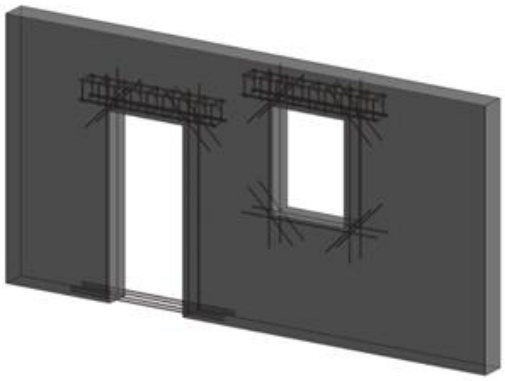
Based on the specified girder height, the software generates girder slabs that fit in the space remaining inside the double wall. Once you insert your first entry, a new line is automatically added in the Girder Types dialog.

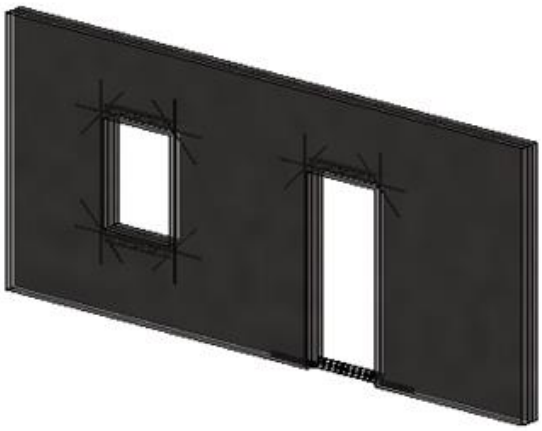


REINFORCEMENT TYPES – DEFINITION FOR OPENINGS

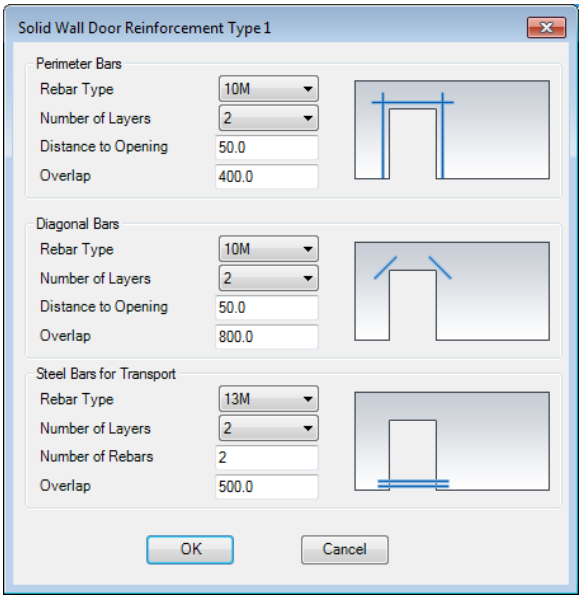
Depending on the configuration, Project Bran automatically inserts reinforcements around openings when they are inserted. It can be reinforcement around doors, windows and openings with optional lintel reinforcement. The reinforcement is only added to opening families (Window, door and wall opening). For manually created families the user has to be sure that the bounding box of the opening is the correct one.

Using the Reinforcement Types for Openings dialog section of the Reinforcement → Reinforcement Types → Definition tab in the configuration dialog, you can define the configuration of the opening reinforcement.





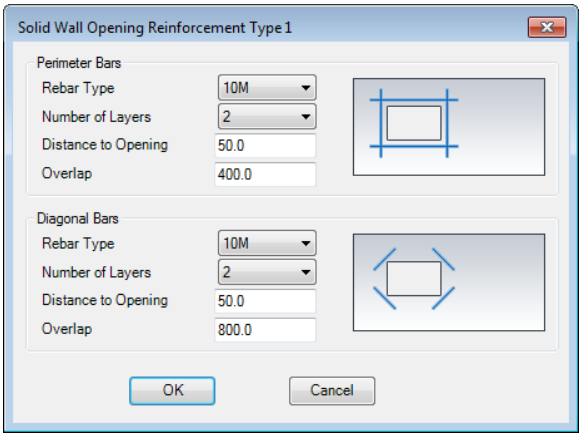
Door reinforcement types



Define the perimeter bars around the door, setting the rebar type, the number of layers (usually 1 – it will be positioned in the center of the wall or 2 – one on the inside and one on the outside of the wall), the distance to the opening and an overlapping.

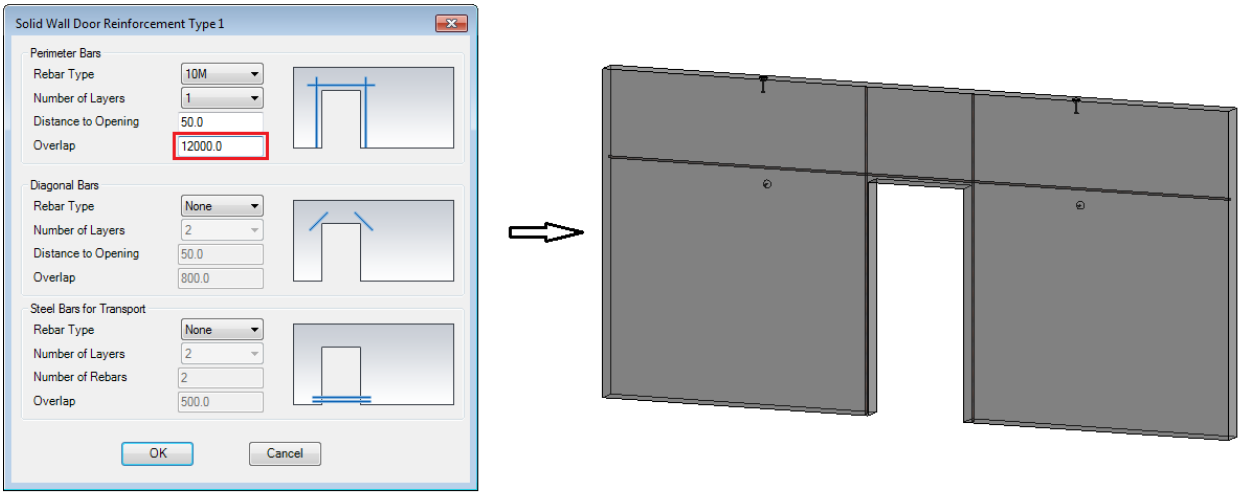
You can also define diagonal bars for the upper corner and bars for transportation at the bottom of the door, using the same process.

Window / opening reinforcement types



Define the perimeter and diagonal bars for windows and other openings.

Note: If the user specifies a very large overlap value, it can generate reinforcement that is spread through the height/length of the wall assembly. This is valid for both doors and windows.

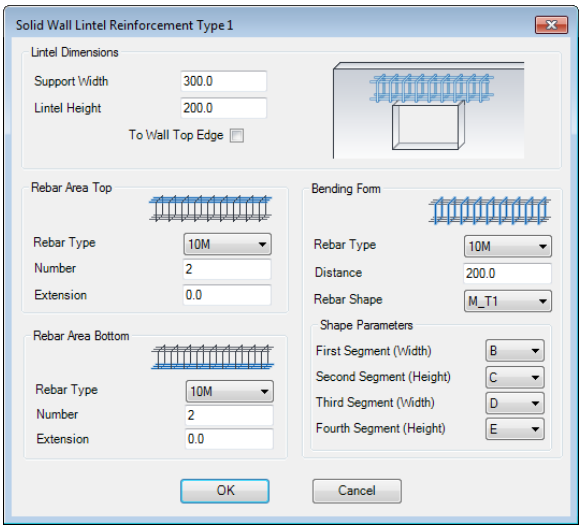


Lintel reinforcement types

For solid wall's lintel you can define the following:

Lintel Dimensions:

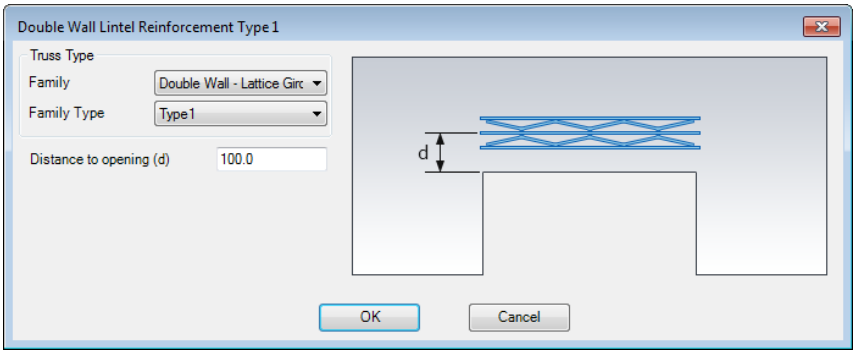
- The Support Width defines the extra length on the left and right side of the opening.
- The Lintel Height can be defined with a fixed value or with a height from the top of the opening to the top of the wall.
- The 'To Wall Top Edge' checkbox, if checked, it generates lintel reinforcements throughout the remaining height between the opening and the top edge of the wall.



Rebar Area Top and Rebar Area Bottom: Define the rebar type, the number of reinforcement bars and an extension on both sides. The *Extension* property gives you the ability to add extra length to the support width.

Bending Form: You can define the rebar shape, the distance between the rebar shapes and their diameter. For the selected shape you must define the defining parameters. Depending on the rebar shape you need to select only the first two for the width and height or select all four.

For double wall's lintel reinforcement you can define type of truss that you want to be used as lintel, and the distance from the center of the truss to the opening.



REINFORCEMENT TYPES – DEFAULT SETTINGS

In this tab you can define the situation when a certain type of reinforcement should be used.

For the wall parts, you can define the area and edge reinforcement for different intervals of wall thickness, as well as choose the desired reinforcement type for each defined situation.

For openings you can define door, window and other opening reinforcements for different intervals of wall thickness, as well as choose the desired reinforcement type for each defined situation.

For the lintel reinforcement you can define different intervals of opening widths and choose the desired reinforcement type for it.

DIVIDING OF WALLS

In order to create solid precast wall panels from walls inside Revit, the wall has to be structural or have at least one layer set to Structural Material (even if only one layer is defined). This layer must have a material defined and the material must have a density assigned. If you try to split an architectural wall, you will receive a warning message saying that the selected wall does not have a structural usage assigned.

In the Configuration dialog - Precast tab → Configuration panel → (Configuration), you can configure the type of mounting parts, segmentation and reinforcement settings which will be taken into account when using the specialized commands. For more details, see Configuration.

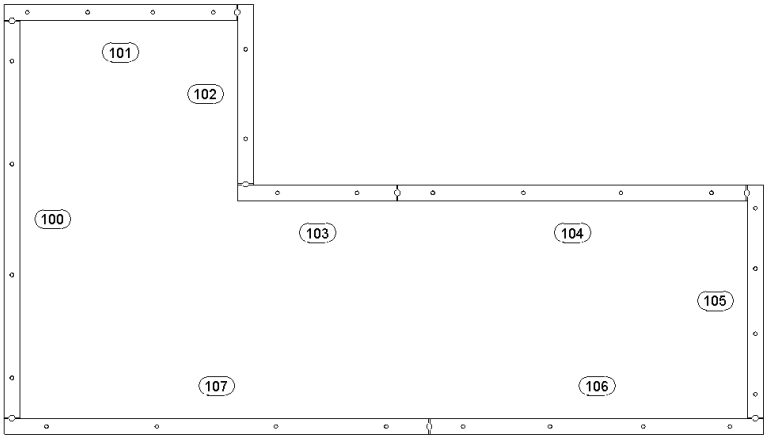
For all walls you can specify the number from which the wall numbering will start for each level. If you need to have different numbers for identical assemblies, you should uncheck the Unique Position Number property for the precast element that you will be using. By default the Unique Position Number box is checked.

AUTOMATIC SEGMENTATION OF WALLS

To divide walls into producible panels, select one or several walls and use the (Split) command, from the Precast tab → Segmentation panel.

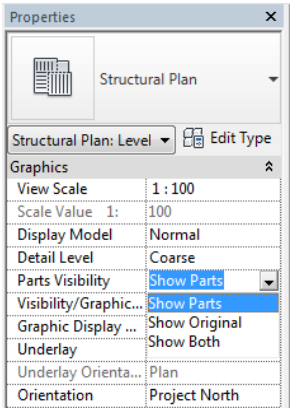
This command will:

- Create parts out of the walls.
- Divide the parts with the Revit divide command.
- Adds wall markers for each assembly (these can be viewed in a plan view)
- Create connections and reinforcements.
- Add them all together to an assembly.



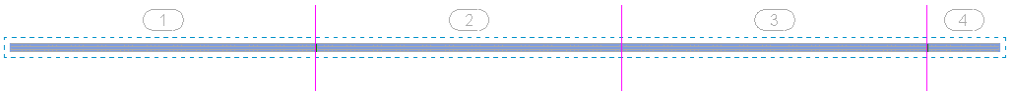
MANUAL SEGMENTATION OF WALLS

After the automatic dividing you can manually modify the dividing. Therefore, you need to select one part of the wall. To select the parts, the part visibility must be on. You can turn on the parts visibility in the properties, by selecting Show Parts or Show Both.



If you select one part of the wall, choose the (Edit Division) command from the Modify Parts tab → Part panel and then (Edit Sketch) from the Modify Division tab.

Now the Revit model shows the dividing lines:

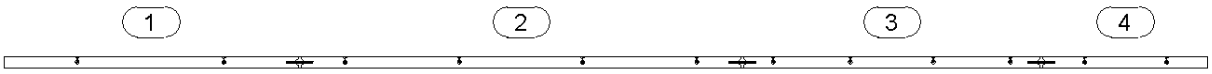


This allows you to move, delete or add dividing lines. After you close the edit mode, the Precast Tools will create new wall panels according to the modified dividing lines.

You can do a manual split from the beginning, by transforming the wall into parts and using the split command on each of them. When doing this, the software checks if the part is within the segmentation parameters, and applies connections and mounting parts.

Division lines should be orthogonal (straight lines) composed of only one segment. When the split command is used on parts with slanted division line, connections (loops and void profiles) will not be created. Only lifters and bushings will be created.

After using the split command, the parts will be segmented according to the rules set in the Configuration dialog. The connections between two adjacent parts are created when using split command. If you use the split command on a part and its neighbors have not been split, this part will not get its connection done because it has no information about the neighbors.



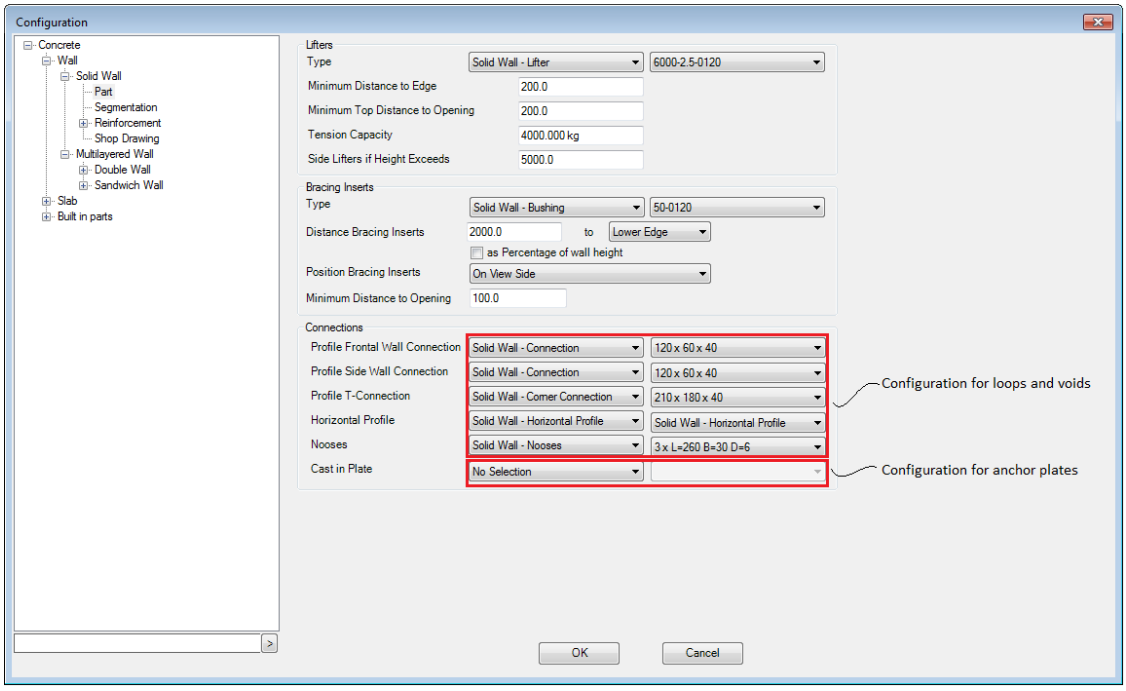
Note: Following wall modifications, the recreation of panels triggers the re-creation of bushings, lifters and in some cases, the connections. If somehow, one or more elements do not respect the segmentation rules after the modification, that specific element will not be segmented according to segmentation rules and will keep the resulted dimensions. If you want to have the division at the division lines, you need to select all the precast assemblies (not the original wall) and re-apply the split command.

CONNECTION BETWEEN PANELS

The wall panels for solid and sandwich walls can be connected using the following options:

- Connection using voids and nooses.
- Connection using anchor plates.
- Connection using pipes and dowels.

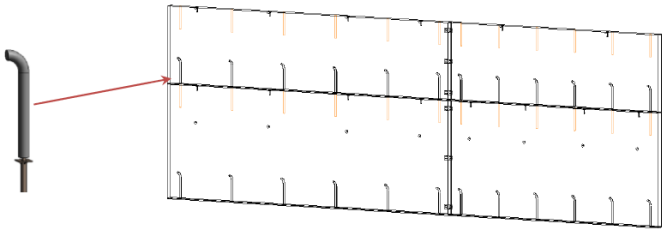
You can set the type of connection in the configuration, using the following dialog:



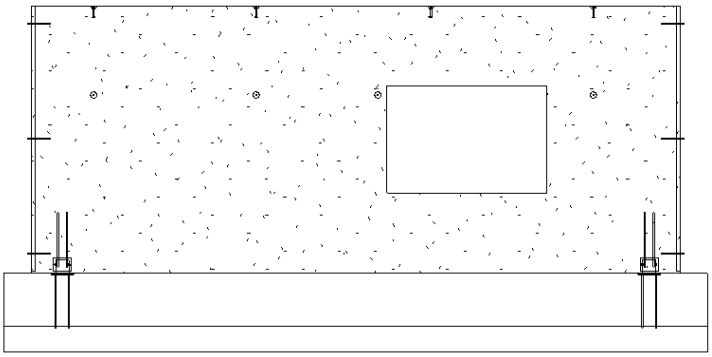
For the connection with loops you can choose a family for the loops and the families for the profiles. To connect the panels with anchor plates you can set the family that the program should use. In this dialog you can also set the profile for the horizontal dividing. All the families are located in the **"**C:\Program Data\Autodesk \Project BRAN 2017\Families\en\ Mounting Parts**"** folder**.** For more details see Connections section.

Vertical connections:

If top and/or bottom mounting parts are needed so that vertical connection between walls or between walls and foundation are in place, the Vertical Connection command can be used. This command can only be applied to precast element obtained only with Project Bran (previously split).

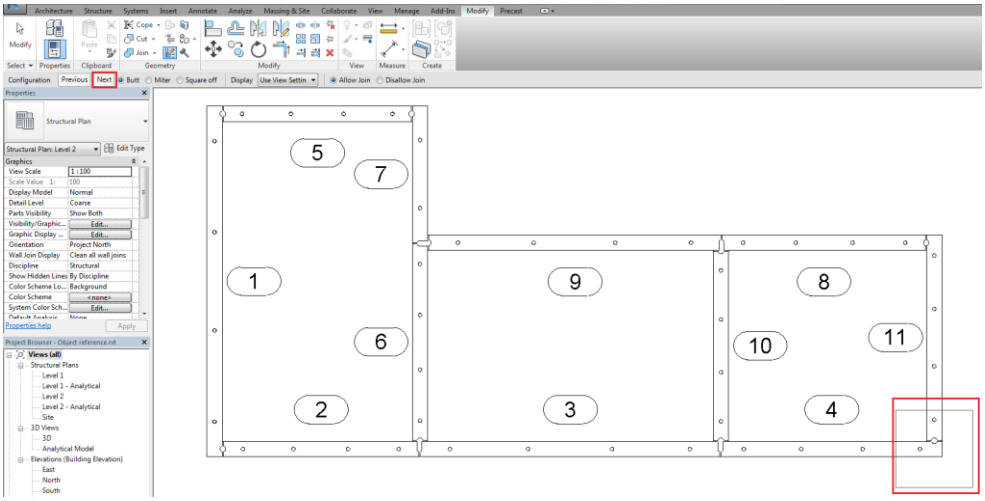


The user can add various families that can be used with the vertical connection command, as long as the families are located in "C:\Program Data\Autodesk \Project BRAN 2017\Families\en\Mounting Parts" and have the name starting with the name of the assembly (e.g. Solid Wall – Example.rfa for solid wall vertical connections).



CHANGE THE CONFIGURATION OF WALL CONNECTIONS

If order to change the configuration of the wall connection you need to go to Modify tab → Geometry panel → (Wall Joins), select the join you want to change and click Next:

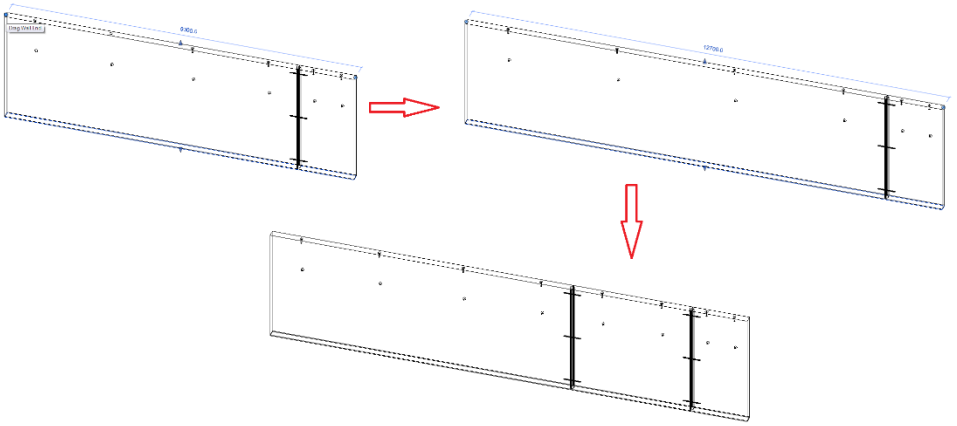


After this process, the connection will not be updated, while the assemblies will be updated. In order to obtain an updated connection, you should delete all the assemblies contained in the wall, that join in the place where the configuration has been changed (i.e. for this example you should delete assemblies 2, 3, 4 and 11), select the original walls and use the split command.

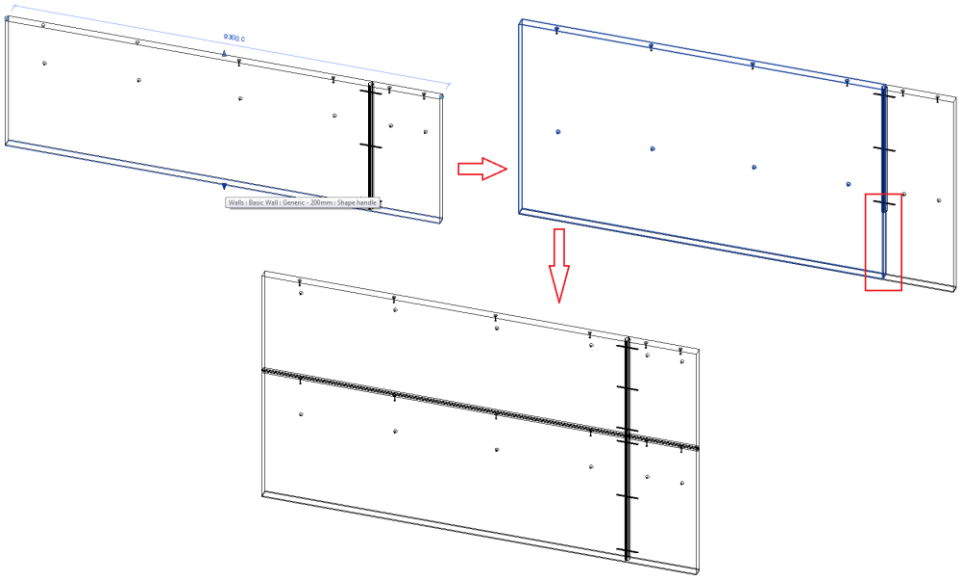
CHANGE THE WALL BOUNDARIES

Changing the wall boundary, either by changing the wall profile or by moving the wall handles or wall end, will trigger the update of mounting parts position (without taking into account the segmentation rules and without updating connections). In order to obtain the right segmentation and connections, you need to select the modified assemblies (not the original wall) and use the split command again. If you select the original wall, it will be re-split without taking into account the previous automatic or manual split.

Modification to wall end:



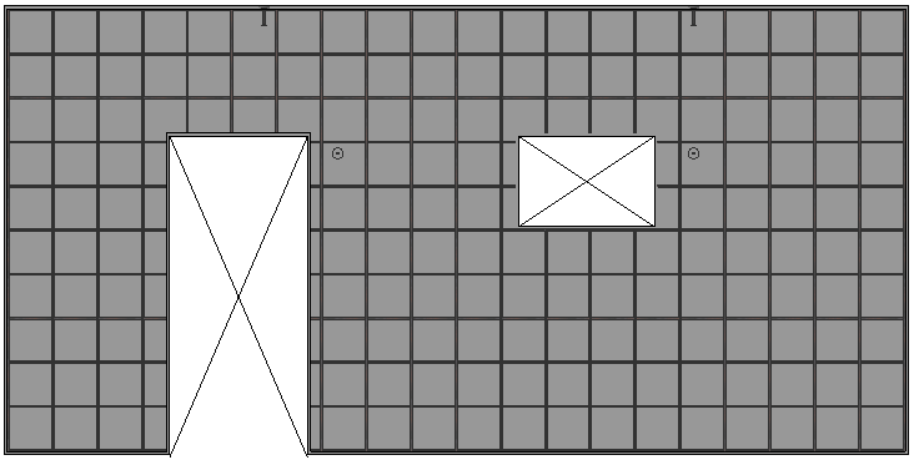
Modification to wall handles:



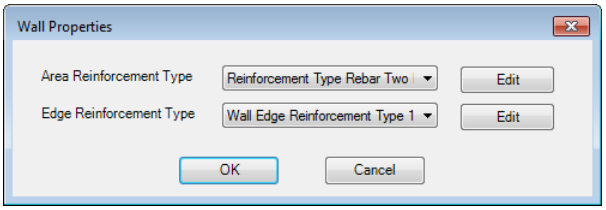
WALL REINFORCEMENT

WALL AREA REINFORCEMENT

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



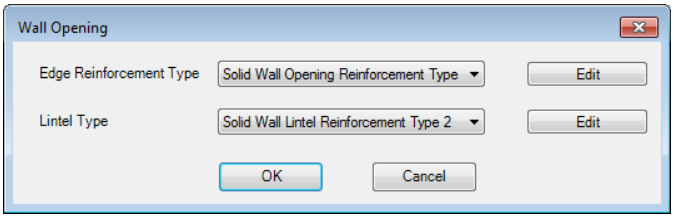
If you select a wall part and start the *Reinforcement* command from the Project Bran Tools, the same dialog appears in which you can customize your reinforcement style according to your needs.



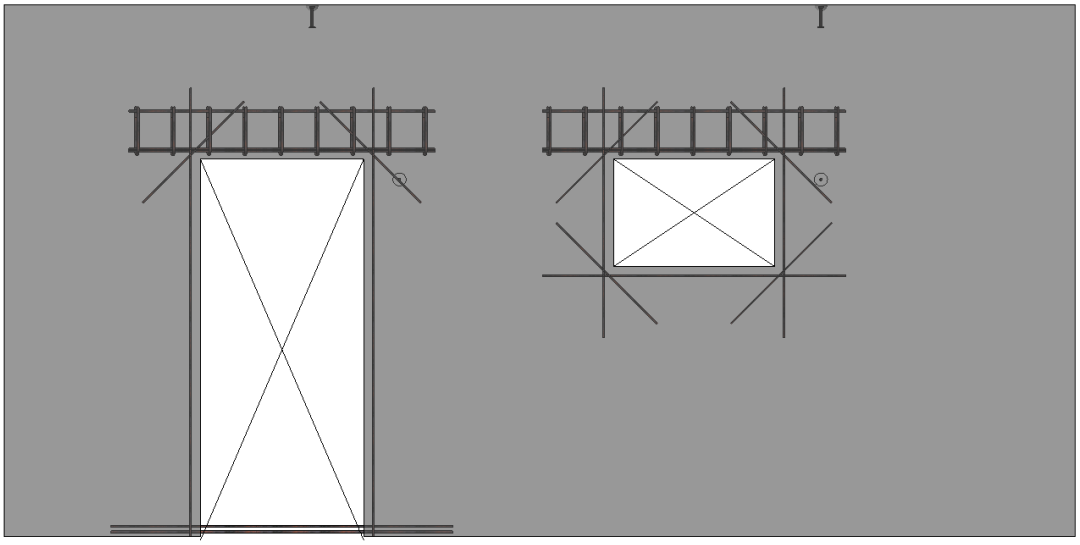
In this dialog you can select the area reinforcement type and the edge reinforcement type for the selected wall. The reinforcement types can be defined in the configuration under the *Definition* of the reinforcement types. For more information visit Reinforcement section.

WALL OPENING REINFORCEMENT

The user has the ability to add opening reinforcement by using the *Opening Reinforcement* command. This is enabled only if the opening is contained in a precast assembly obtained using Project Bran. Select the door opening or window opening and use the Opening Reinforcement command to obtain the dialog which allows you to add opening reinforcement.



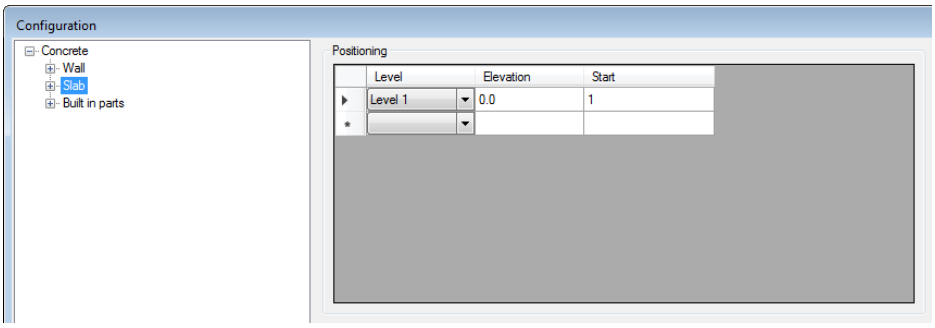
In this dialog you can configure reinforcement type for perimeter, diagonal and lintel bars. The reinforcement types can be defined in the configuration under the *Definition* of the reinforcement types. For more information visit Opening Reinforcement section.



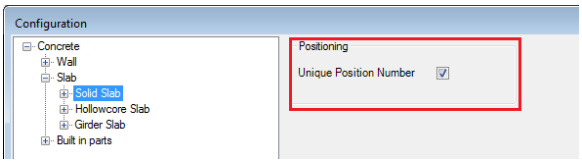
SLABS

SETTING UP PROJECT BRAN

1. **Positioning**: For each floor you can assign a different starting number for the walls.



2. **The Unique Position Number Box:** The first thing you can specify is the unique position number for each type of precast element. This is taken into account when the software assigns position numbers. This option is useful when you want to have different numbers for the same assembly. The unique position number option is available when selecting any specific precast element inside the Configuration dialog:



3. **Slab Creation**: 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.

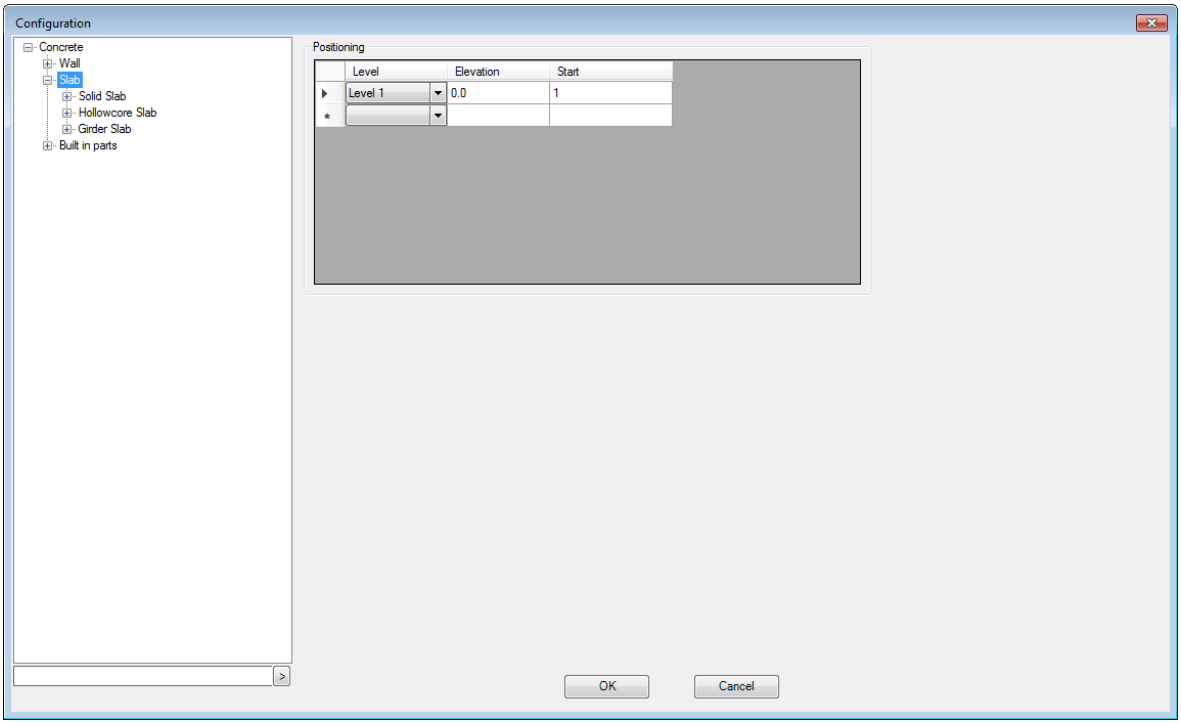
In the *Configuration* (***Project Bran*** tab → ***Configuration*** panel → ***Configuration)*** you can configure Position Numbers and Segmentation parameters.

CONFIGURATION

When using the Configuration command, a dialog opens allowing you to change different settings. For more information see Configuration for walls.

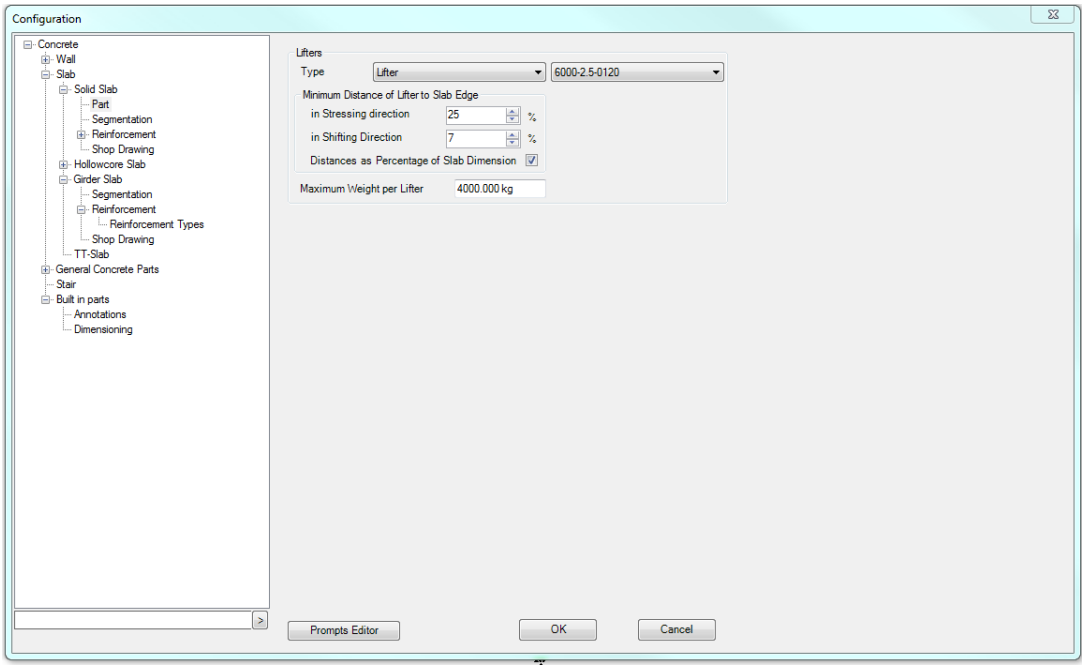
POSITIONING

Inside the Slab branch, you can set the automatic numbering rules for slabs. For each floor you can assign a different starting number for the numbering.



PART

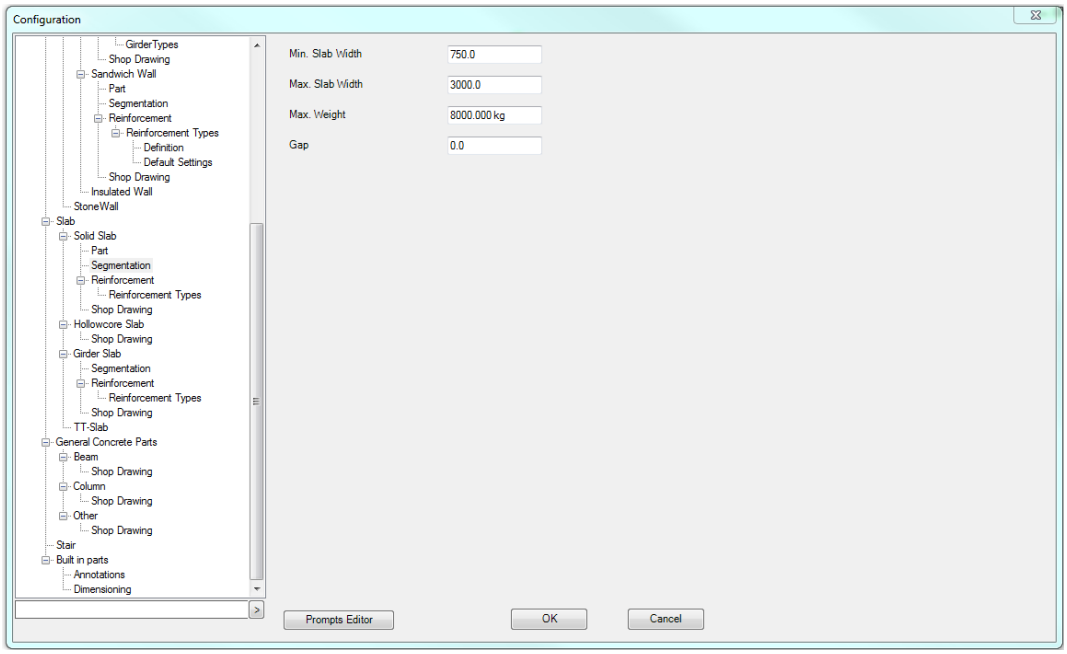
Solid Slabs: In this section the user has the ability to specify the type of lifters, the distance to edge either as length or percentage of slab dimension and the capacity of lifters.



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.

SEGMENTATION

Solid and Girder Slabs: The most important parameters for dividing are the minimal and maximal slab length and height as well as the maximal panel weight. The dividing will always be within the limits of the parameters.



Segmentation parameters:

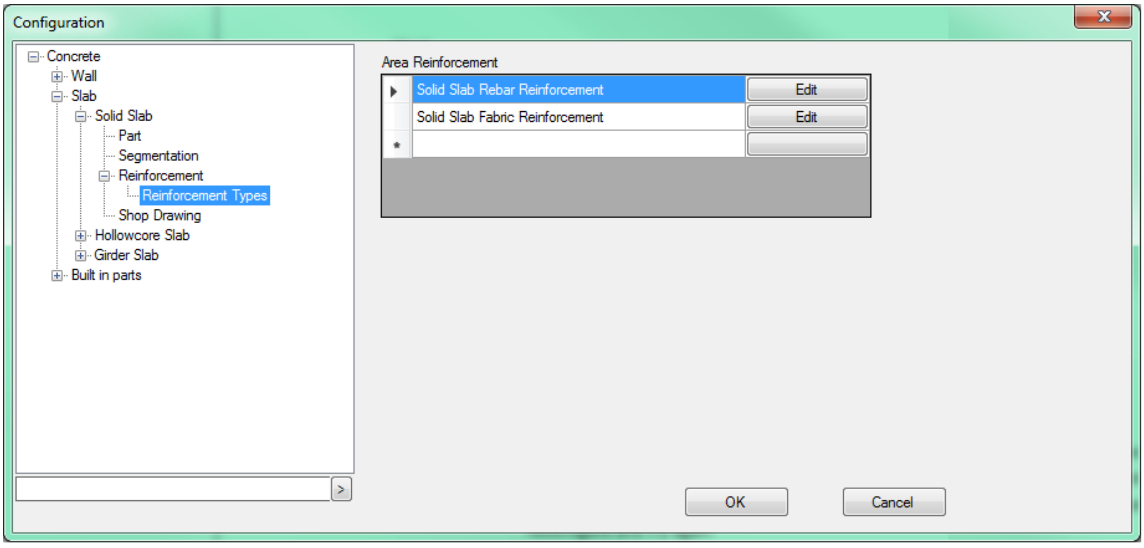
- Min. Slab Width: This parameter sets the minimum width for a solid slab assembly;
- Max. Slab Width: This parameter sets the maximum width for a solid slab assembly;
- Max Weight: This parameter sets the maximum weight for a solid slab assembly; this parameter forces the segmentation at the position 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.

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.

REINFORCEMENT

REINFORCEMENT TYPES - DEFINITION

In order to add new reinforcement style go to ***Project Bran*** tab → ***Configuration*** panel →***Configuration*** → ***Slab*** → ***Solid Slab (Girder Slab)*** → ***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.



Reinforcement command can be applied on precast assemblies obtained either from manual or automatic segmentation.

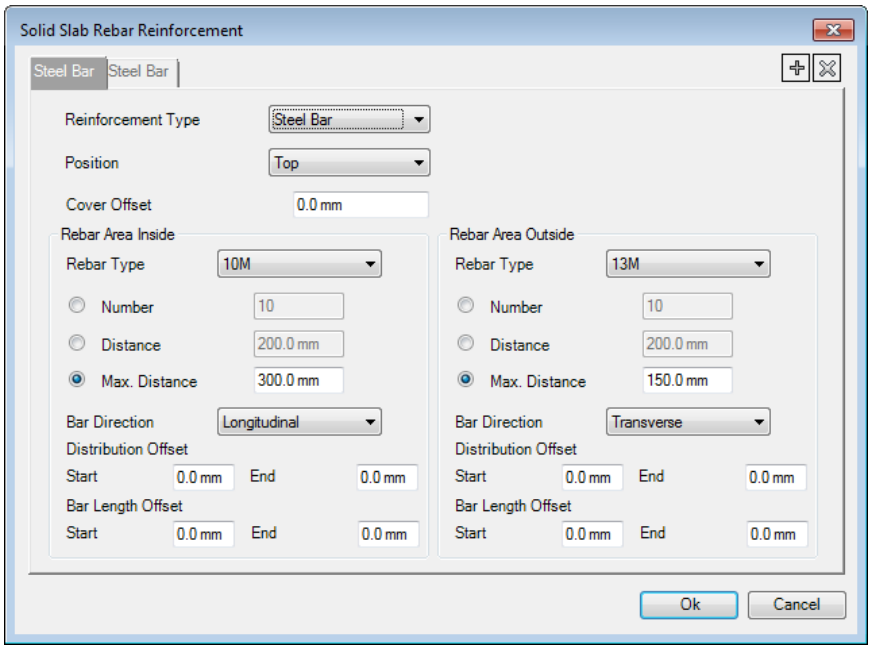
Set up the Area Reinforcement

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

Number of layers: you can add maximum six tabs 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, Meshes or Truss (this is available only for Girder Slabs).

Steel Bar Reinforcement Type*:*



- Position – define the position of the reinforcement choosing one of the two options from the drop-down list: Top or Bottom.
  - Top - steel bars will be placed on the top face of the slab (taking into account element's cover)
  - Bottom - steel bars will be placed on the top face of the slab (taking into account element's cover)
- Cover Offset – With this option the user can specify additional cover offset for the rebar area.
- Rebar Area Inside / Outside

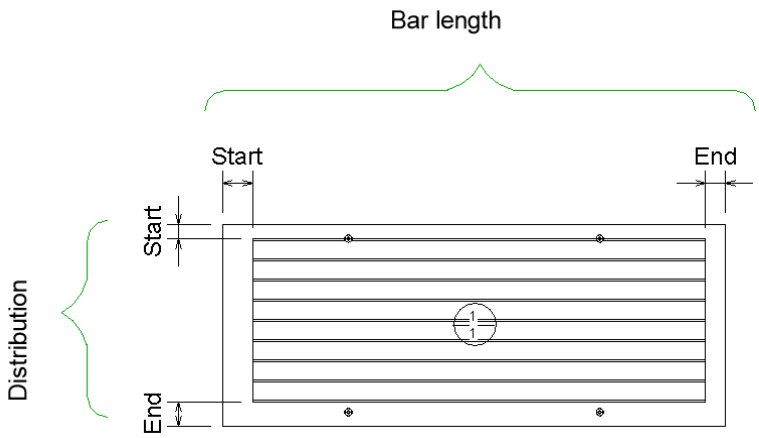
  In the current layer you can define the rebar relative position. You can specify which reinforcement goes first (is bound to the cover – Rebar Area Inside). You can choose the bar direction for the two available sets of reinforcement (longitudinal / transverse).

  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
  - *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.

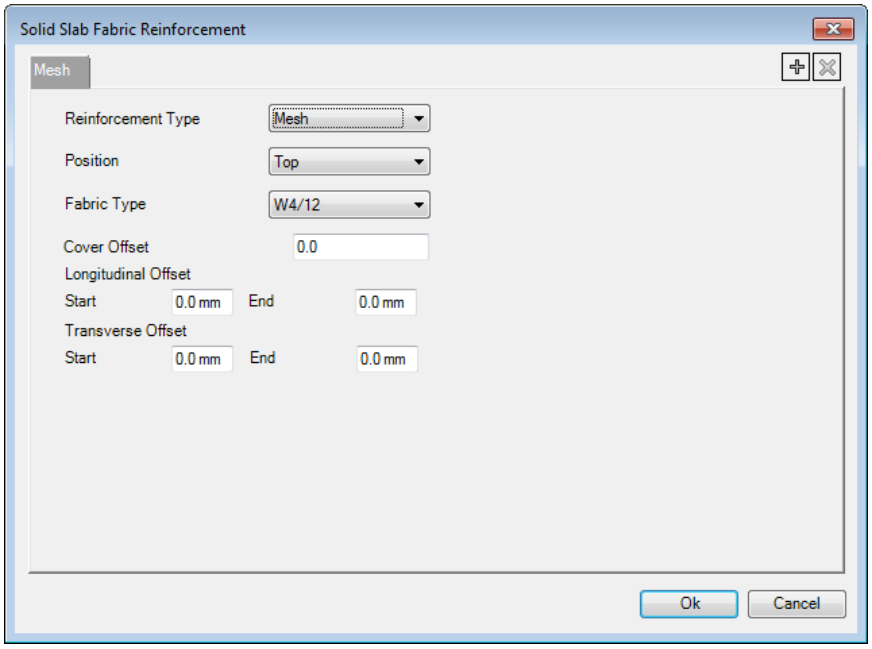
The offset properties are relative to the slab boundary. The offset overwrites the actual rebar cover and gives you the possibility to extend/trim the reinforcement.

Slab start/end offset parameters:

If the slab has been drawn starting from top left to bottom right, the start/end parameters are defines as described below for longitudinal direction of the bars.



Mesh Reinforcement Type*:*

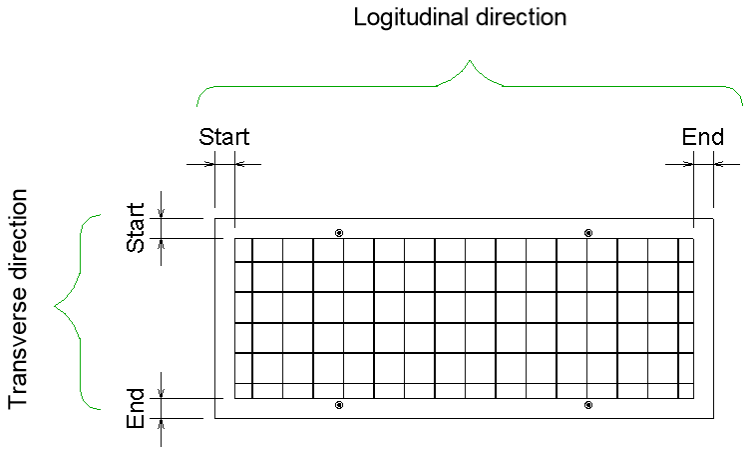


- Position – define the position of the mesh choosing one of the two options from the drop-down list: Top or Bottom.
    - Top – the mesh will be placed on the top face of the slab (taking into account element's cover)
    - Bottom – the mesh will be placed on the top face of the slab (taking into account element's cover)
- Fabric Type – you can define a type of mesh to be used in a certain reinforcement layer.
- Cover Offset – With this option the user can specify additional cover offset for the rebar area. If the offset is zero, the outside bars of the mesh are placed at cover.
- *Longitudinal Offset*: This parameter allows the user to adjust the offset in the longitudinal direction of the solid slab assembly.

- *Transverse Offset*: This parameter allows the user to adjust the offset in the transverse direction of the solid slab assembly.

Solid slab start/end offset parameters:

If the slab is drawn starting from top left to bottom right, the start/end parameters are defines as described below for longitudinal direction of the bars.



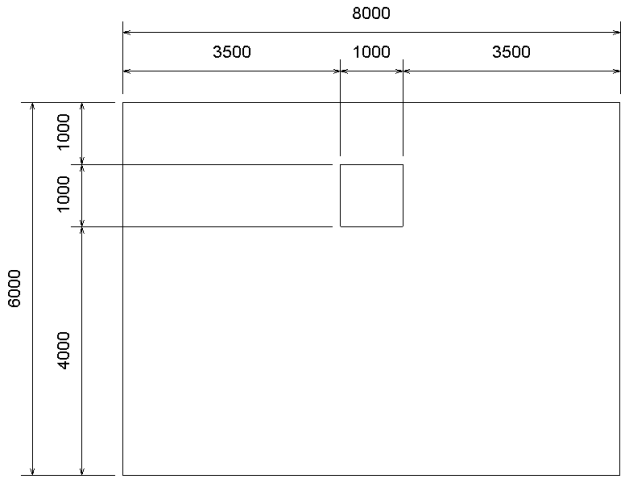
REINFORCEMENT TYPES – OPENING REINFORCEMENT WITH REVIT EXTENSIONS

Project Bran does not currently have the ability to add reinforcement around slab openings. In order to achieve this, you should make sure that the openings for slabs are created using the *By Face* command or the *Vertical* command.

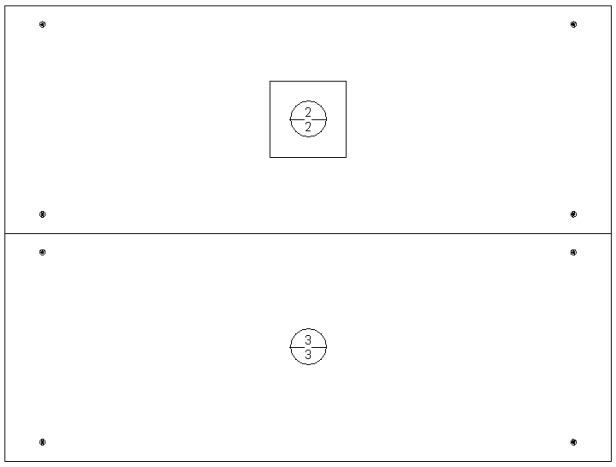
Workflow example:

In this example the workflow to add slab opening reinforcement to an assembly is described.

Step 1. Create the floor with relevant opening in it.

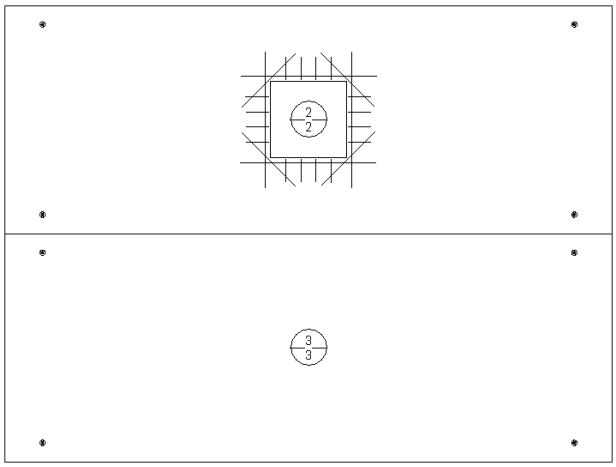


Step 2. Follow the procedure from workflow example from Dividing of Slabs in order to obtain precast assemblies.



Step 3. Reinforce slab openings using Revit Extensions.

a. Select the slab opening to be reinforced.
b. Go to Extension tab → Autodesk Revit Extensions panel → Reinforcement command→ Slab Openings sub-command.
c. Customize your reinforcement to fir your needs.



Step 4. Use the Mounting Parts command to add the resulted reinforcement in the relevant assembly.

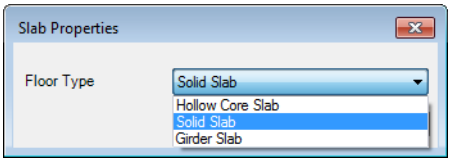
Step 5. Reinforce the slab assembly using Project Bran's *Reinforcement* command. The main difference between the area reinforcement and opening reinforcement is that the area reinforcement is hosted in the part, whereas the opening reinforcement is hosted in the floor. All reinforcement is inside the assembly and can be viewed in shop drawings or in the CNC file. For more information see Shop Drawings section and CNC Export section.

DIVIDING OF SLABS

In the Configuration dialog - Precast tab → Configuration panel → (Configuration), you can configure the type of mounting parts, segmentation and reinforcement settings which will be taken into account when using the specialized commands. For more details, see Configuration for slabs.

AUTOMATIC SEGMENTATION OF SLABS

When using the *Split* command on slabs, Project BRAN divides them into producible panels taking into account the segmentation parameters from Configuration. You can choose the slab type (Hollow Core Slab, Solid Slab, Girder Slab) from the *Slab Properties* dialog:

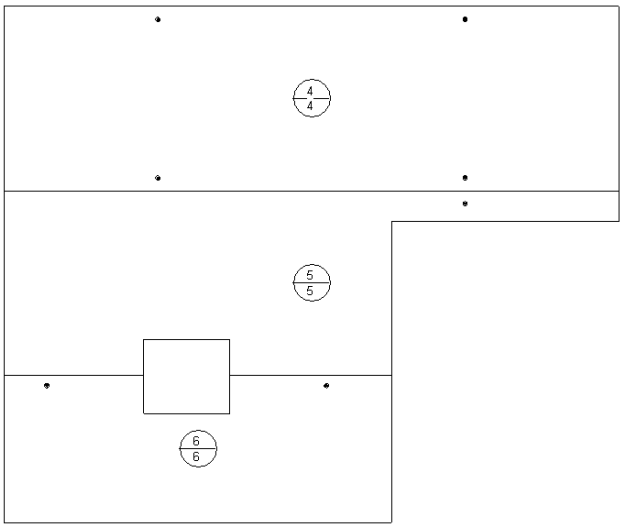


This command will:

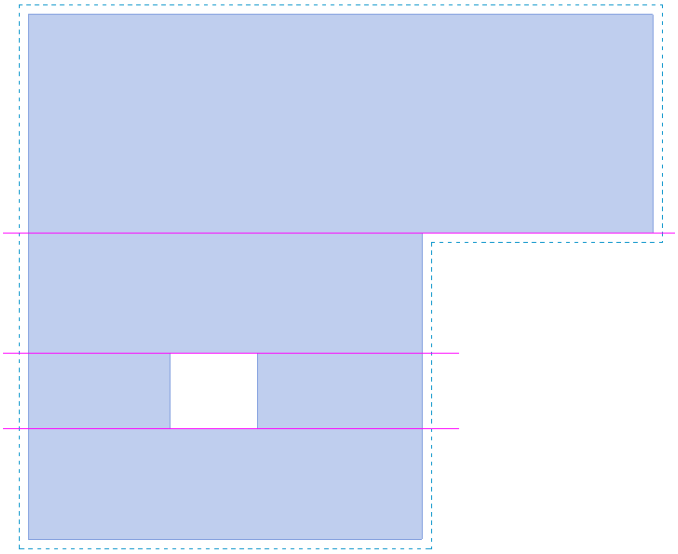
- Create parts out of the slabs.
- Divide the parts with the Revit divide command.
- Adds slab markers for each assembly (these can be viewed in a plan view)
- Create connections and reinforcements.
- Add them all together to an assembly.

MANUAL SEGMENTATION OF SLABS

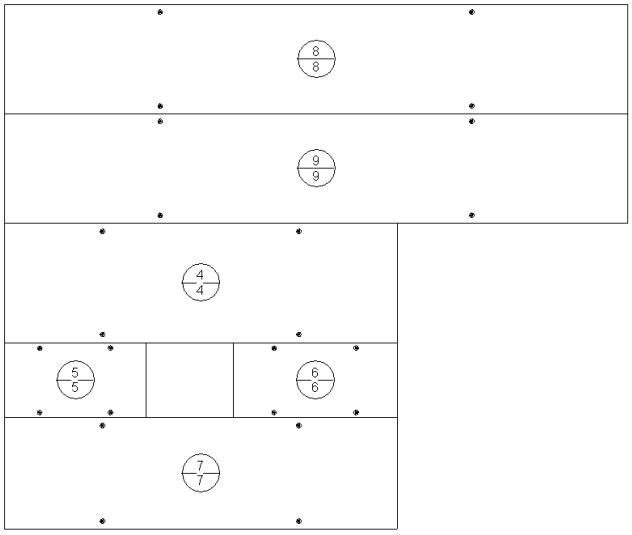
There might be some situations when using the segmentation parameters would generate assemblies with non-rectangular shapes (see picture below).



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. To obtain a floor with different segmentation or element rules in some sections, you can define these sections before starting to create the slab elements. To do this, you must first convert the floor into Parts by using ***Modify*** tab → ***Create*** panel → ***Create Parts***. The part of the structural layer can be divided to create the necessary floor sections, using the *Divide Parts* command*:* ***Create | Parts*** tab → ***Part*** panel → ***Divide Parts***.

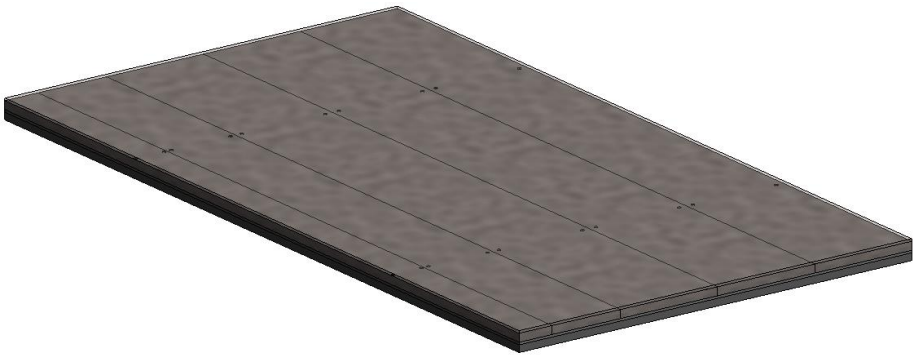


After the floor has been divided into relevant parts, the user can select and split them into producible assemblies. Once the Split command has been performed on one part, the user is asked which precast slab he would like to have and the original floor inherits the proprieties selected for the first part (e.g. Solid Slab, Girder 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.



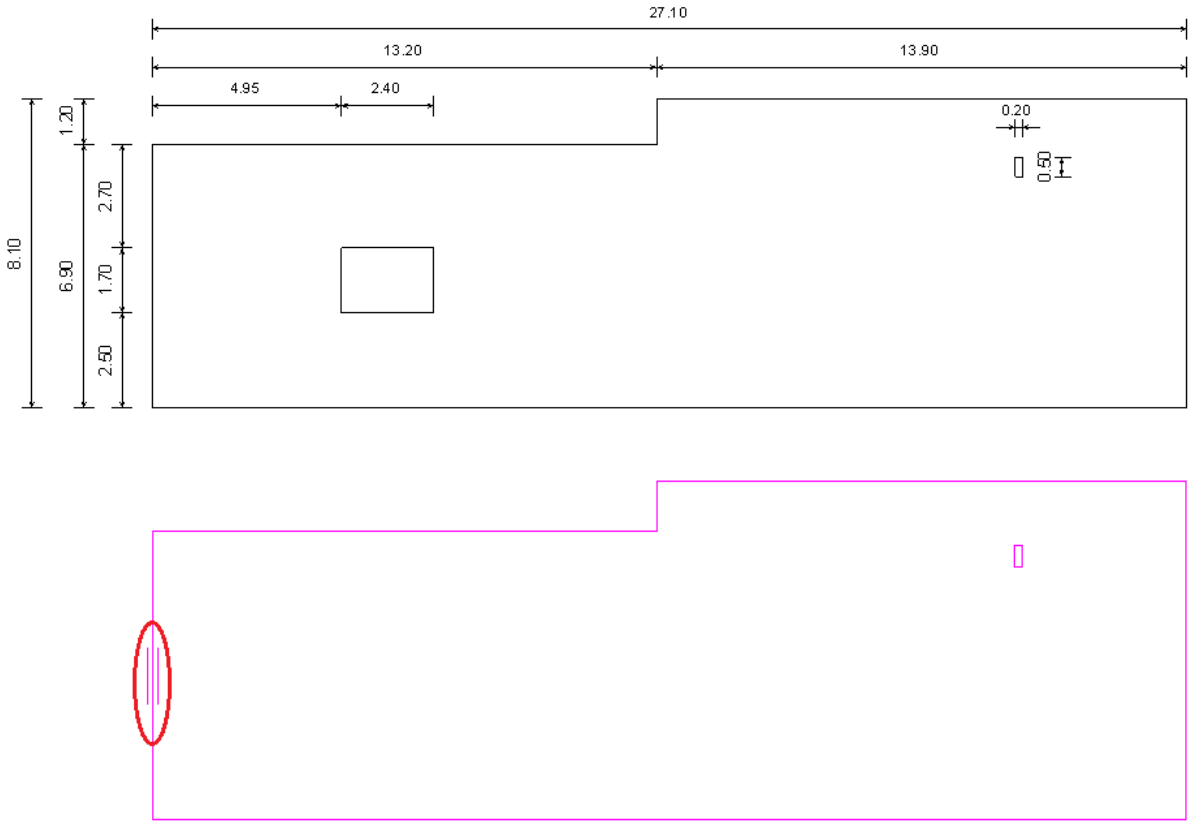
Multi-layered slabs

If somehow the user wants to also have in the same floor the poured concrete, finishes and other layers, he can still have precast assemblies if and only if the floor has been previously been transformed into parts. Once the floor is transformed into parts, the user can select the part that is the structural layer, and it will be segmented into producible assemblies.



Workflow example:

In this example the floor is a structural one, and has the following layout:



The span direction goes from bottom to top and the shifting direction from left to right. No L-shaped elements shall be created and the large opening uncut elements should be created.

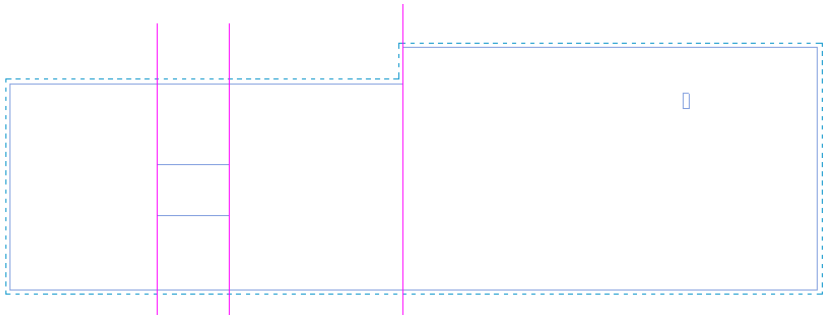
Step 1. Select the floor and use the *Create Parts* command: ***Modify*** tab → ***Create*** panel → ***Create Parts***. This will create a part for each layer of the floor.

Note: If you create a floor with more than one layer, you must hide any of the nonstructural layers from the view in order to make the structural layer visible.

In order to do this, select the visible layer and use *Temporary Hide/Isolate* → *Hide Element*.

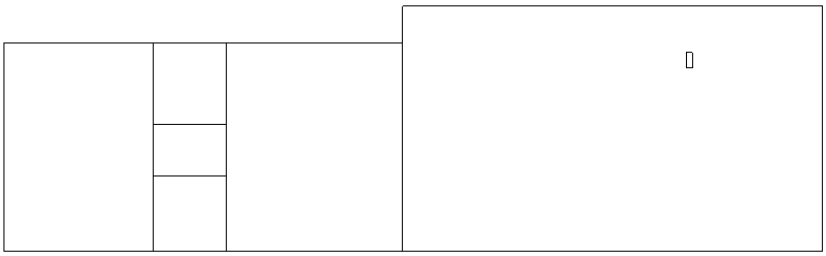
Step 2. In order to create parts out of the floor, the following steps have to performed:

1. Select the structural Part
2. Go to ***Modify | Parts*** tab → ***Part*** panel → ***Divide Parts***.
3. Use *Edit Sketch* from the ***Modify | Division*** tab
4. Create division lines at the section borders – these must be long enough to cut the dashed line around the floor layout.

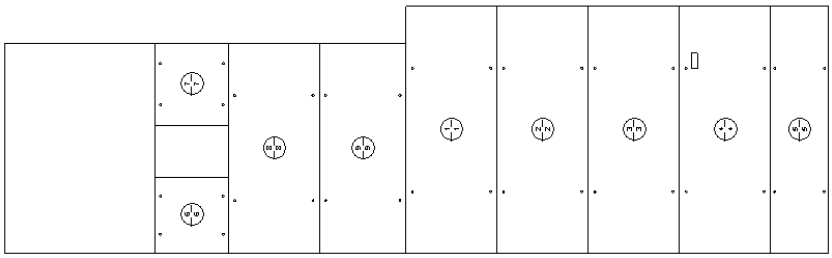


5. Click the green checkmark to close edit mode: *Modify | Division* → *Edit Sketch* → *Mode* → *Finish Edit Mode*.
6. Click the green checkmark again to finish the division mode: *Modify* | *Division* → *Mode* → Finish *Edit Mode*.

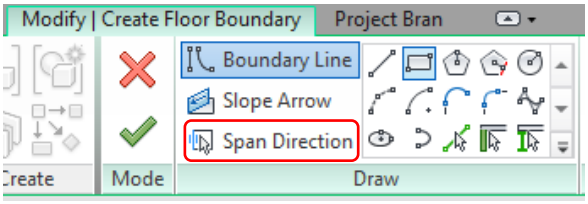
The result should look like this:

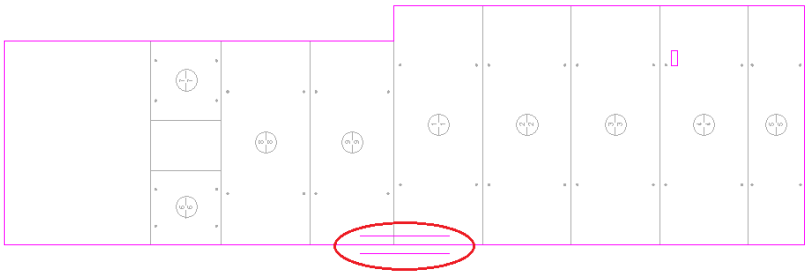


Step 5: Select any part and use the *Split* command: ***Project Bran*** tab → ***Segmentation*** panel → ***Split***. The Floor Properties dialog appears and you have to select the type of precast slab that you want to be generated. When selecting and splitting the remaining parts, these will automatically be segmented with the segmentation parameters and having the same type as the first part.

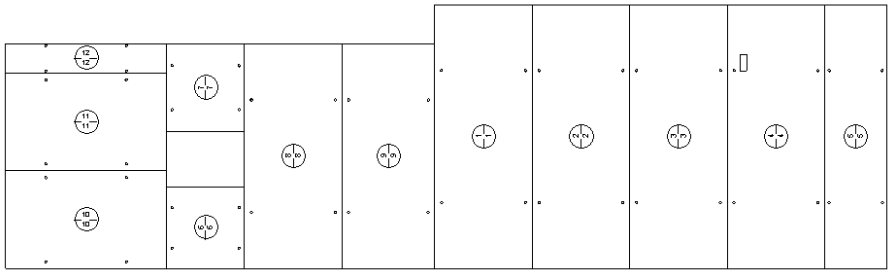


If for some reasons some parts need to have the precast assemblies aligned with a different direction than the one defined initially, you should select the original element and change the Span Direction according to your needs by editing the floor boundary.





After changing the span direction you can go and split the part. The final result should look like the in the image below.



SHOP DRAWINGS

GENERAL

To create shop drawings select the elements for which you want to create shop drawings and use the *Shop Drawings* command: ***Project Bran*** tab → ***Fabrication*** panel → ***Shop Drawings***.

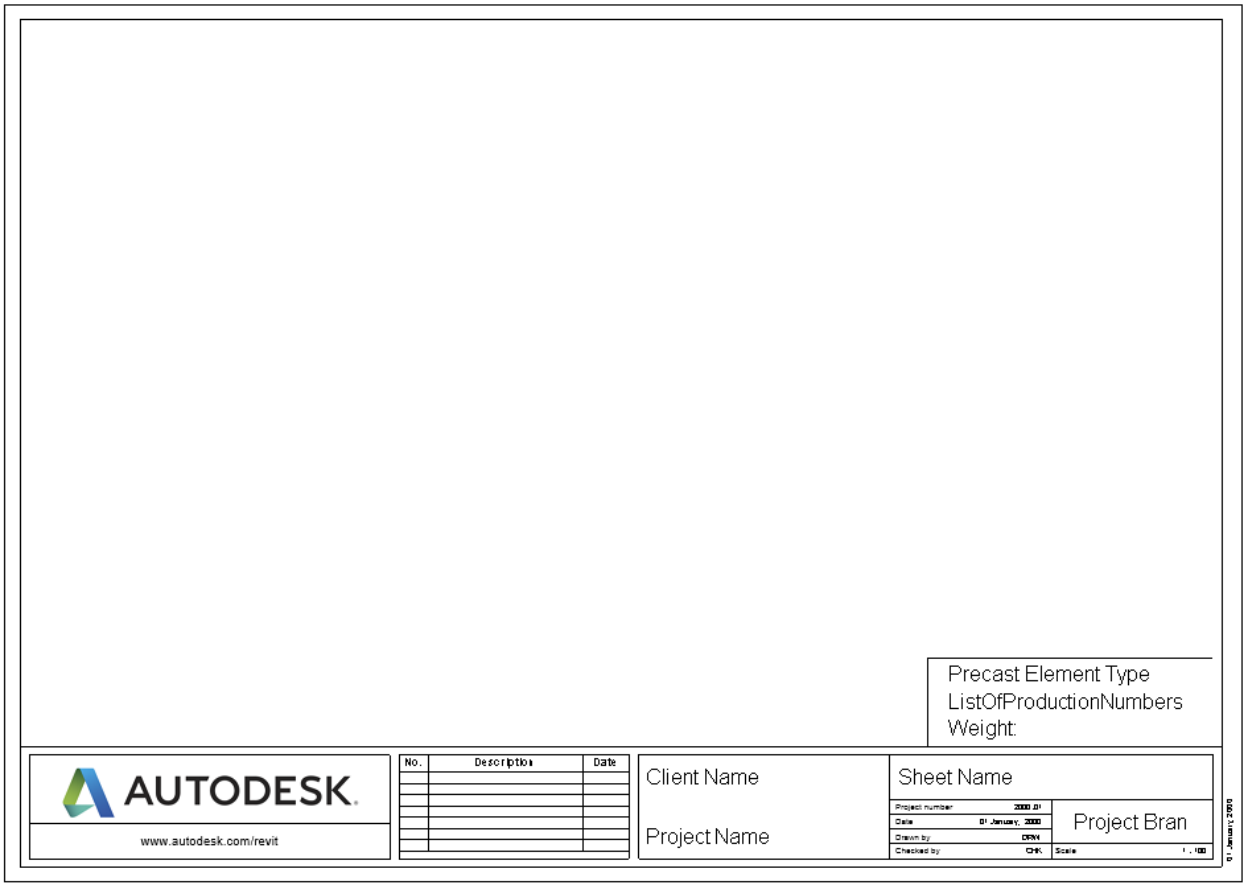
For each wall or slab assembly you can automatically create a shop drawing. Therefore, you need to define and set a title block in the project data.

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.

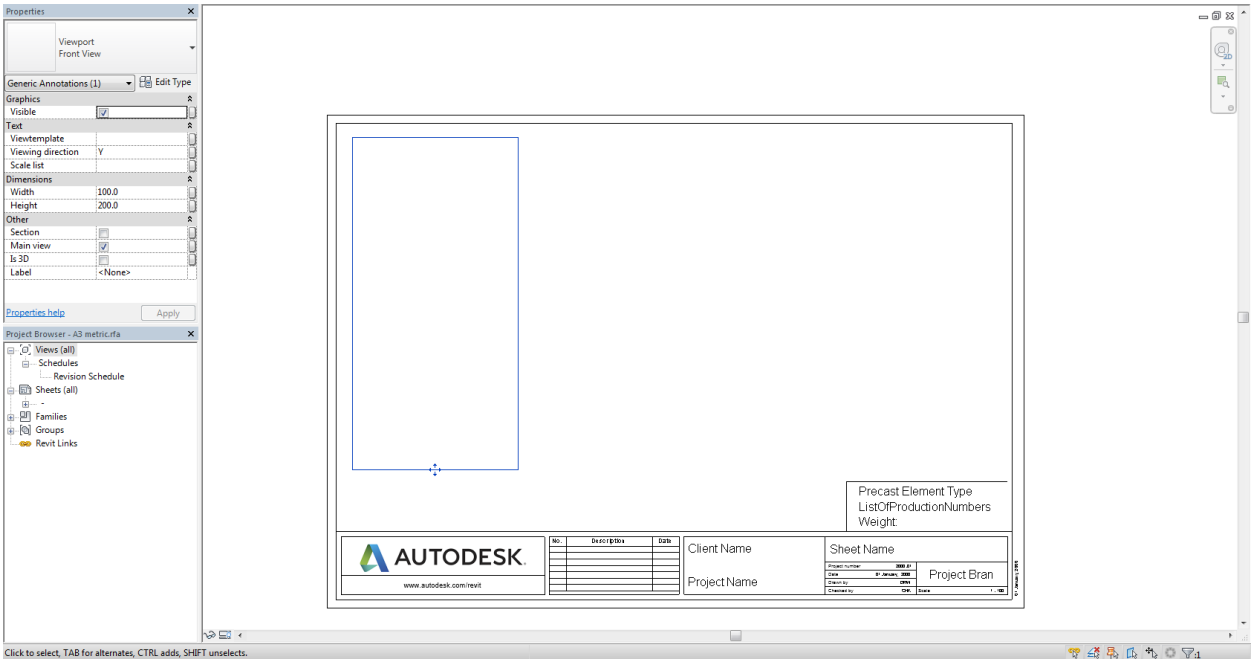
3. The following Shared Parameters with additional element information are available for labels on the title block:
    - PosNoSheet – Position number of the element
    - ListOfProductionNumbers – List of the production numbers for the element on the sheet
    - NumberElementSheet – The number of elements that must be produced
    - ThicknessSheet – Thickness of the element
    - WeightSheet – Weight of the element
    - VolumeSheet – Concrete volume of the element

    To insert the labels:

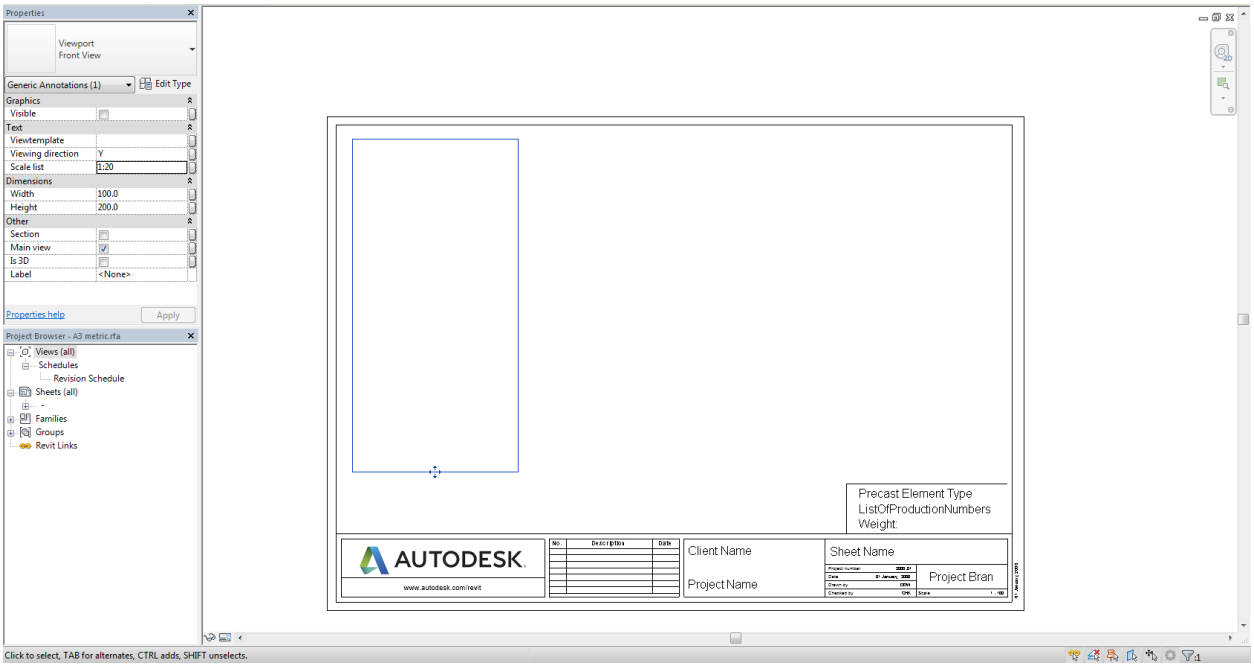
    - Create and place a label on the title block: *Create* → *Text* → *Label*.
    - In the *Edit Label* dialog click the *Add Parameters* button.
    - In the *Parameter Properties* dialog click the *Select* button and select the parameter you need. Close the dialog.
    - Add the selected parameter to 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 Project Bran Program Data Folder, "Title Blocks" folder (e.g. "C:\ProgramData\Autodesk\Project BRAN 2017\Families \en").

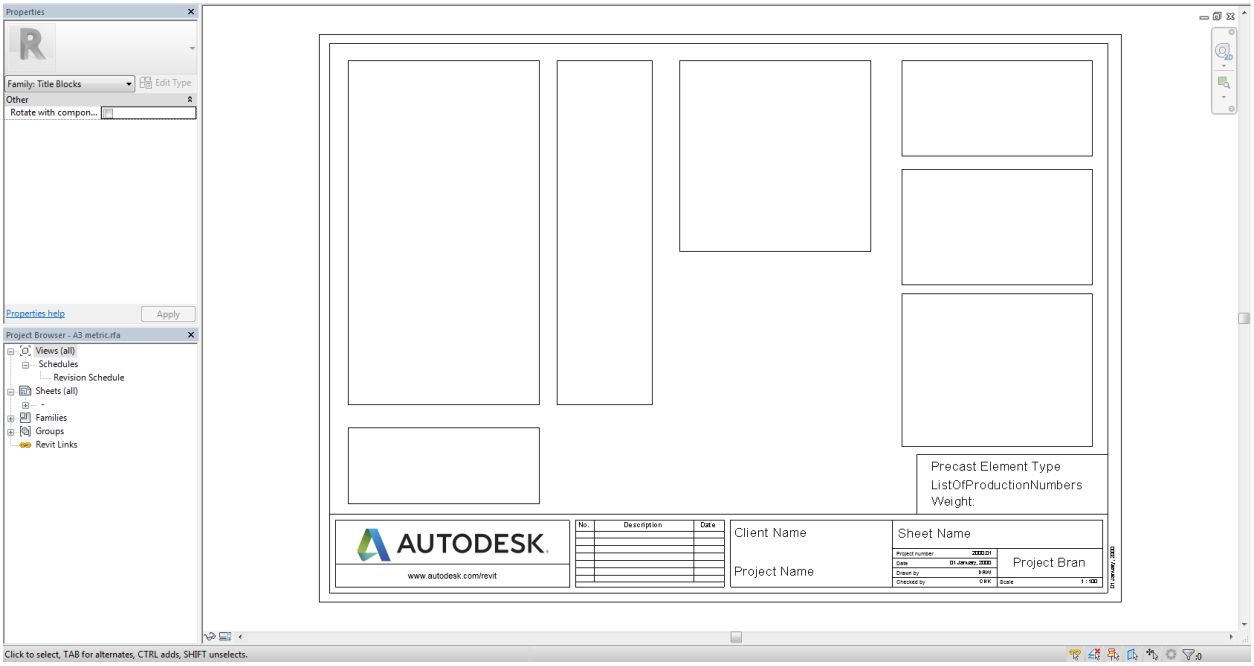
5. Insert Viewpoints: *Create* → *Detail* → *Symbol*; before placing the view port, select the view port type from the properties. You can use the following view ports: Top View, Side View, Front View, 3D View, Material Takeoff, and Part List.



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.



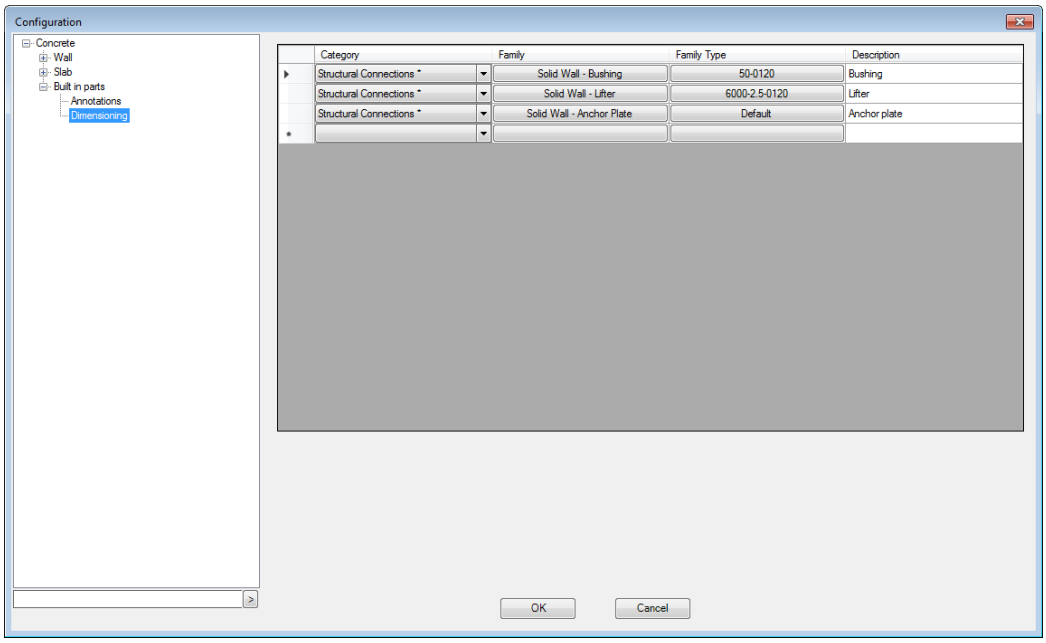
7. Repeat for all views that should be placed on the shop drawings:



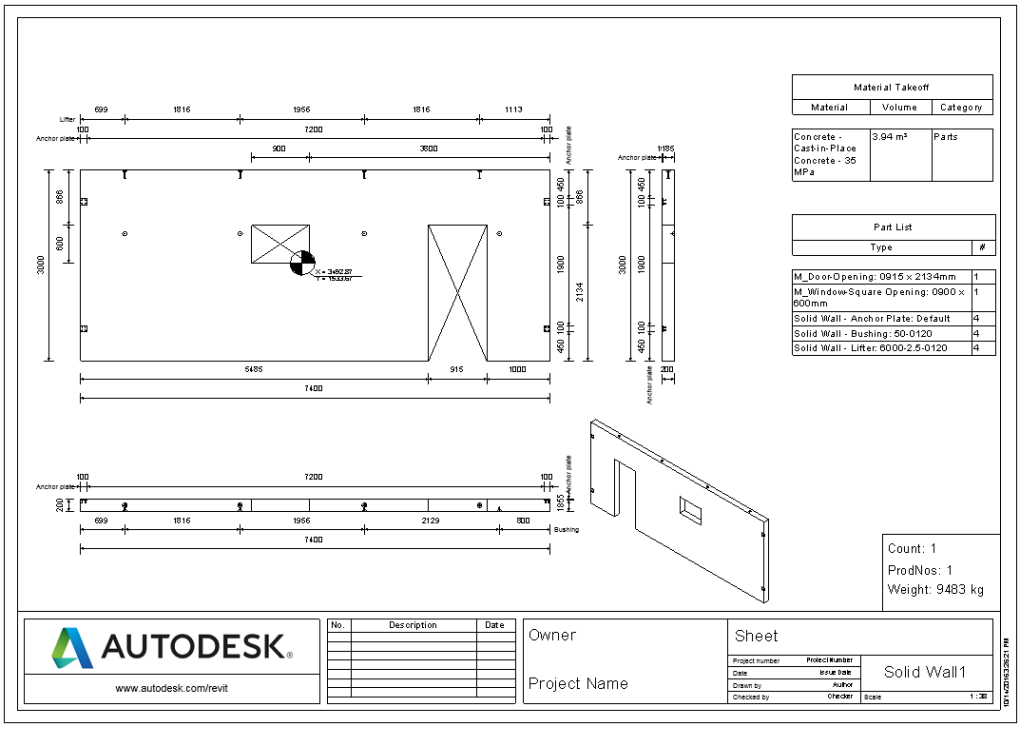
8. Save the title block.

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.

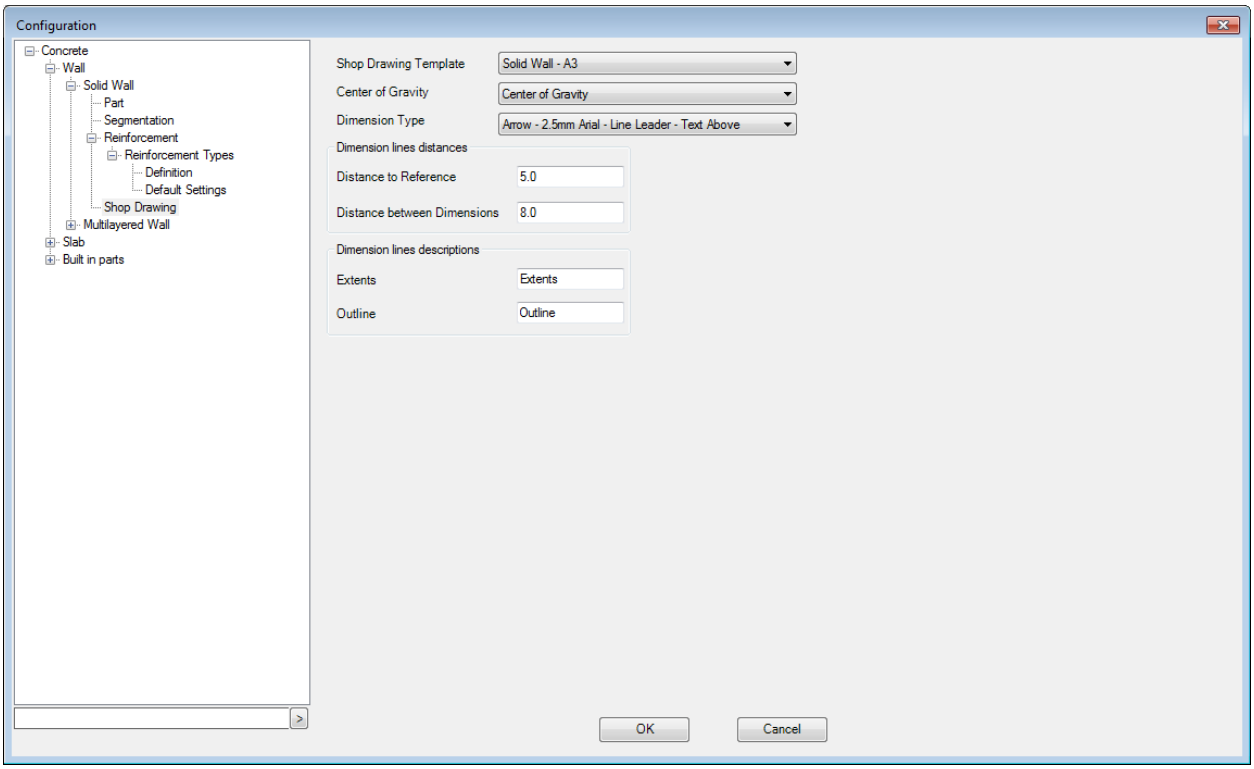


Note: The family might be available for adding description to the dimension line, but if the family doesn't follow the procedures to add dimension line helper inside the family (see Dimensioning Mounting Parts) there will be no dimension added in the shop drawings.

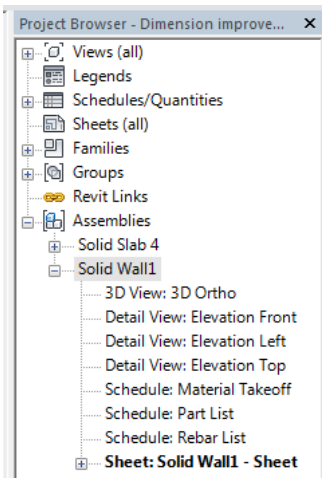
WALLS

In the *Configuration* tree dialog you can choose the title block for the shop drawings, the family positioned at the position of centroid or the dimensioning style. Furthermore, you can define the distance from the reference as well as the distance between dimensions. In the *Dimension lines description*, the user has the ability to add label for dimension for extents or outline.

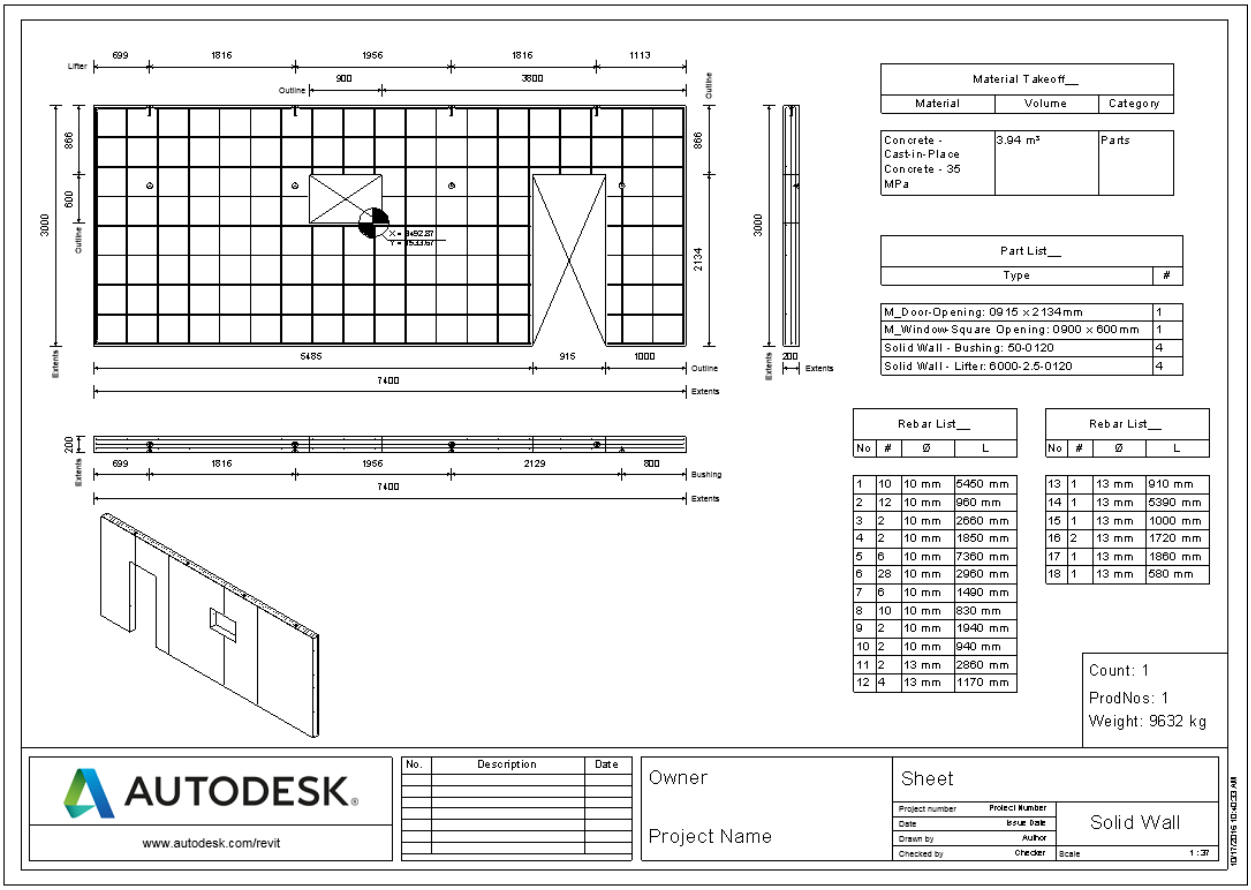
In order to be able to select the drawing template, the user has to copy the customized template at the following location C:\ProgramData\Autodesk\Project BRAN 2017\Families\en\Title Blocks and make sure that the name contains 'Solid Wall - ' and the desired name. Following this procedures ensures that in the drop-down menu you will only have the templates associated with solid walls.



To create the shop drawing you need to select all the assemblies and start the Shop *Drawings* command. The created shop drawings are located in the Revit project browser, under Assemblies.



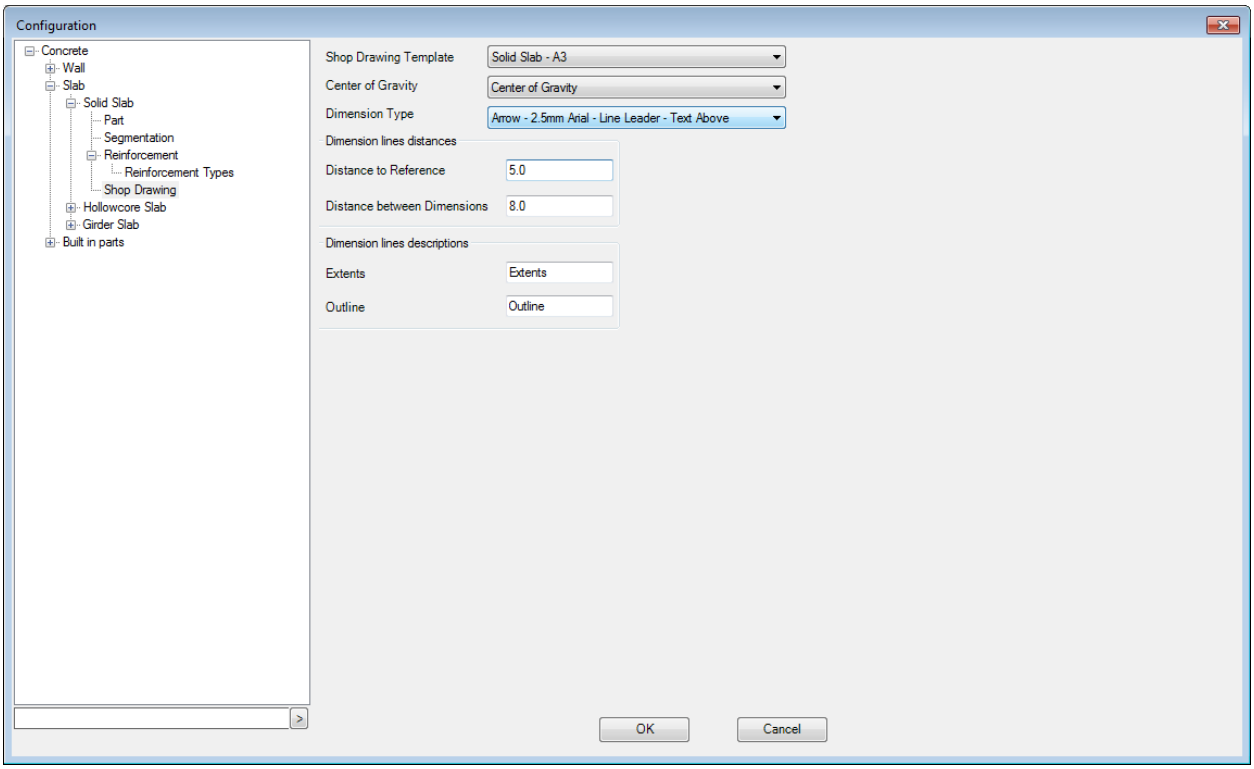
Below is an example of an automatically generated shop drawing:



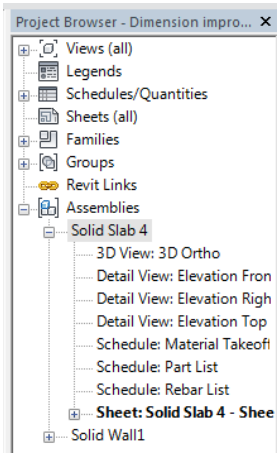
SLABS

In the *Configuration* tree dialog you can choose the title block for the shop drawings, the family positioned at the position of centroid or the dimensioning style. Furthermore, you can define the distance from the reference as well as the distance between dimensions. In the *Dimension lines description*, the user has the ability to add label for dimension for extents or outline.

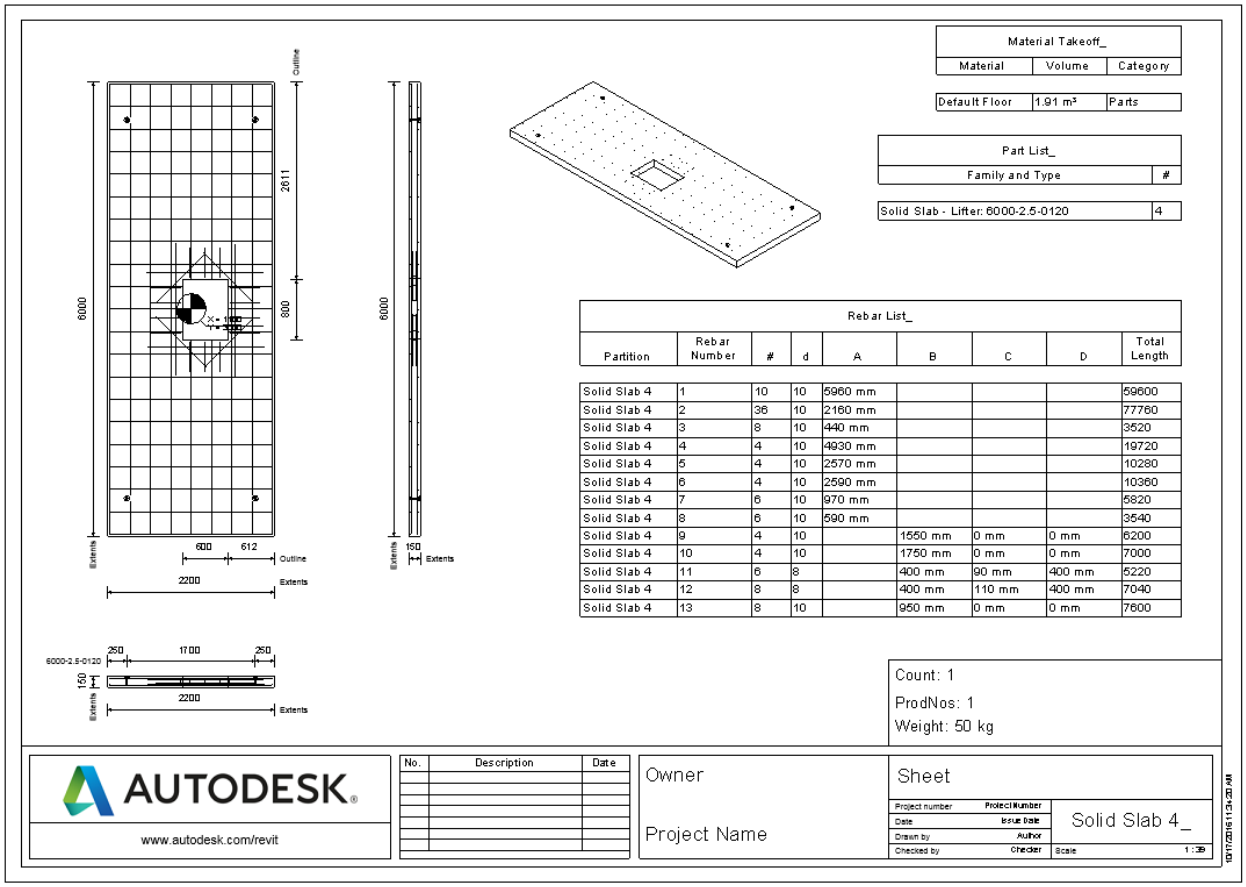
In order to be able to select the drawing template, the user has to copy the customized template at the following location C:\ProgramData\Autodesk\Project BRAN 2017\Families\en\Title Blocks and make sure that the name contains (e.g. 'Solid Slabs - ' for solid slabs and the desired name). Following this procedures ensures that in the drop-down menu you will only have the templates associated with solid slabs.



To create the shop drawing you need to select all the assemblies and start the Shop *Drawings* command. The created shop drawings are located in the Revit project browser, under Assemblies.



Here is an example of a shop drawing containing reinforcement around opening generated with REX:

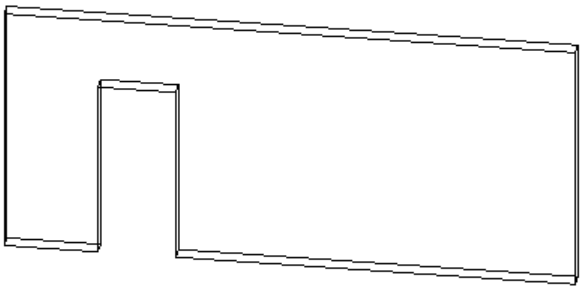


MANUALLY CREATED ASSEMBLIES

Project Bran 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:

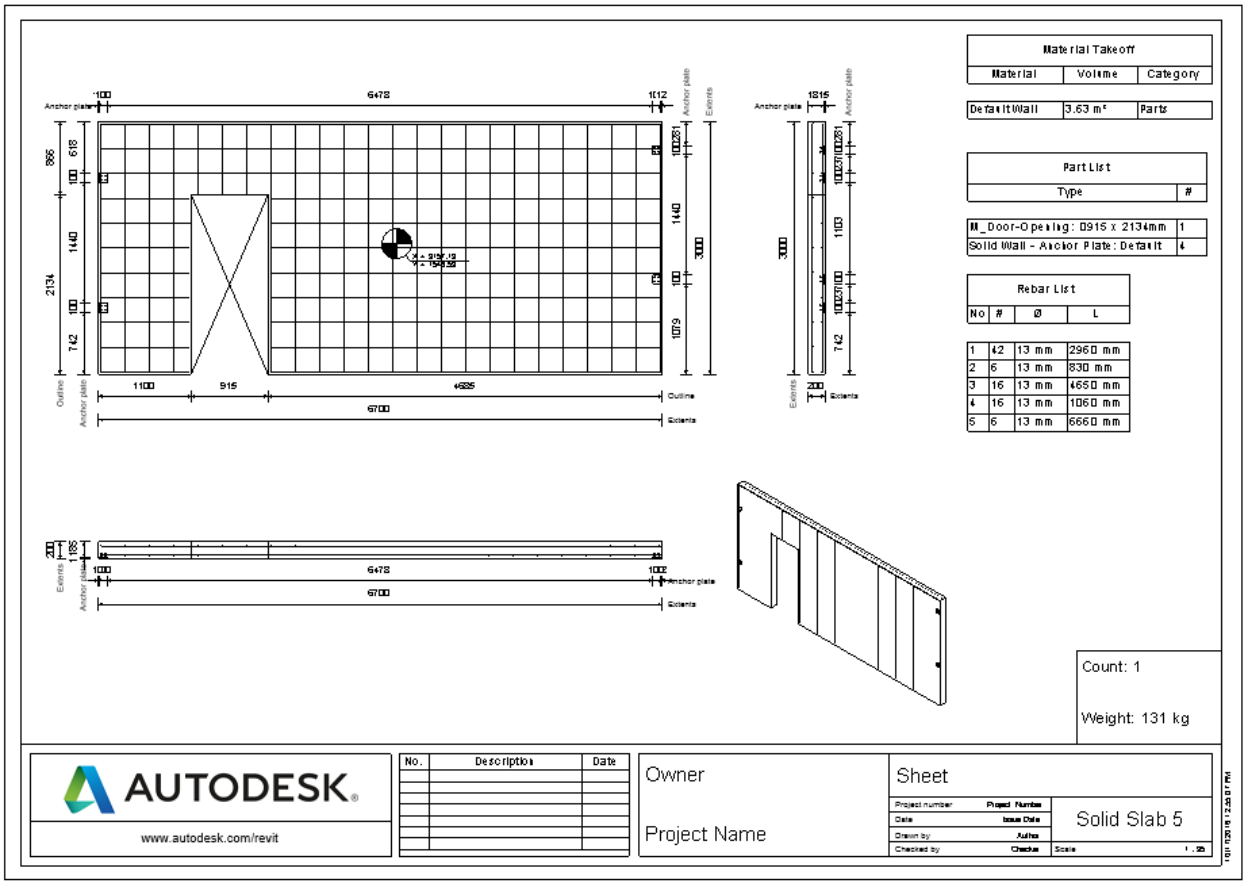
Step 1. Create a planar element (slab or wall).



Step 2. Create part(s) for the planar element.

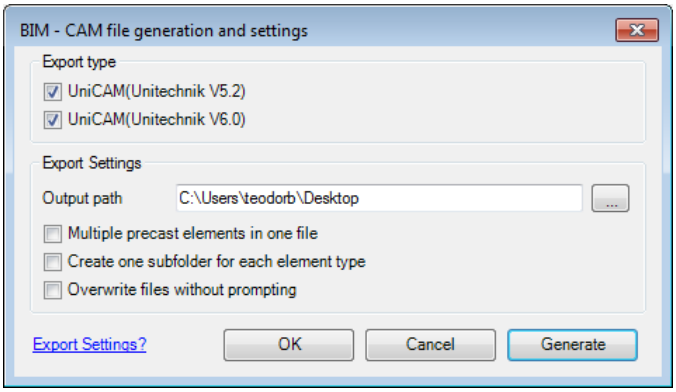
Step 3. Create the assembly containing the part and associated elements (mounting parts and reinforcement)

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.



CNC EXPORT

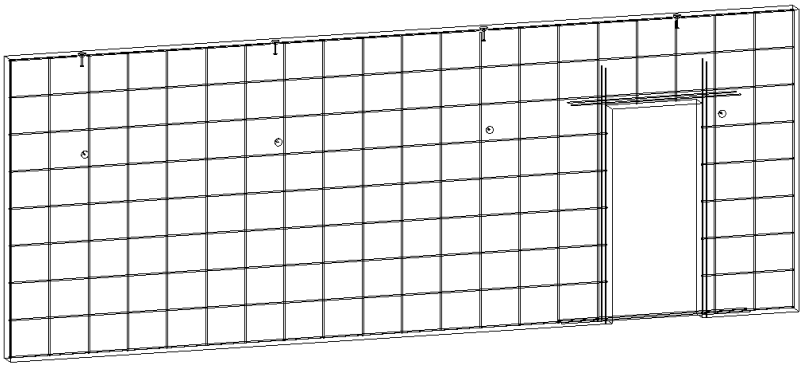
In order to create CNC files you need to select all the assemblies and start the CNC Export command.



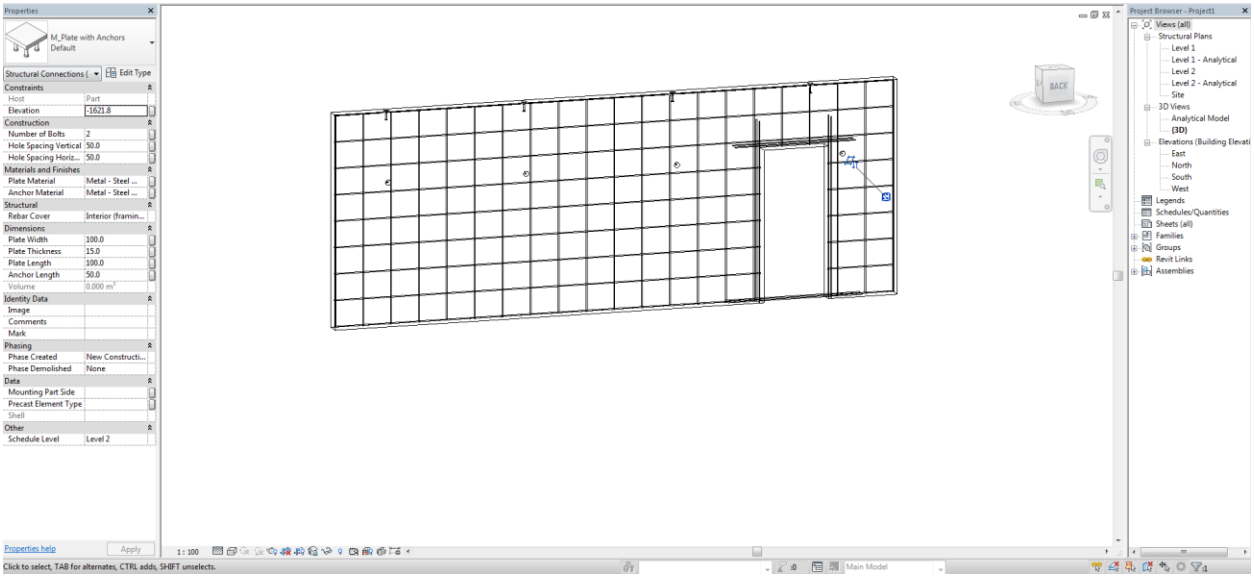
In this dialog you can select the export type (Unitechnik V5.2 or Unitechnik V6.0), the output path for the files and different option for when exporting. In order to generate the files you have to press the *Generate* button.

Workflow example:

Create a solid wall with the desired geometry then split and reinforce it.

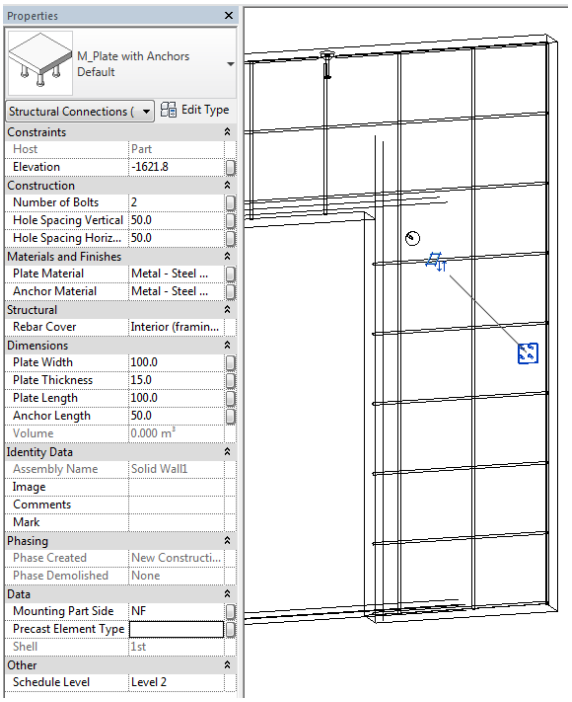


Add a mounting part on the wall (not in the assembly).

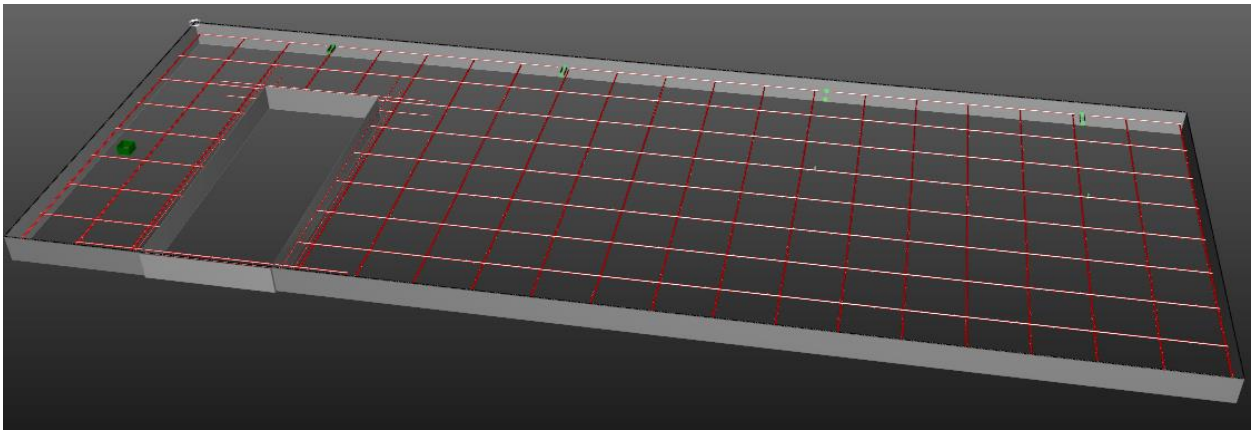


The mounting part does not have the data for ‘Mounting Part Side’ and ‘Precast Element Type’ populated. When using the Mounting Parts command the mounting part is added to the assembly and the relevant data is populated.

The near face and far face parameters are related to the viewing direction of the wall. On the viewing direction you will find the near face.



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

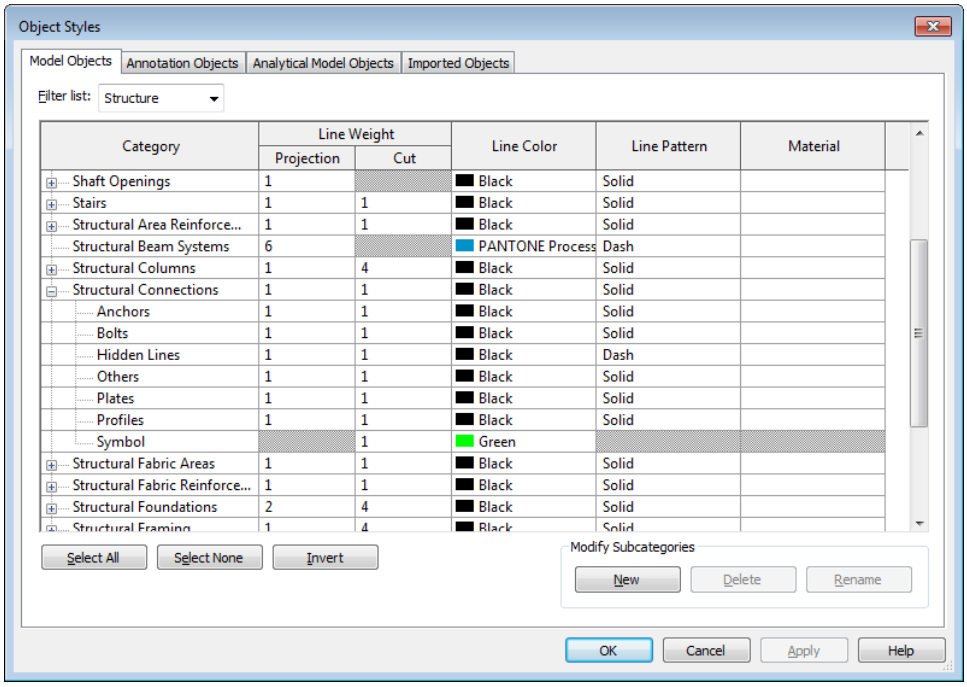
2. Define lines for the dimensioning

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:

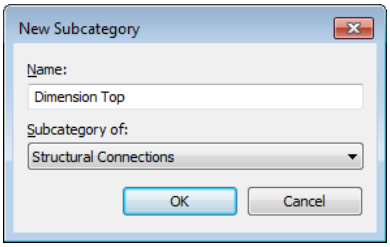
- *Dimension_Top* → View from top
- *Dimension_Front* → View from the front
- *Dimension_Side* → View from the side

2.1. Defining sub-categories

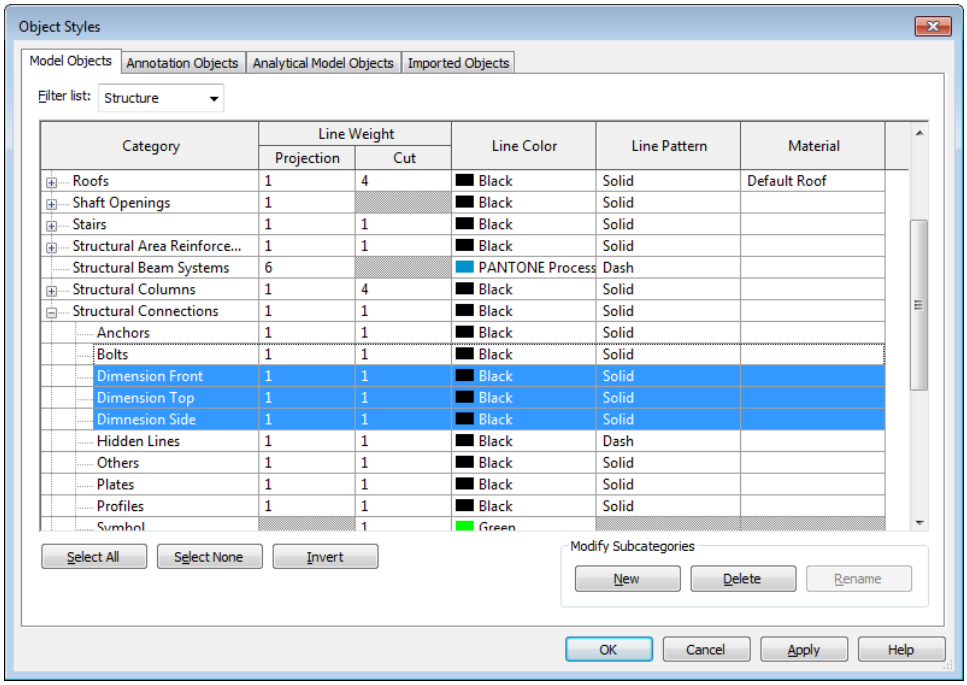
First one or more sub-categories with the desired names must be created: Manage → Object Styles:



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).



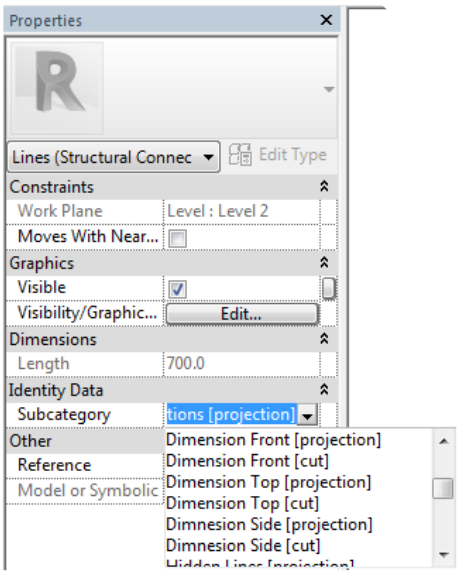
Repeat the steps in order to obtain the all necessary sub-categories.



Close Dialog *Object Styles* with OK.

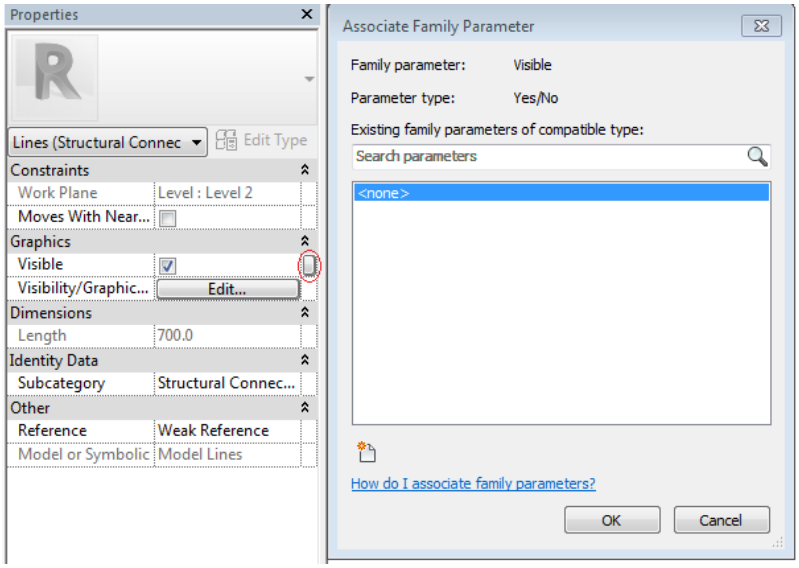
Create a family that you will later on use in your project.

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).

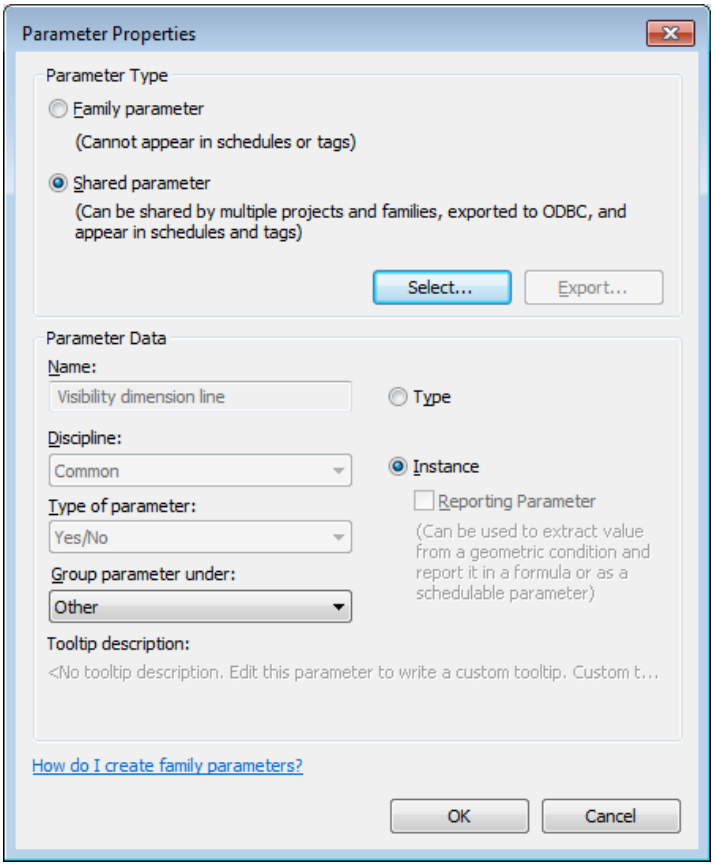


2.2. Define parameter for visibility

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":

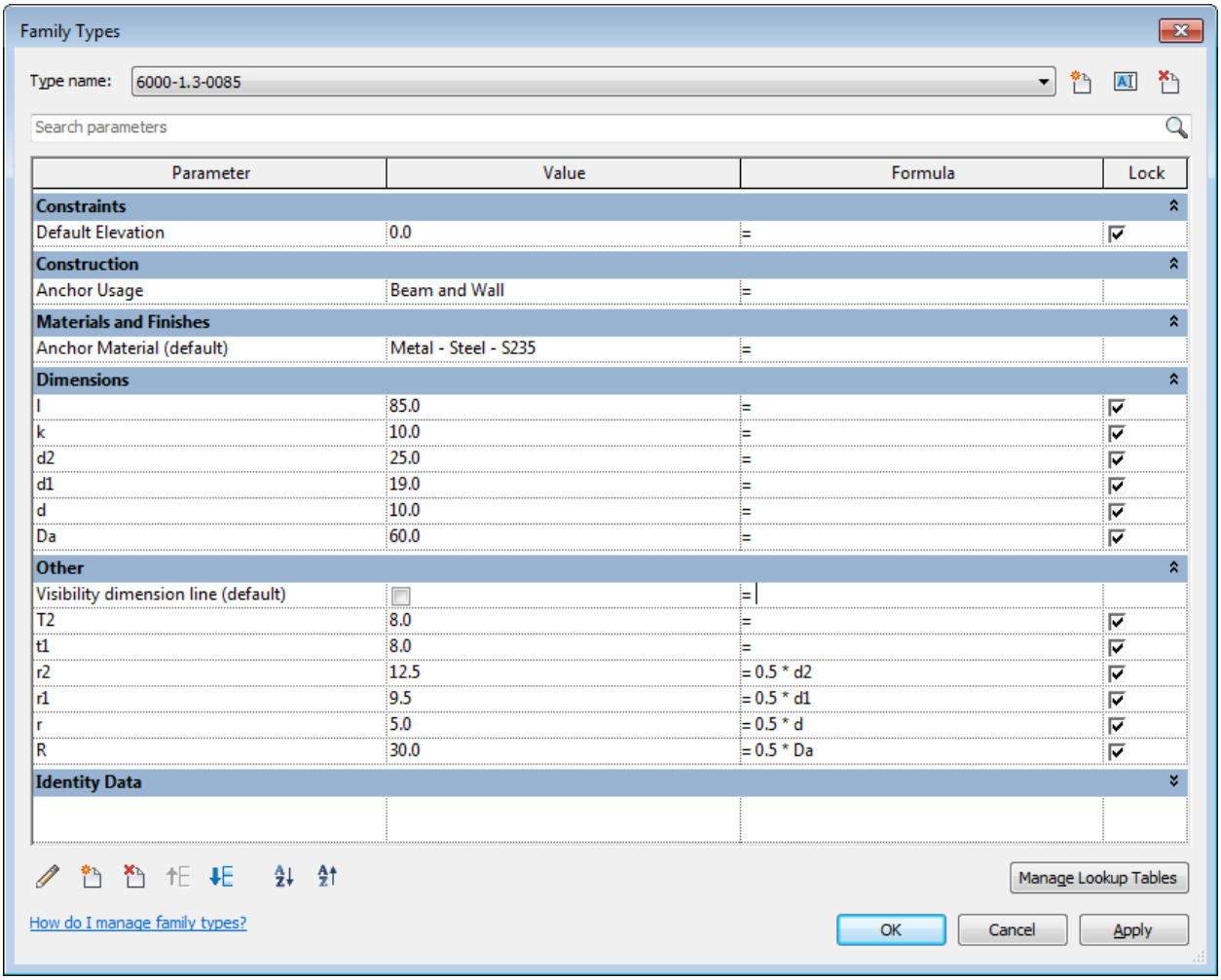


Select the parameter "Visibility dimension line" (if it is already in the list). If this parameter is missing, click on "Add Parameter" and add the parameter from the shared parameter file as an instance parameter.



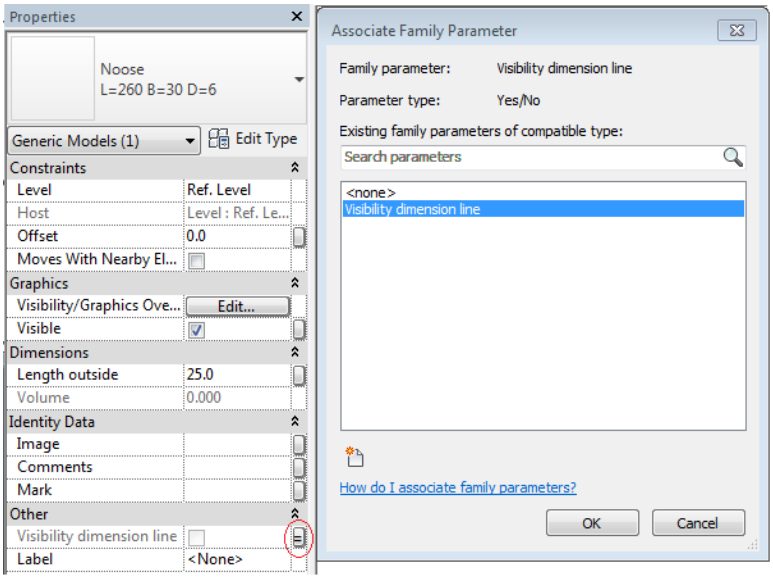
Close the dialog with OK, select the parameter in the parameter selection dialog and close this one also with OK. Repeat as necessary for all dimension lines.

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.

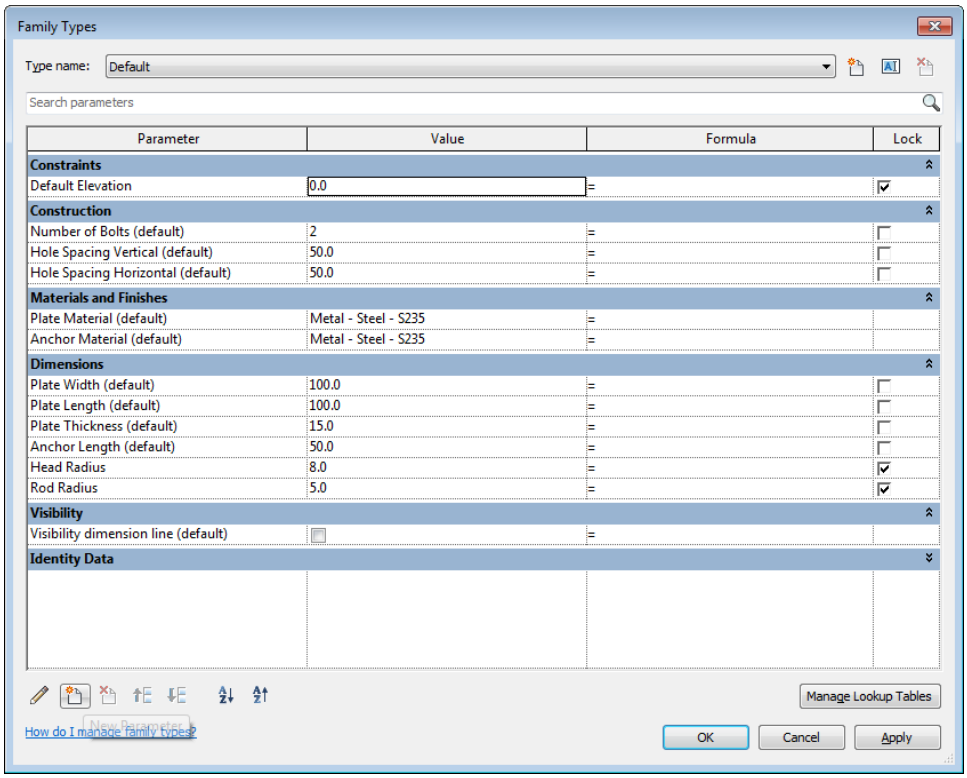
3. **Dimensioning the extents of a mounting parts**

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.

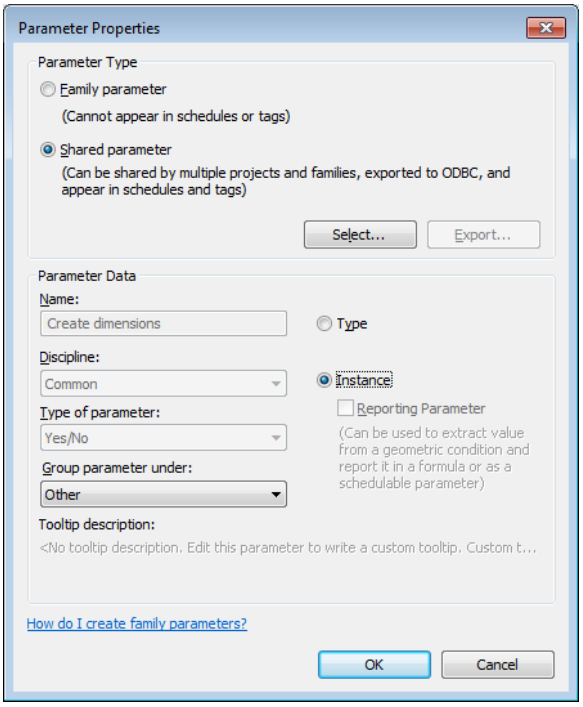
But if even one dimensioning line is defined in the mounting part, the extents will not be dimensioned at all (define dimension lines at the extents of the element if those are needed). If the mounting part cuts a piece out of an element, this hole is not dimensioned together with the main parts contour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

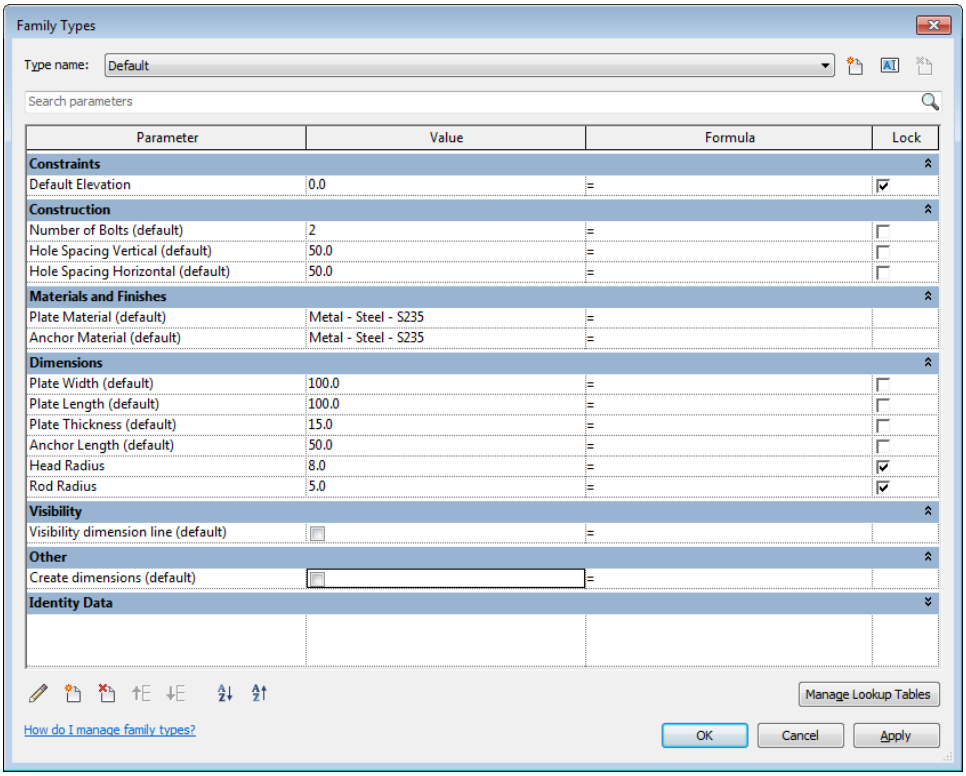
If a mounting part should not have its extents dimensioned, it can use the shared parameter “Create dimensions” as an instance parameter and set this to false. Modify Tab→ Properties Panel→ Family Types → New Parameter



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

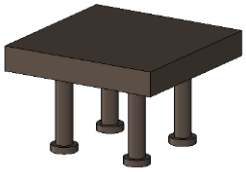
The dimensions of all mounting parts of the same family are dimensioned on one line, if they are placed on the same side of the host. Controlling for this is the center of the element. All mounting parts with the same family name on top of the center of the host are dimensioned on top of the element. 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.

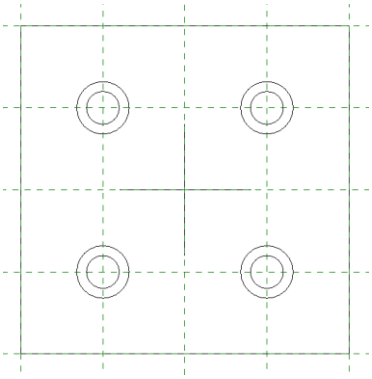
Step 1. **Open/Crete your mounting part family**

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**

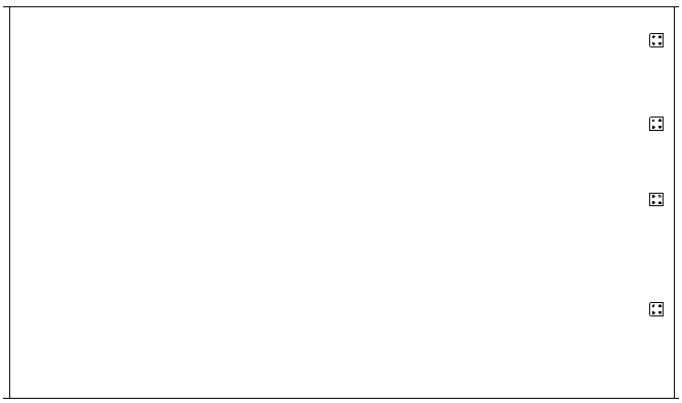
Switch to the Floor plans Ref. Level and create one horizontal and one vertical model line, intersecting in the center of the anchor plate:



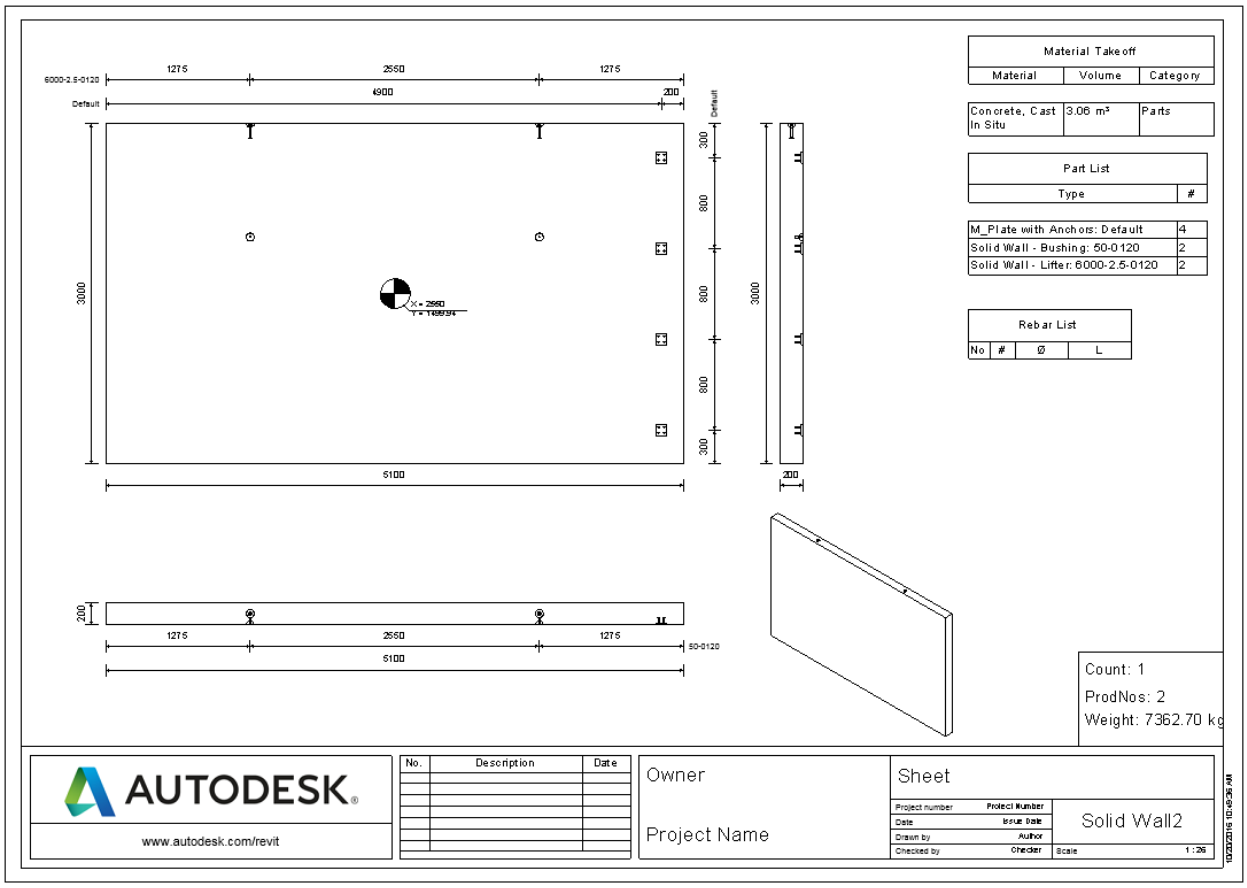
Define the sub category "*Dimension_Top*" and set it as the sub category for those two lines (see chapter 1). Now define the instance parameter "Visibility dimension line" and set the lines to invisible (see chapter **Error! Reference source not found.**).

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

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

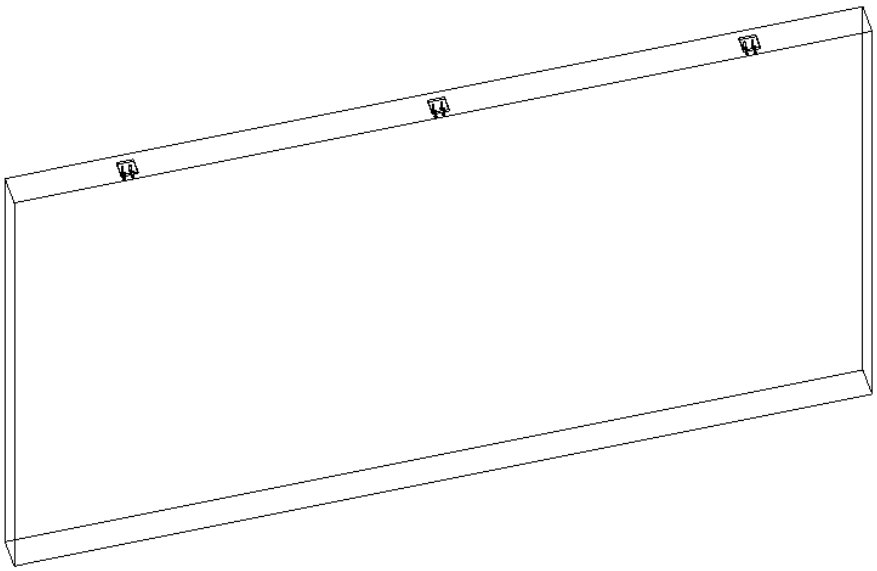


Split the wall, use the *Mounting Parts* command to add the anchor plates into the assembly and create a shop drawing for it. The center of the anchor plates is dimensioned in the view from top (on one line, since they are of the same family):

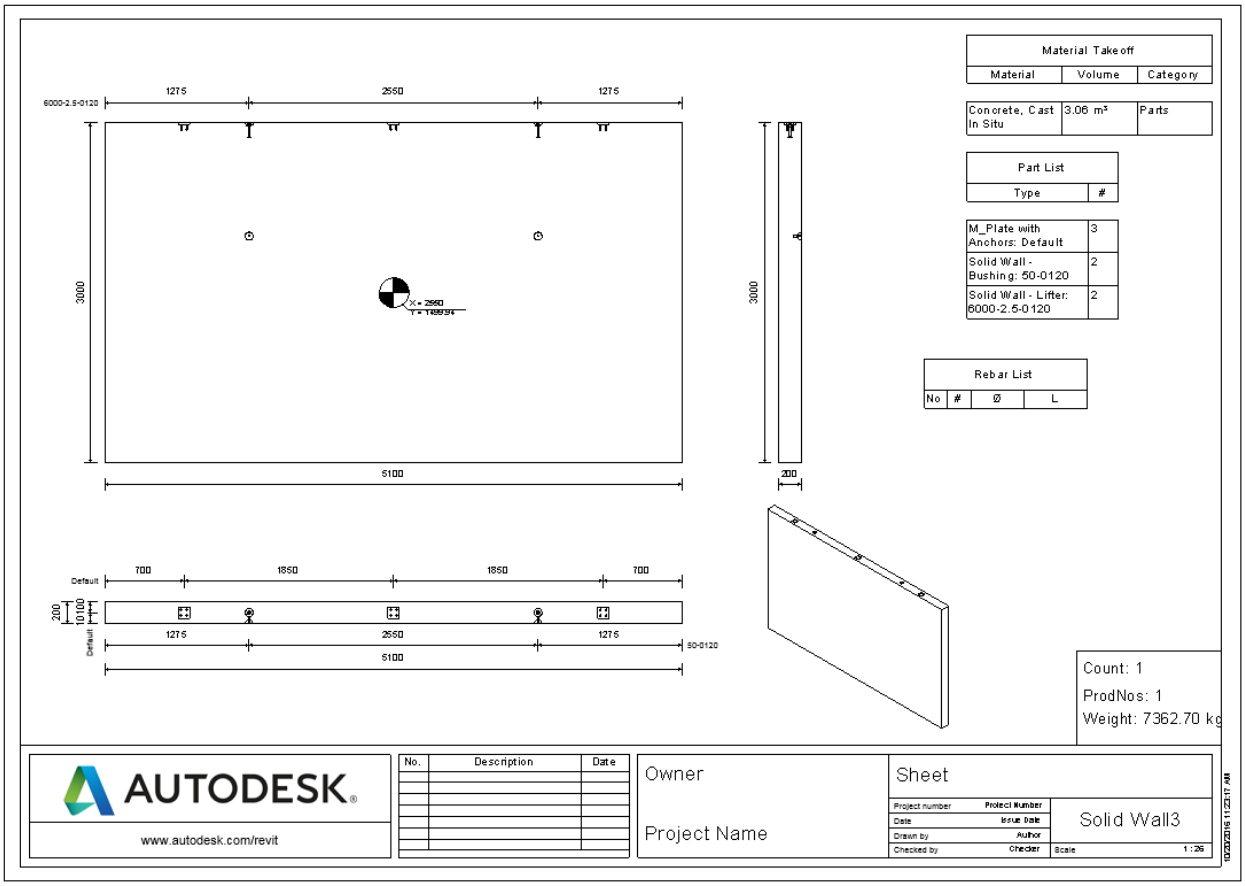


Based on the name of the sub category, the dimension lines should be created only when we are looking from the top (as seen in the family) at the anchor plates. For this wall that's only in the frontal view of the wall.

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:

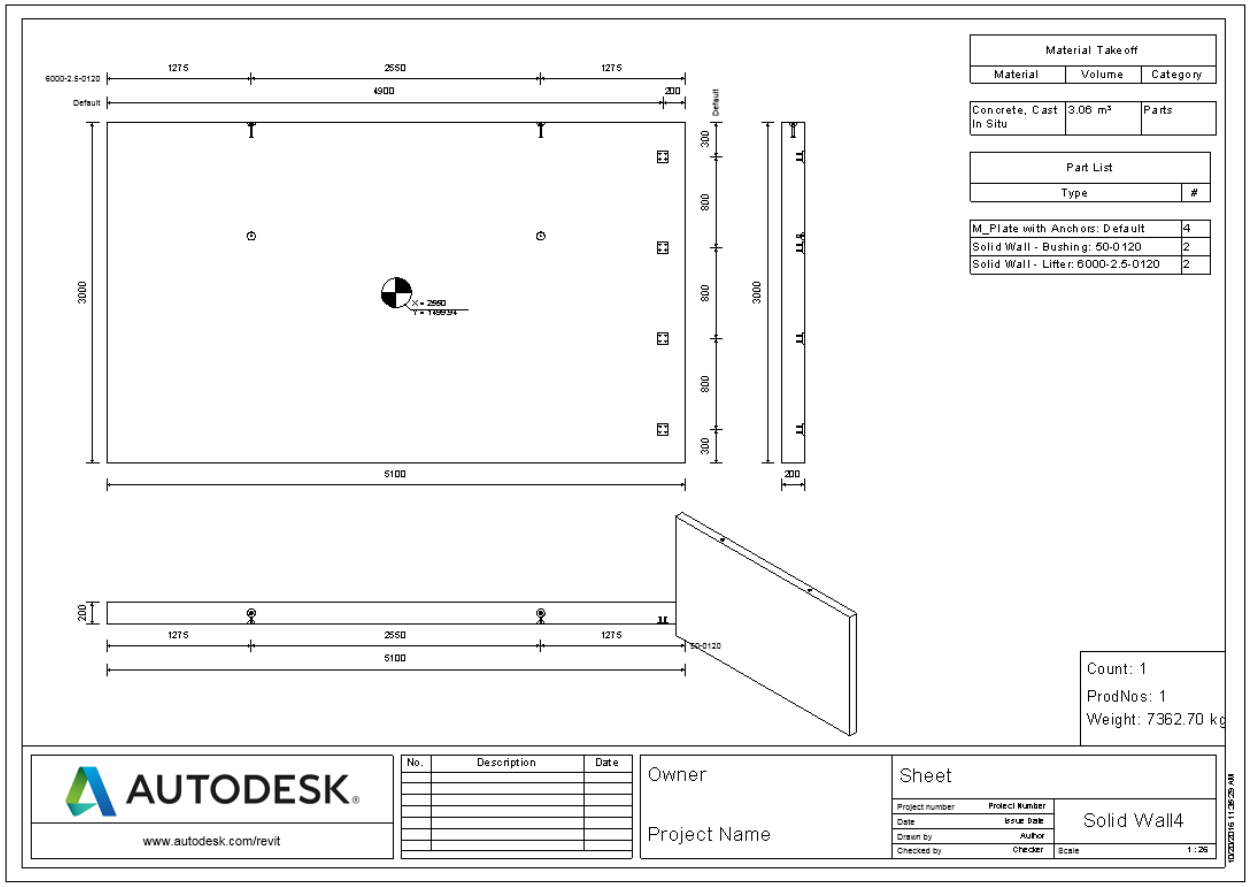


Step 3. **Dimensioning using only one dimensioning line**

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:

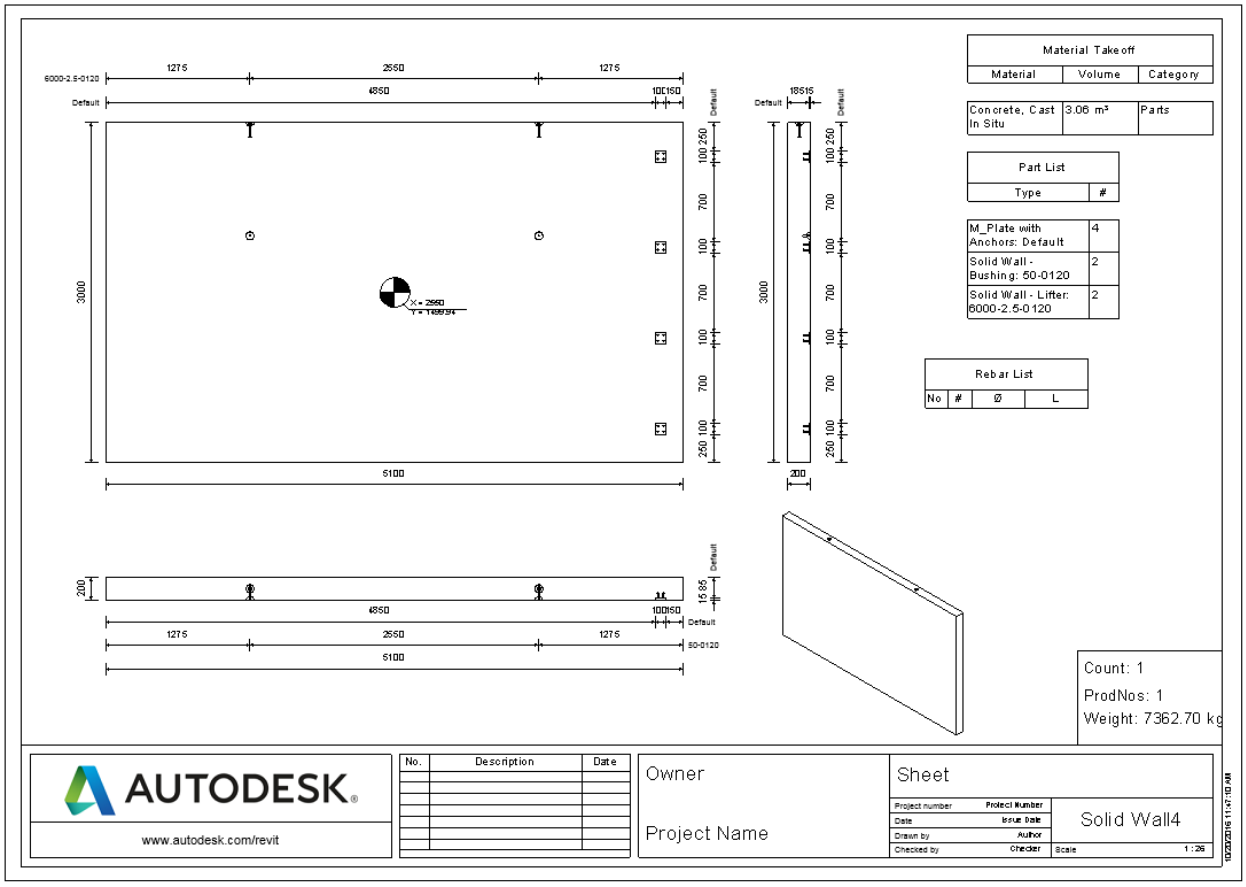


But there are two important differences:

- 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 4. **Dimensioning using the contour of the family**

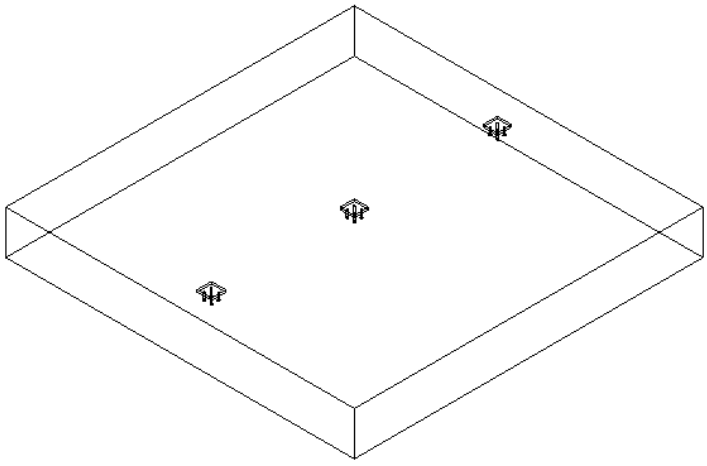
Remove the dimension lines and load the family into the same project, and create new shop drawings. Now the extents of the anchor plates are dimensioned in all views (using one dimension, since it's always the same family).



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 5. **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: