As-Built for Revit

FARO

SYNERGY
POSITIONING SYSTEMS

As-Built for Revit 2018.0

The information and guidance in this manual is currently the most up to date available. FARO Technologies, Inc. assumes no liability whatsoever for the use of this software, infringement of patent rights or rights of any third party that result from the use of this software.

FARO Technologies, Inc. reserves the right to make changes to this software, in order to bring about technical improvements in the software, at any time and without any prior notice.

No part of this documentation may at any time or in any form whatsoever be reproduced (printed, copied or the like), or edited, reproduced or distributed in any electronic system or manner without the express prior written consent of FARO Technologies, Inc..

Release: May 2018

Contact

FARO Technologies, Inc.

www.faro.com

www.knowledge.faro.com

Trademarks

AutoCAD®, AutoCAD® LT and Revit® are registered trademarks of Autodesk Inc.

BricscadTM is a registered trademark of Menhirs NV.

A few Words concerning this Manual

Dear As-Built users:

We thank you for the trust that you have placed in us and wish you lots of success in your daily work with As-Built for Revit.

You can be assured that we have done our best, to develop As-Built for Revit into a system, with which you can solve your tasks better.

To guarantee you a good start in using As-Built, the manual contains practice exercises, that lead you step by step through the displaying and processing of point clouds. Chapter 3 gives you an introduction to working with point cloud data in Revit and explains the steps necessary for preparing the data in Autodesk ReCap.

In the following sections you'll learn the most important functionalities of As-Built for Revit. Finally the command reference familiarises you with the most important details of all of the individual commands and enables you to work with your own projects. We therefore recommend that you spend some time at the beginning to go through these sections.

Please don't be afraid when you have questions or problems to take advantage of our support team. Your questions and suggestions on As-Built for Revit are very important to us and are always welcome. A system such as this is significantly influenced by the ideas and creativity of the users.

In this sense we are looking forward to working together with you and wish you much pleasure when working with As-Built for Revit.

Contents

1 What can As-Built for Revit do

As-Built for Revit is a Revit application for modelling 3D building models from 3D point cloud data.

The millions of points captured by 3D laser scanners can show there their significance, where the appropriate advanced modelling tools are available - directly in Revit.

As-Built for Revit significantly extends the Revit point cloud functions. The following functions are available:

- **Create walls quickly and precisely** directly in the point cloud. The software automatically selects the most appropriate wall type, based on the determined wall thickness. Should no appropriate wall type be found, a new adapted wall type will be automatically created by As-Built for Revit.

- **Automated alignment of walls for creating rectangular floor plans** (orthogonality) and continuous axial alignments, even over several floors. The relevant tolerances to be observed are defined by the user and can be subsequently reviewed in detail for each wall component.

- **Quick and precise creation of pipelines.** As-Built for Revit automatically selects the most appropriate segment type, based on the determined pipe diameter. If no suitable segment is found you can create a segment that fits the surveyed data.

- **Placement of pipe fittings and precise alignment of pipe runs.** With As-Built for Revit you create connected pipe runs, that on the one hand are fitted onto the point cloud and on the other hand comply with the Revit consistency constraints (coaxial axis runs and predefined angles for pipe bends and branch connections).

- **Fitting beams and columns.** As-Built for Revit precisely places structural elements in the point cloud. Beam and column profiles can be automatically recognised.

- **Create a digital terrain model** from the 3D coordinates and 3D model lines obtained from the point cloud.

- **Place family exemplars** such as **windows, doors, beams, columns** etc. directly in the point cloud using real 3D point snap.

- **Working with construction aids:** Create 3D model lines and construction points using real 3D point snap in the point cloud, independently of the Revit work plane. Work planes can be fitted by selecting just a single point or an unlimited number of points in the point cloud. Restrictions such as parallelism or orthogonality can be optionally set. Intersection lines and intersection points between the fitted planes or even between model planes can be easily determined.

- **Planar scan view:** Included in the supplied package is the stand-alone software VirtuSurv.

This allows a flat, high resolution scan view, similar to a panorama photo. In VirtuSurv 3D coordinates can be very easily read off and sent directly to Revit over a connection. Appropriate tools allow components such as walls, doors, windows or beams to be created directly in Revit using the VirtuSurv scan view.

- **Ortho images** from point clouds: A high resolution, photo like raster image can be created from a specific view of the point cloud, which can then be used as a true to scale image plan. A typical application is its use in the families editor as a model reference. The ortho image can be optionally computed as a colour image or in X-Ray mode, where the edges are significantly better visually discernible.

- **Surface analysis:** The deviation of the Revit model surface from the point cloud can be computed and clearly and graphically displayed. Exporting the analysis results allows them to be further used both in external applications and also within the project in the form of modelling assistance.

- **Working in the families editor:** Point cloud regions can be inserted directly in the families editor. Construction aids such as model lines or construction points can be easily created using the VirtuSurv scan view in the families editor. Scaled ortho images can also be used for the modelling of families. Model lines from the project environment can be saved as a modelling aid, directly as a families document.

- **Working with linked documents:** The coordinates for a "shared coordinate system" can be retrieved directly from the point cloud.

2 Installing As-Built for Revit

The installation of As-Built for Revit is comprised of several steps that must carried out one after the other. Administrator privileges are required for the installation.

2.1 System requirements

As-Built for Revit can be installed for Revit 2015 until 2019. You can use the Autodesk products Revit Architecture, Revit MEP, Revit Structure or the complete Revit version from the Building Design Suite.

You need a Microsoft® operating system, Windows 7, 8, 8.1 or 10 in the 64-bit version. To work with point clouds in Revit, Autodesk recommends a multi-core Intel® Xeon®, or an i-Series processor, or the AMD® equivalent with 16 GB RAM and DirectX® 11 compatible graphics card and SSD (Solid State Drive) hard disk with at least 5 GB of free memory.

Please take note that FARO cannot offer any support for combinations of a Revit version and an operating system which are **not** supported by Autodesk.

2.2 Installation

The current setup files of As-Built for Revit can be downloaded from the FARO internet site:

www.faro-3d-software.com

Please run the setup file of As-Built for Revit by double clicking and following the instructions of the installation program.

Start Revit after the installation is complete. The "As-Built" tab appears automatically in the user interface of Revit.

2.3 Installation of VirtuSurv

To be able to work with scan views As-Built for Revit uses the VirtuSurv software. When installing As-Built for Revit you can simultaneously install it automatically. Should this not happen correctly, you can subsequently install it manually. To do this run the setup file for VirtuSurv from the installation folder of As-Built for Revit, by double clicking it and then follow the instructions of the installation program. The current VirtuSurv setup files can also be found on the FARO internet site.

3 Working with point clouds in Revit

This section gives you an introduction to working with point clouds in Revit, and describes the basic procedures for the modelling process "Scan to BIM". You will build on these foundations in the next chapter and there learn and apply the individual modelling steps in As-Built for Revit in various tutorials. A full explanation of all of the commands can be found in the **5 As-Built for Revit Command Reference**

3.1 Point Clouds in Revit

3.1.1 Inserting point clouds

A point cloud object can be directly inserted into Revit, if it exists in a ReCap format (RCP or RCS). It is also possible when importing to select various raw scan formats (e.g. E57, FLS, LAS etc..). These are first, in a background process, converted into the ReCap format before the point cloud can be subsequently inserted.

The point cloud data are only referenced into the project. Please note, that when you pass on projects with point clouds that you must also pass on the referenced point cloud data. Otherwise the point clouds cannot be displayed by the person receiving the project.

To insert a point cloud into the current project, select Insert tab → Link → Point Cloud .

Select the appropriate point cloud file and the desired type of positioning and then click on **[Open]**.
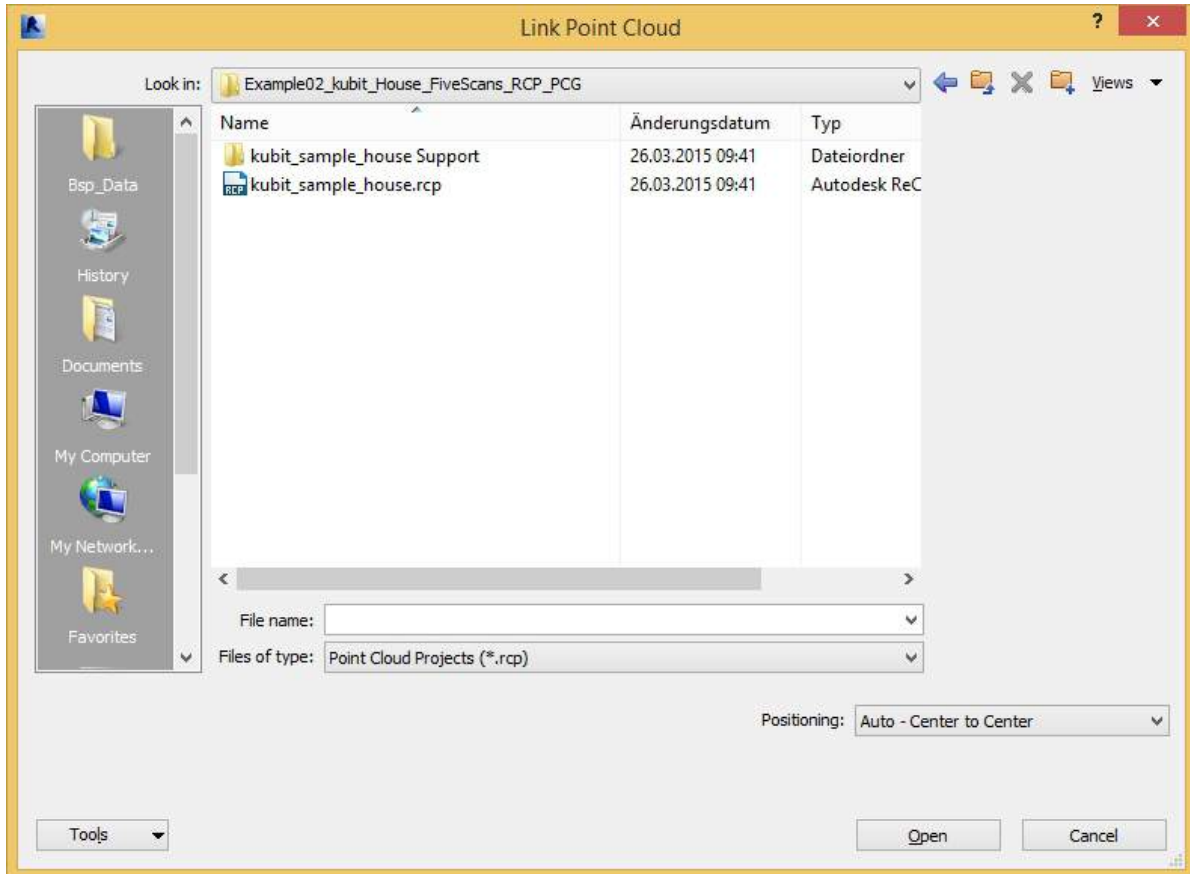
It is essential that you pay particular attention to the chosen type of positioning of the point cloud. Here, you have the choice of four different options:

■ Auto - Center to Center

The point cloud is placed so that the centre point of the bounding box of the point cloud is placed on the centre point of the bounding box of the model. If there are no objects that have been already modelled, the centre point of the model is the project base point.

■ Auto - Origin to Origin,

The point cloud is so placed, that the origin of the internal point cloud coordinate system is at the origin of the Revit project. Provided that the project base point has not yet been changed, it is then identical to the origin of the Revit project. The Y - axis of the internal point cloud coordinate system is aligned to the current project north.

Attention: Should the point cloud be referenced in a surveying coordinate system (geo-referencing), this could in some cases mean it being very far removed from the project base point. In Revit it is inherent in the system though that objects can only be modelled up to a distance of approximately 20 miles (approx. 32 km) from the project origin. Point clouds whose coordinate values lie in this region, can only be limitedly used (see also chapter **3.1.2 Point clouds with large coordinate values**).

■ Auto - By Shared Coordinates

The point cloud is so placed that the origin of the internal point cloud coordinate system corresponds to the origin of the shared coordinate system. The Y - axis of the internal point cloud coordinate system is aligned to geographic north of the shared coordinate system.

■ Auto - Origin to Last Placed

This option is only available if you have already inserted a point cloud into the project. The point cloud that is to be inserted will inserted relative to previously inserted point cloud, even if it has been moved after insertion.

When working with As-Built for Revit it is extremely important that you work with a shared coordinate system corresponding to the internal point cloud coordinate system. In this way it is possible to insert other external data (e.g. a site plan as a CAD drawing) relative to the coordinate system in the Revit project, and to export other drawings and models from Revit also relative to the to the point cloud coordinate system.

You have two ways in As-Built for Revit to set up such a shared coordinate system:

1. Insert the point cloud with the option "Auto - Center to Center". You can subsequently rotate and move it at will, to bring it into a more favourable position for processing (e.g. it is recommended to align a long building axis with the east-west direction of the Revit project). With the command → **Point Cloud - Coordinates - Acquire PC Coordinates** you can now align the shared coordinate system with the internal point cloud coordinate system. To now connect the point cloud with the VirtuSurv project select the command → **Point Cloud - Coordinates - Linking PC Coordinates**

2. If you start working on the project in VirtuSurv, you can by using the command → **VirtuSurv - Set Project Coordinates** set the origin and alignment of the shared coordinate system. Should you subsequently insert the point cloud into Revit, use the option "Auto - By Shared Coordinates".

3.1.2 Point clouds with large coordinate values

It is inherent in the system that Revit 2015 - 2016 cannot correctly process point clouds with large coordinate values, such as often happens with surveying coordinate systems (e.g. Gauß-Krüger coordinates). Revit can however display point clouds with large coordinate values, although various problems occur:

■ The Revit point cloud snap does not function

■ The point cloud, will to some extent, be inserted into Revit with an offset of several centimetres from its correct position.

■ The Revit programming interface (API) gives false coordinate values to As-Built for Revit. All commands that interact with the point cloud are thereby unusable.

In Revit 2017 these problems are as far as possible resolved, never the less an appropriate warning message is displayed, when a point cloud with large coordinates is to be inserted into the project:
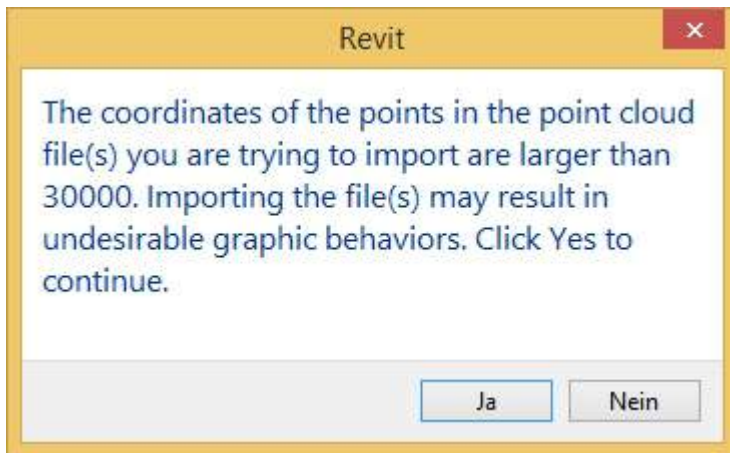
When however you still want to work in such a surveying coordinate system, it is recommended that it be done as follows:
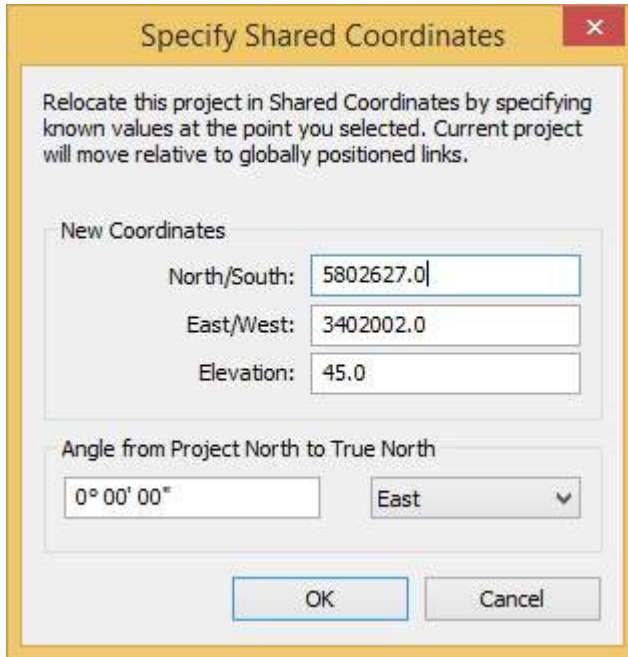
1. Reduce the coordinate values in you registration software before exporting to ReCap format. When the registration has been done in the surveying coordinate system using geodetically surveyed targets, you can, for the registration, simply remove the first few figures of the coordinate values of the targets (the coordinate values must not be larger than 30 km).

   Example:

   ```
   From (X (east/west) = 3402002 m; Y (north/south) = 5802627 m; Z (height) =
   45 m)

   results (X (east/west) = 2002 m; Y (north/south) = 2627 m; Z (height) = 45
   m)
   ```

2. Insert the point cloud into Revit with the option "Auto - Center to Center". You can subsequently rotate and move them as you will, to bring them into a more favourable position for processing. With the command → **Point Cloud - Coordinates - Acquire PC Coordinates** you can now align the shared coordinate system with the internal point cloud coordinate system.

3. To customise the shared coordinate system so that it once again is on the original, non reduced coordinates, you can by using the Revit command "Specify Coordinates at Point" (tab Manage → panel: Coordinates) the reduced figures can once again be entered. Here it is best to select the Revit project base point.

3.1.3 Display and clipping point clouds

The display and clipping of point clouds in Revit can be controlled independently for each view. To do this open the settings for visibility/graphics (tab View → Graphics → Visibility / Graphics):
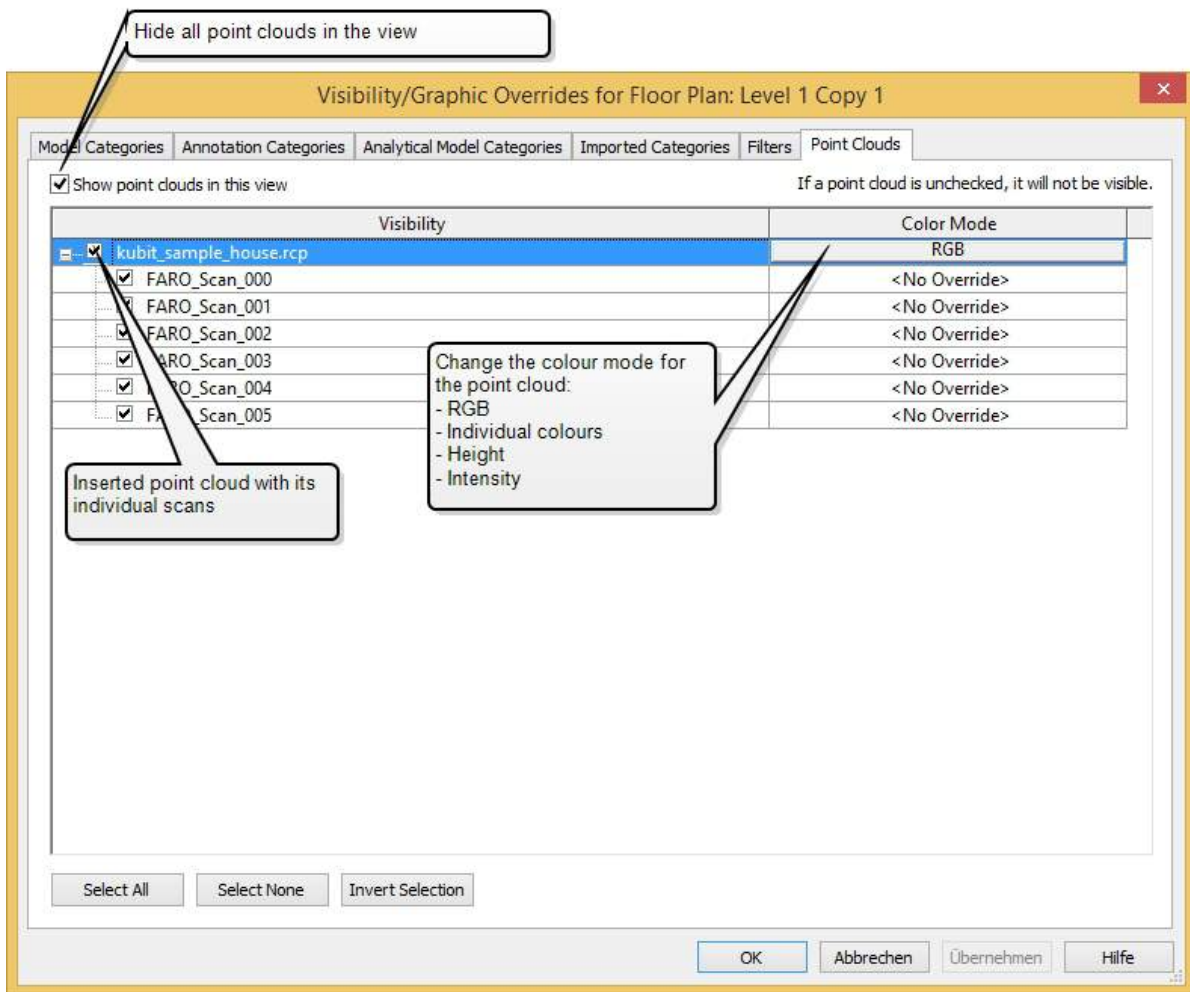
Attention: When working with As-Built for Revit, you should avoid hiding individual scans or regions. As-Built for Revit provides a 3D point snap for some of the commands. Because of the limitations of the Revit API (application programming interface) it is not possible to take into consideration any hidden individual scans or regions. It is possible in this case, that As-Built for Revit snaps to points that are in a hidden individual scan or region. The hiding of complete point cloud objects (RCP) is not affected by this.

Point clouds in the Revit can be sectioned in the same way as all other Revit model elements. The visible area of a point cloud is limited in the 3D view by the extents of the section box and the cropped region:

In planar views, as for example floor plans, elevations or sections, the visible area of the point cloud is limited by the properties of the view range of the current view. In floor plan views it is recommended that the bottom of the view range is placed a few centimetres above the view plane. Thereby the point cloud points on the floor will be excluded and the building floor plan is more easily recognisable:
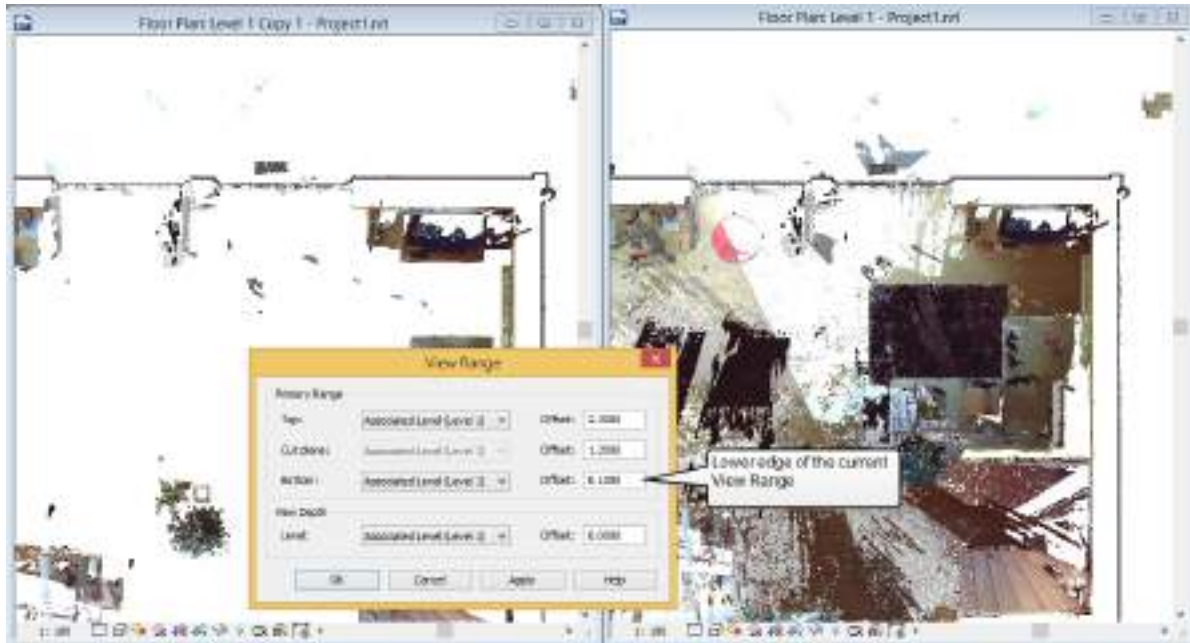
Displaying the point cloud in a plan view: The bottom of the view range with an offset of 10 cm (left) and no offset (right).

The 3D point snap of As-Built for Revit takes into account the clipping of point clouds in a view range, section box and the clipped region of the current view.

3.1.4 Working with the As-Built Section Manager

When modelling in the point cloud it is often meaningful and necessary, independently of the model, to limit the visible region of the point cloud. The section tools together with the Sections Manager in As-Built offer numerous features, to divide the point cloud into named sections and if required to hide and display them.

The clipping of the point cloud is done independently of the views defined in the project. The clipped regions will be saved as named sections in the Section Manager and saved together with your project. It is at anytime possible and in every 2D or 3D view to access any one of the defined sections. It is also possible to divide the sections hierarchically. You can find detailed information on the definition of sections and their management in the Section Manager, in the Command Reference (→ **Regions - Manager**) at the end of this manual.
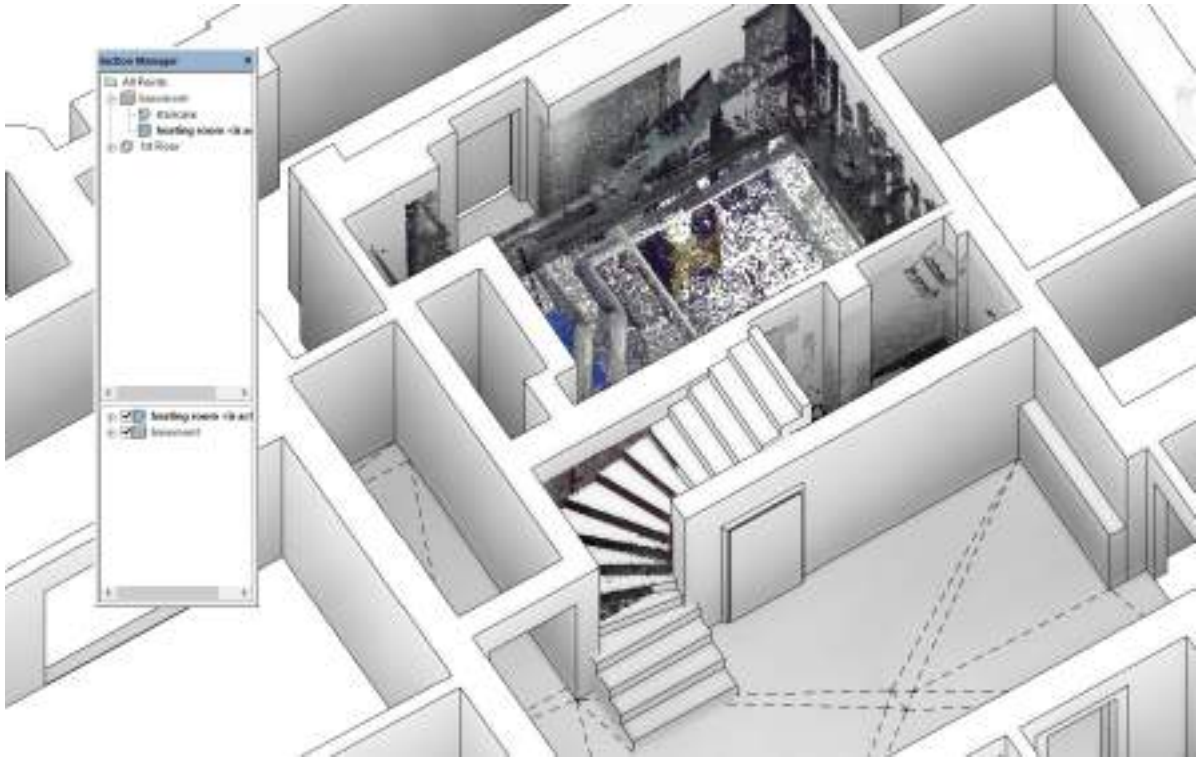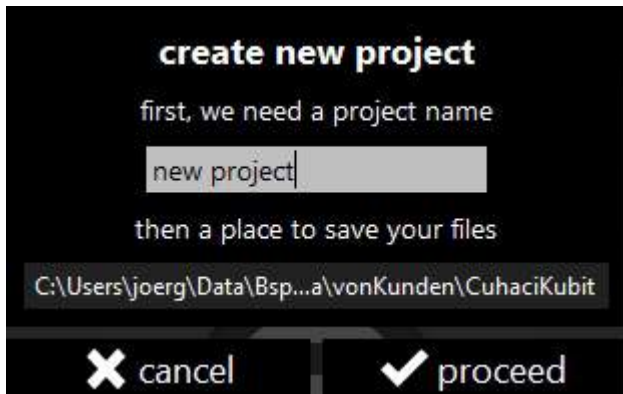
Model section with point cloud section and Section Manager

3.2 Preparing the point clouds using Autodesk ReCap

To create point clouds in the Re Cap format, start Autodesk Recap and click on "New Project".

Enter a project name and select the location where it should be saved. Then confirm with "proceed":

Select the scan files to be imported using the button "import files" or "select folders for import". You can also directly place the files in the import window using "drag and drop". ReCap supports most of the popular scan data formats, including the ASCII format PTS, the standard data format E57 and the older Autodesk PCG format.
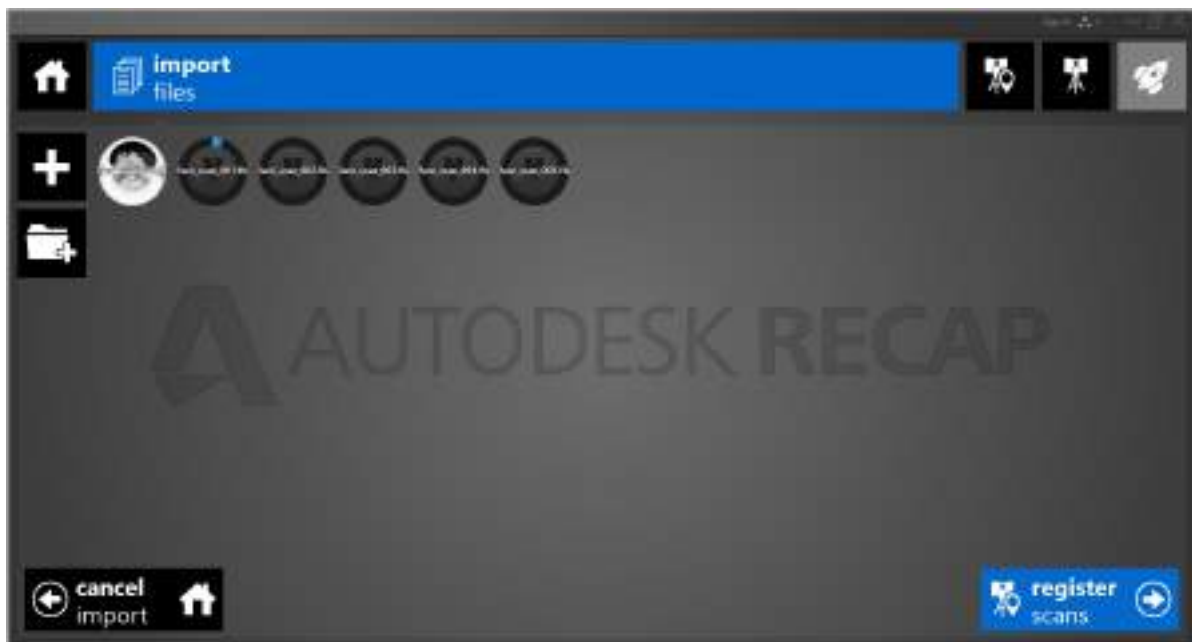
As soon as the import has been started, you can continue with the registration of the scans. Should this have already been done in other software, you can skip the registration in ReCap and proceed directly to the indexing. Move the mouse cursor over the button "register scans" and click on "skip registration" and then click "index scans".
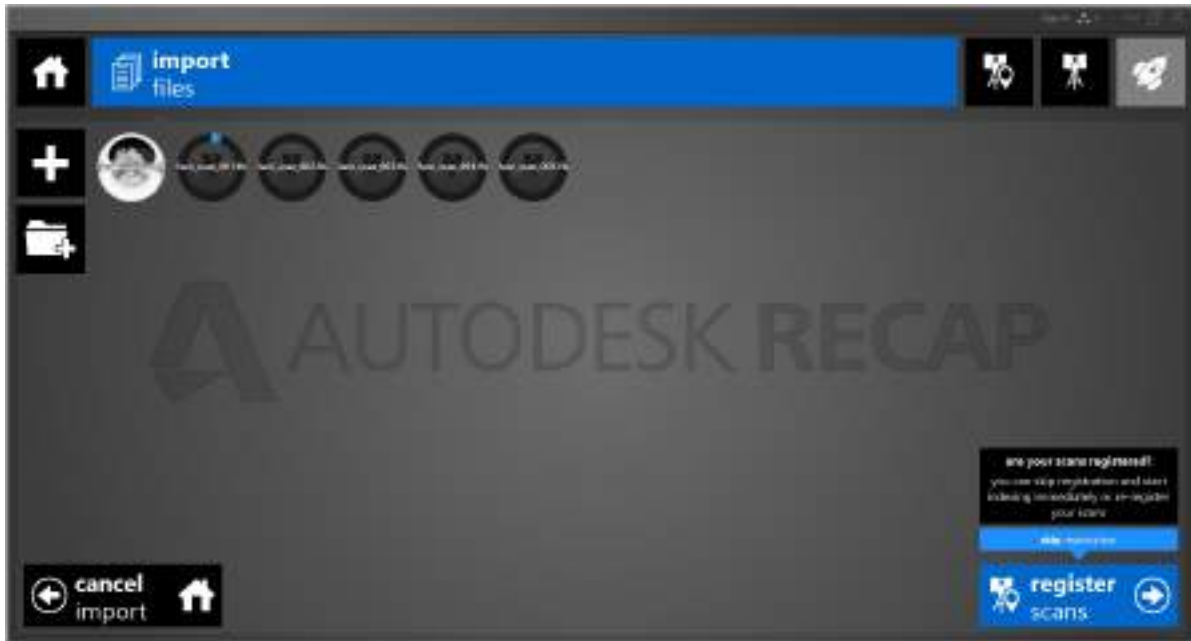
During indexing the scan data will be converted into the ReCap format. As soon as the first scan has been successfully imported and indexed, you can start the project.
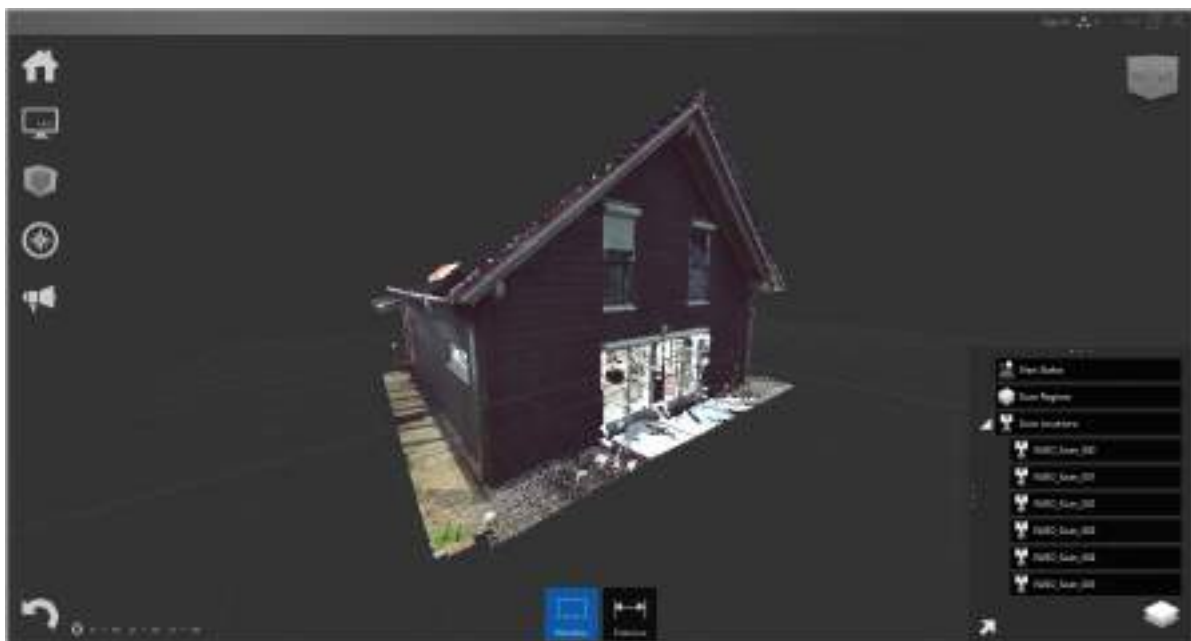
ReCap creates a RCP file (ReCap Project) and for each imported individual scan a RCS-file (recap Scan). The RCP file references each individual RCS file. Both the RCP file as well as the RCS files can be inserted into Revit.

In ReCap you can use the function "bounding box" and the "cropping tool" to limit the display of the point cloud to specific areas. Active clippings will also be taken into consideration after the import into Revit, the bounding box however not. In Revit you have however no access to named scan regions.

Additional information about Autodesk ReCap can be found at the following link:

http://docs.autodesk.com/CLOUDS/1.0/ENU/AutodeskReCap/index.html

3.3 Typical Workflow Scan-To-BIM

For each new project, that you do using As-Built for Revit, there are several preparatory steps which are necessary.

Setting up ver-
tical sections

Placing levels

Creating super-
ior geometry

Create components in
an aligned view

Create families and
in-place families

The majority of modelling tasks in Revit are based on orthogonal views (floor plans, sections or elevations), therefore you should first define the important views at the beginning of the project.

3.3.1 Setting up vertical sections

When you start a new Revit project from a template, there are already multiple levels defined in the project. These level must first be fitted onto the point cloud, or new levels must be created. The best way to do this is in a section view.

Insert the point cloud into your project using the option "Auto - Center to Center" (see also **3.1.1**), and then change to the "Site" view. Here you have the best overview, because the view boundaries are set to a large area.

Before the section is created, the display of section lines must first be activated in the "Site plan" view. To do this open the dialogue box "Visibility/Graphics Overrides" from the "Properties" palette, and activate "Sections" in the tab "Annotations categories".
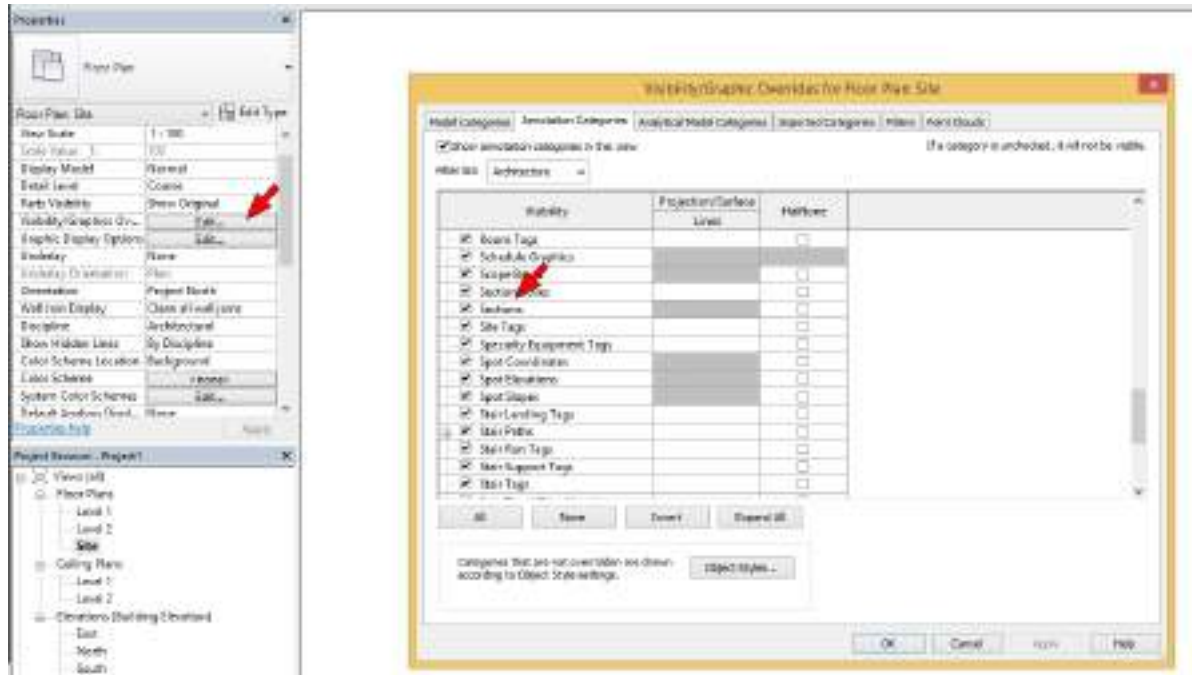
Now create a new section line in the site plan (Ribbon, View tab → Create Group → Section ◐) and place it so that it makes a large cross-section through the point cloud. The section width can be set relatively small, as the section will first be used only for determining the floor levels:

Change to section view

3.3.2 Placing levels

The levels can now be appropriately fitted to the point cloud in the newly created section view. First the point cloud should be moved so that the height of the scanned floor at ground floor

level has the height 0 in the Revit Model. For this you need to select the point cloud and choose the command "Move" [icon] (Tab Modify| Point cloud → Modify Group). Select a point on the floor

as the start point for the move. With the activated point cloud object snap (can be set from the tab Manage → panel Settings → Snaps [icon]) Revit finds a point on a meaned plane in the point

cloud. As the target point for the move select a point on level 0:

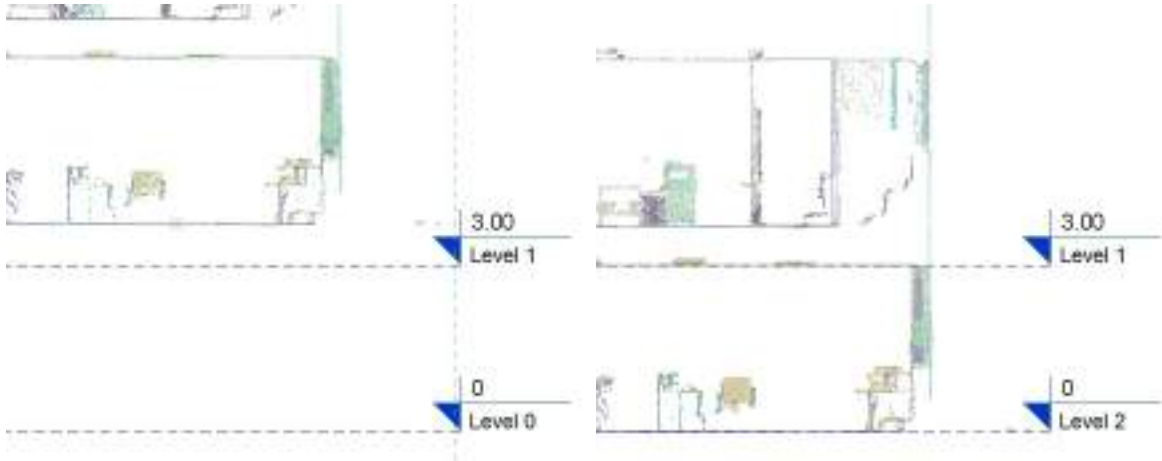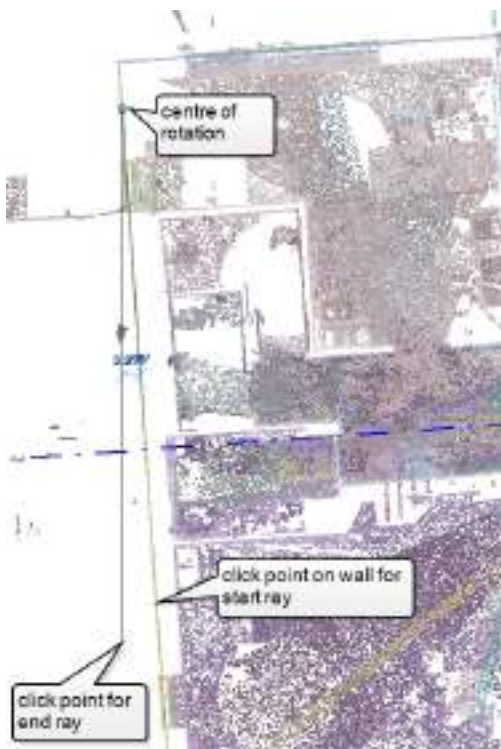Thereafter you can set the height of all other further levels from the point cloud or create new levels in the section view as required.

3.3.3 Setting up the project coordinates

Now switch to the plan view for level 0. Should your point cloud be rotated in relation to project north, than it should now be aligned. For this you need to select the point cloud and choose the command "Rotate" [icon] (Tab Modify| Point cloud → panel

Modify). First place the centre of rotation (using the button [Place] in the options bar) on a clearly visible external wall face in the point cloud. Once again use the point cloud snap. Then you can select a second point on the same external wall face to define the start ray for the rotation. This point should, where possible, lie a long way from the rotation point. Now you can pick a point in the project, so that the end ray of the rotation is set orthogonal to the project north.

The point cloud is now aligned in height and rotation to the Revit project coordinate system. To be able to

restore the reference to the point cloud coordinate system, the "shared coordinate system" of Revit should be appropriately aligned.

To do this select the As-Built command ➔ **Point Cloud - Coordinates - Acquire PC Coordinates** and then pick in the point cloud. The "shared coordinate system" is now aligned to the point cloud coordinate system.

You can check this by selecting the project base point in the "Site plan". The coordinates are then appropriately displayed.

Project Base Point
Shared Site:
N/S 2645173.5
E/W 2000143.2
Elev 42581.2
Angle to True North 3.873°

3.3.4 Creating superior geometry

The project is now so far prepared that you can start modelling the main components. As-Built for Revit provides with an extensive range of functions.
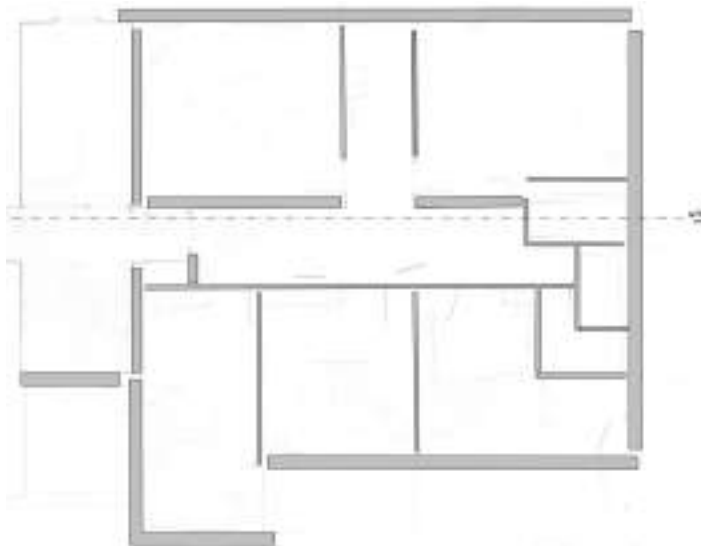
Start with the preparation of the floor plan (➔ **Object Tools - Prepare Point Cloud**) to be able later to carry out the fitting of the walls over the whole storey (➔ **Object Tools - Fit Wall**).

The command assists you in the precise placement of the walls, as well as in the creation of new wall types relative to the wall thicknesses found in the scan.

Create the appropriate walls for each storey. Gaps between the walls or non continuous wall faces are not now a problem. After creating the walls use the As-Built command → **Alignment - Align Walls**. This command allows you to establish perpendicularity, parallelism, wall intersections and continuous alignments within predefined margins of error, also simultaneously over multiple storeys.
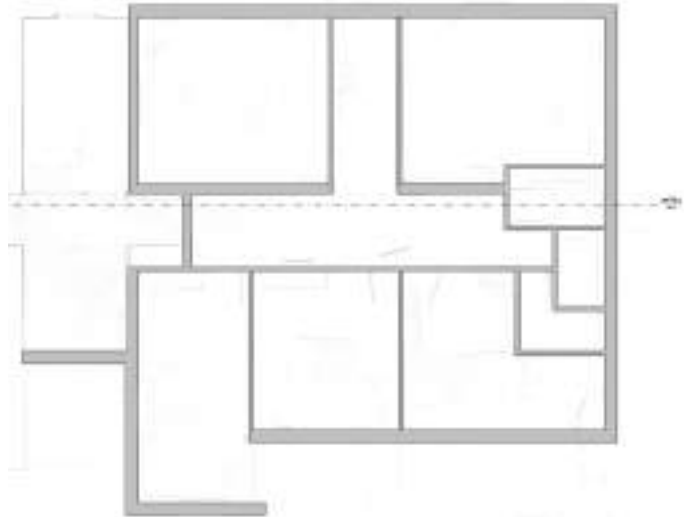
3.3.5 Create components in an aligned view

For the placement of individual components as for example doors and windows, it is recommended that you create many aligned views (framing elevations) and sections in the model. Afterwards the components can be inserted and aligned in the section views.

Should you need views that are precisely aligned orthogonal to a component (for example a wall), then framing elevations can be very useful. You need to first to create a reference plane which will then serve as the view plane. You should either use the standard Revit commands or the As-Built commands for fitting planes → **Work Plane Tools - Fit Plane (n Points)** or → **Work Plane Tools - Fit Plane (1 Points)**. Give the reference plane a name.

Now it is possible to create a framing elevation (tab View → panel Create → Drop down list View→ Framing elevation⬜), which will be precisely aligned to the named reference plane:
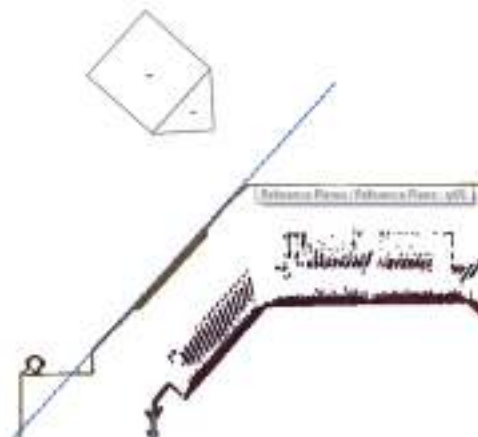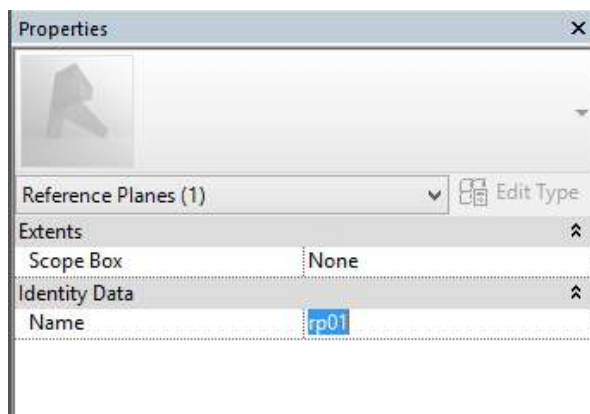
Alternatively you can also use VirtuSurv (see section **4.2**), to be able to place components such as windows, doors, beams or columns.

3.3.6 Create families and in-place families

If it is required by your project that new component families must be created, then As-Built for Revit supports you with many versatile functions.

Point cloud regions can be exported from the 3D view → **Point Cloud - Export PC Region**), so that they can then be inserted into the families editor (→ **Connect Point Cloud - Insert PC Region**). Construction aids such as model lines or construction points can be created using the VirtuSurv scan view in the families editor. Scaled ortho images (→ **Object Tools - Ortho Image**) can be inserted into the families editor to be used as a model reference (→ **Connect Point Cloud - Insert Ortho Image**).

If you have created model lines in the project environment as construction aids, then you can save them directly from the families document and process them further in the families editor (→ **Model - Lines to Family**).

4 Modelling with As-Built for Revit

Revit already offers easy ways of modelling building models from point clouds. As-Built for Revit extends these capabilities with its own powerful tools for creating and customising components and construction aids in the point cloud.

When modelling with As-Built for Revit there are two different methods available for you to use:

- The creation of components and construction aids directly in the Revit point cloud. As-Built for Revit offers you for this purpose powerful tools to fit objects (for example walls and work planes) on to the point cloud, as well as a 3D point snap in the point cloud, which is useful when using the As-Built for Revit commands. You can find a quick start in this way of working in the following chapter **4.1** .

- The creation of components and construction aids in a photo like scan view. These scan views are made available through the external software VirtuSurv, and it uses its own optimised scan data format. Working in the scan view is often far more intuitive and faster than in the 3D point cloud. An overview of how to work with VirtuSurv can be found in chapter **4.2**

When you start a project with As-Built for Revit, you don't have to choose one of these two methods. You can use them both in parallel, use the one that appears to you to be the most appropriate for the modelling task in hand. All of the functions, that are made available through VirtuSurv in the scan view, can also be used on the point cloud directly in Revit. It is often the case, however, that working in the scan view is far more intuitive and ergonomic.
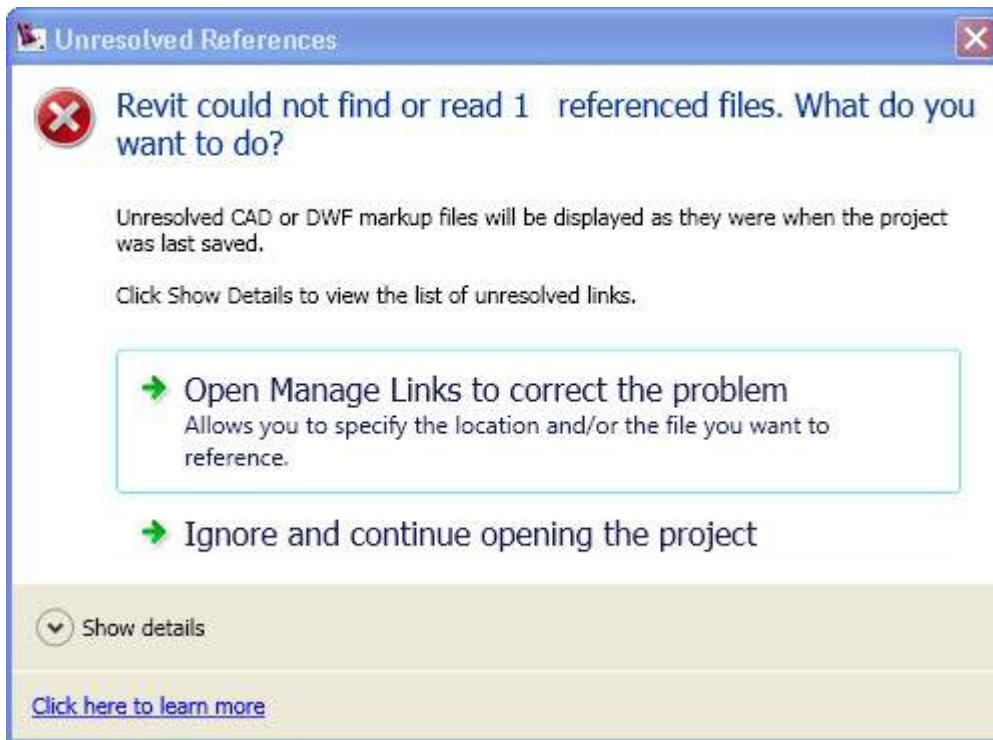
4.1 Tutorial 1 - Modelling a room in Revit

This tutorial demonstrates the typical workflow of how you can create a building model from a point cloud. You can find this tutorial on the FARO web site for download:

http://download.faro- 3d- software.com/all_ languages/Tutorials/As- Built/Tutorial_ As-BuiltRevit_1.zip

4.1.1 Preparing the Project

Unzip the ZIP archive. You will find therein the folder "Building Tutorial" with the REVIT project file (RVT) as well as the point cloud data (RCP and RCS). Open the Revit project file.

The project file already contains a reference to a point cloud. Because of the changed saved path it cannot be found and the following message appears:

Select the option "Open manage links to resolve the problem". The Revit dialogue box "Manage Links" is opened. Select the tab "Point Clouds":[1]

---

[1]Should the message not be displayed, then open the dialogue box "Manage Links" from: tab Manage → panel Manage Project → Manage Links.
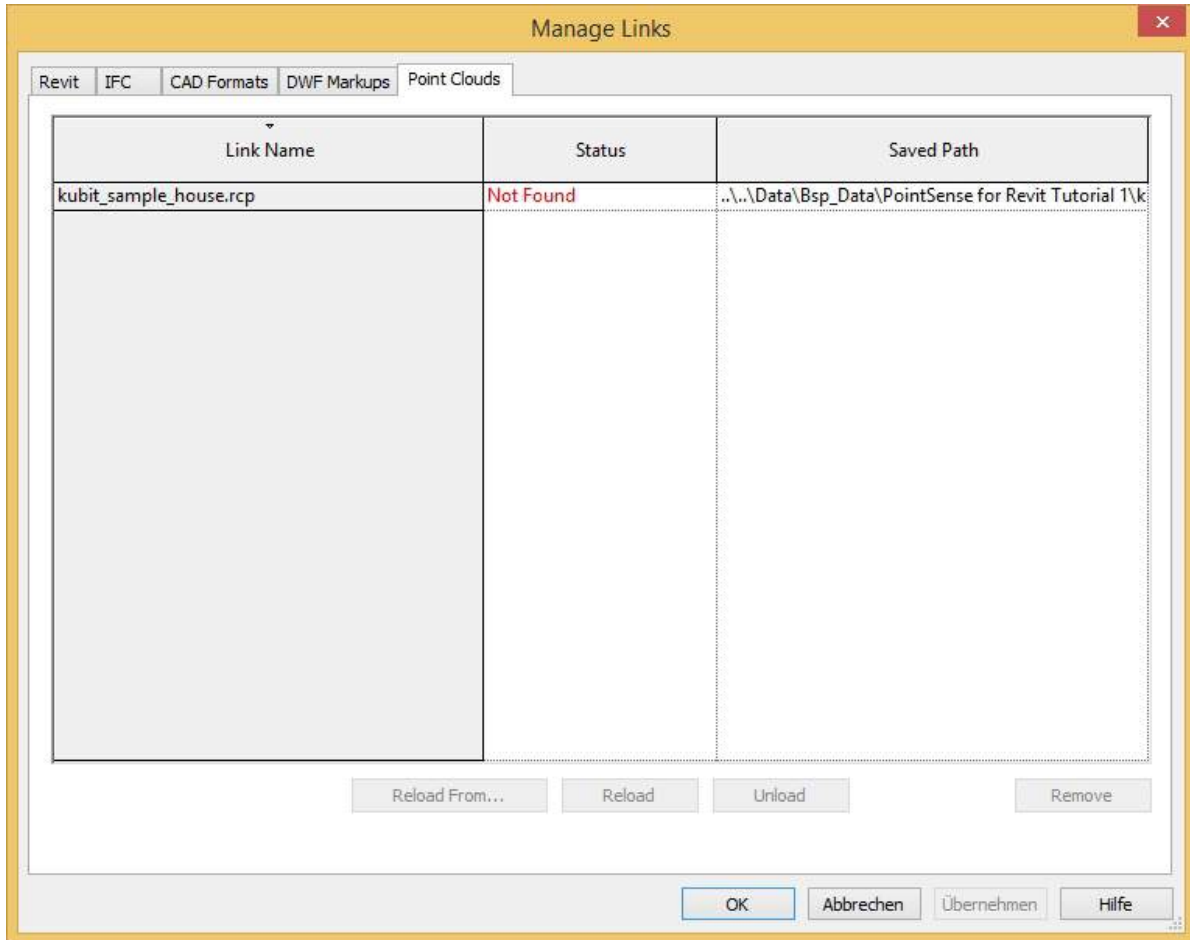
Left click on the entry "kubit_sample_house.rcp" and then click **[Reload From ...]**. Navigate to the folder containing the tutorial data and then select the point cloud file "kubit_sample_ house.rcp ". Exit the dialogue box by clicking **[OK]**. The point cloud is now correctly referenced in the project.

View the project in the default 3D view (Ribbon View → Create Group → 3D View 🏠). You

see the point cloud of a small building and some walls that have already been prepared. For the tutorial only a part of the scan data is required, therefore the loaded point cloud does not reflect the whole building.

Note: The point cloud will be displayed highlighted whenever the mouse pointer hovers over it. This can sometimes be very irritating when you are working. To prevent this happening disable the option "Select links" on the icon below the drawing window or by unchecking the box in the "Select" drop down menu on the "Modify" panel in the "Architecture" tab on the Ribbon:

Change to "Section a-a" in the Section view (Project Browser → Sections→ double click on "Section a-a"). The point cloud has already been aligned so that the floor "Level 1" (height 0 mm) has been placed on the level of the floor and already a second floor "Level 2" (height 2680 mm) has been inserted.

Now switch to the plan view of "Level 1" (Project Browser → Floor Plans→ double click on "Level 1"). Now you see the point cloud and the previously prepared walls. The visibility of all of the objects (walls and point cloud) is controlled by the view range settings. Open the

corresponding dialogue box in Revit, on the Properties palette (Properties Floor Plans → View Range → Edit):
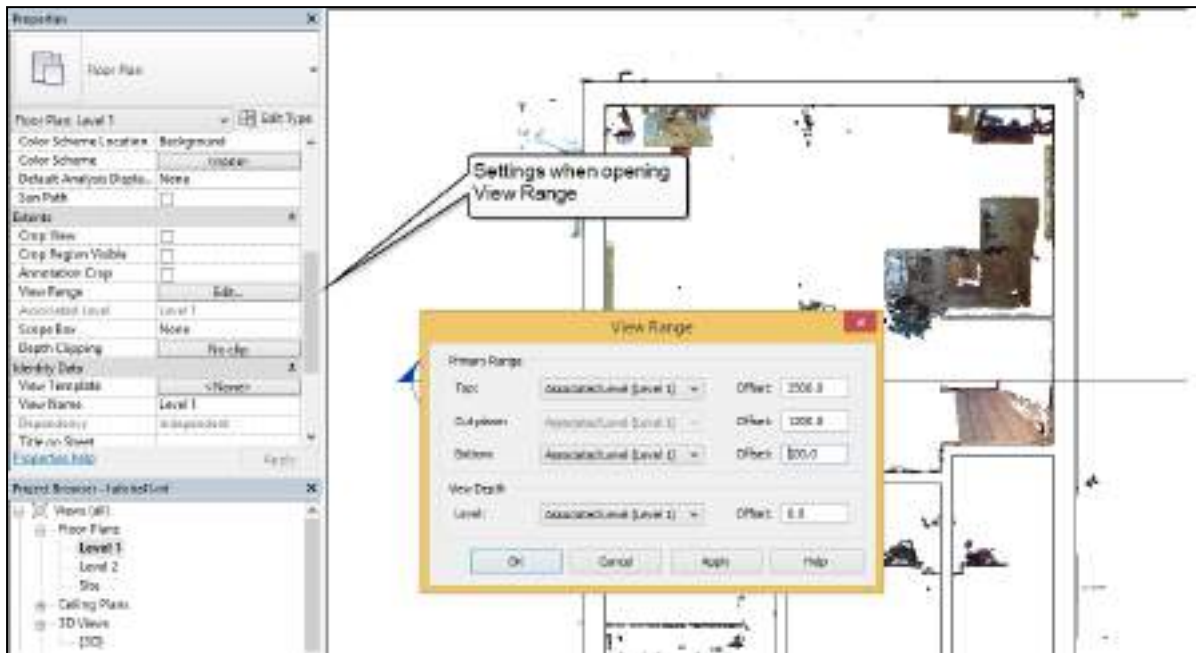

You can see that the bottom edge of the view range has been set a few millimetres above the plane "Level 1". This way you can hide points on the floor to make the floor plan more visible. Close the dialogue box by clicking on **[Cancel]**, to return to the plan view.

## 4.1.2 Preparation of the point cloud and fitting the walls

After you have become a little familiar with the project, the missing walls can now be created using As-Built for Revit. Fitting the walls to the point cloud is done in the floor plan, but before we start using this command the point cloud needs to be prepared a little.

In the preparation of the point cloud certain properties, which are needed for the fitting of the walls, will be pre-computed and displayed in a grey scale image. This pre-computation is not absolutely necessary, but it considerably speeds up fitting the walls and is therefore strongly recommended.

Change to Floor Plan "Level 1" and start the command → **Object Tools - Prepare Point Cloud** . Should the dialogue box "Switch Active View" be displayed, select the desired view and confirm the choice by clicking the button [Switch View].
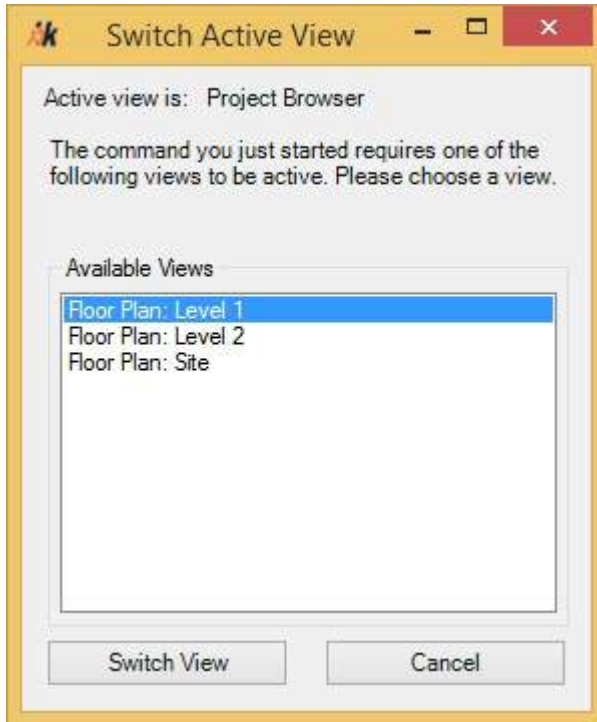
In the "Prepare Point Cloud" dialogue box click on the button **[Start]**, the pre-computation of the point cloud is started. This may take a few minutes. The progress of the computation can be followed in the status window:
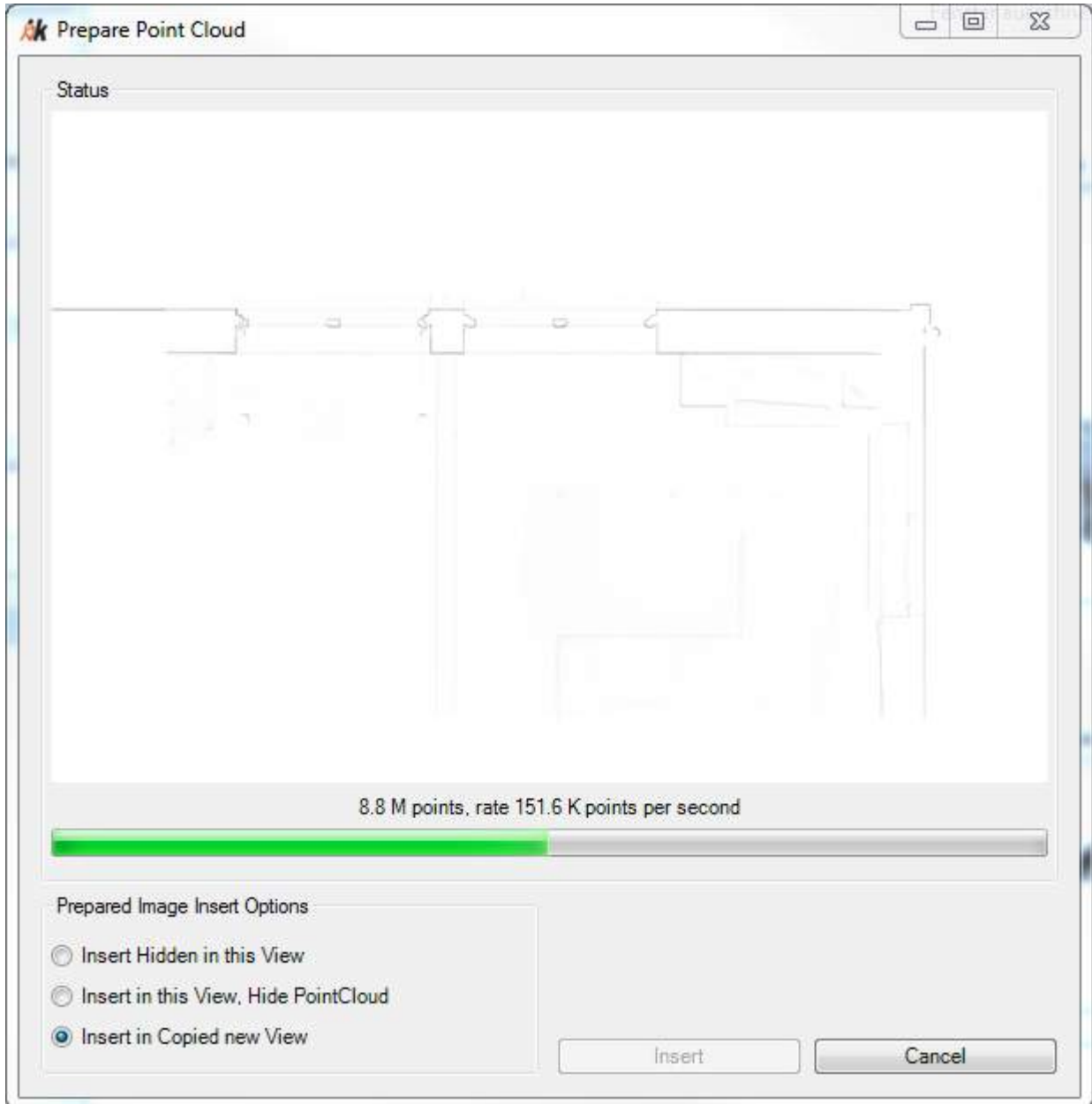
As soon as the pre-computation is completed, select the option "Insert in Copied new View", and then click on the button **[Insert]**.

You can now see that the point cloud in the Floor Plan has been replaced by the pre-computed image. The point cloud object itself is not deleted, but only hidden in the view. By selecting the option "Insert in Copied new View" As-Built for Revit it has created a copy of the Floor Plan view "Level 1" with the name "Level 1 prepared". The point cloud is still visible in the original view "Level 1".

In addition to the accelerated fitting of walls, the pre-computed image has yet another advantage in that details are significantly better to identify than in the point cloud. Zoom in even closer on to the door and you can clearly see the difference:
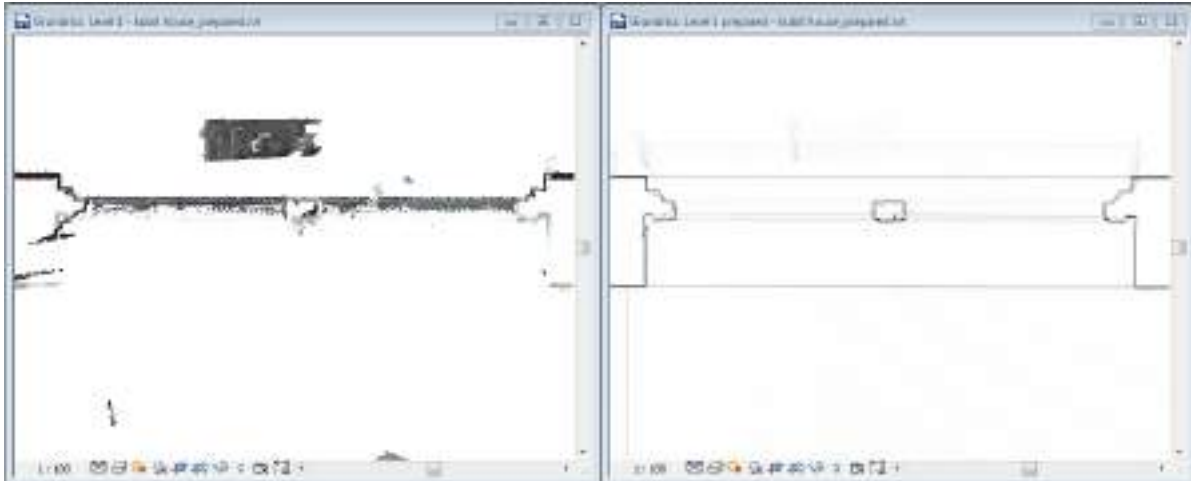
Compare the level of detail in the plan view: The sectioned point cloud (left) and the pre-computed image (right)

Note: Whenever the mouse pointer hovers over the pre-computed image, it will be displayed highlighted. This can sometimes be very irritating when you are working. To prevent this disable the option "Select links" on the icon below the drawing window or by unchecking the box in the "Select" drop down menu on the "Modify" panel in the "Architecture" tab on the Ribbon:

In the next step the missing walls will now be inserted into the floor plan. Start the command → **Object Tools - Fit Wall** , the following dialogue box appears:
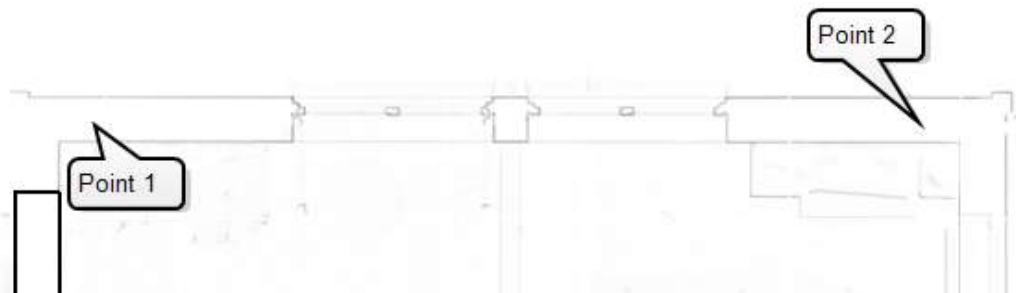

Leave the settings as they are and start fitting by using the button **[Start Wall Fitting]**. The following prompt appears in the Revit status line:

|	
`Click to enter the start point for wall fitting`	*Click the first point on the inside of the exterior wall*
`Click to enter end point for wall fitting`	*Click the second point on the inside of the exterior wall*

After the second point has been clicked, As-Built for Revit determines the precise alignment of the wall and the wall width within the point cloud. The following dialogue box appears:
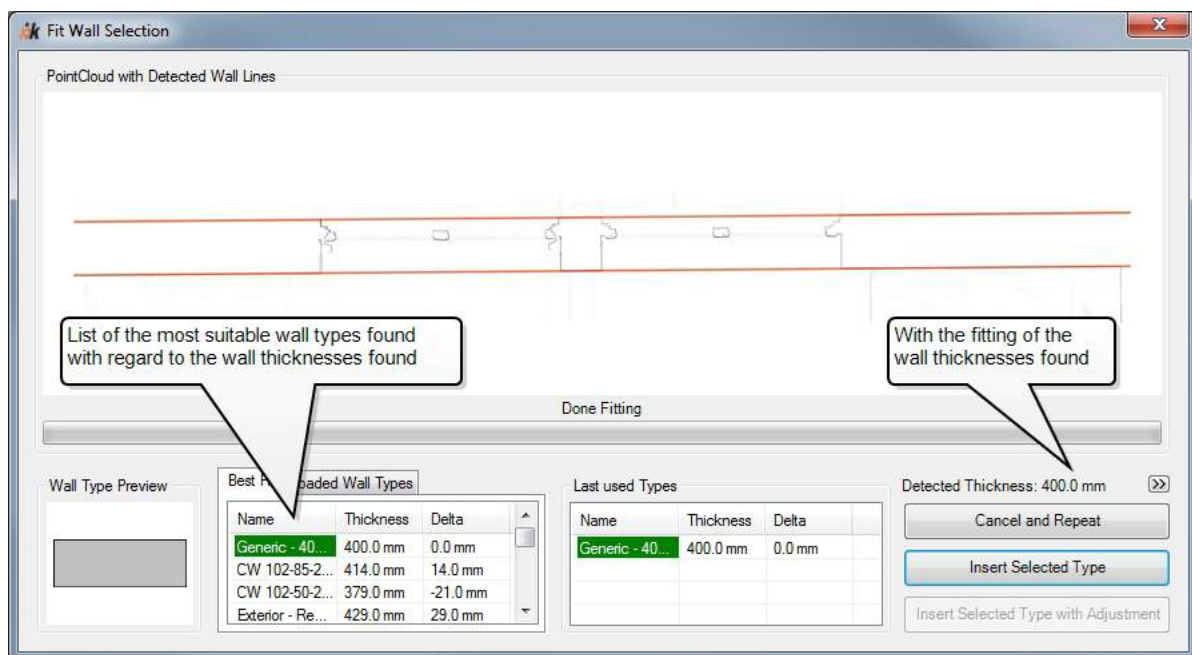

As-Built for Revit searches all of the wall types that are loaded in the project to find the most appropriate type according to the determined wall width, and offers these in a pre-selection-list. You can see that the fitting process has recognised a wall with a thickness of 400.0 mm, so As-Built for Revit has suggested the wall type "Generic 400.0 mm". Insert the selected wall type into your project by clicking the button **[Insert Selected Type]**. You can see that the wall has been inserted in the correct position in the project. Proceed with the next wall without exiting the command. The prompt is displayed again in the status bar:

Click to enter the start point for fitting the wall

*Click the first point on the inside of the next exterior wall*

Click to enter the end point for fitting the wall

Click the second point on the inside of the exterior wall a short way before the partition wall to the stairs


Point 1

Point 2
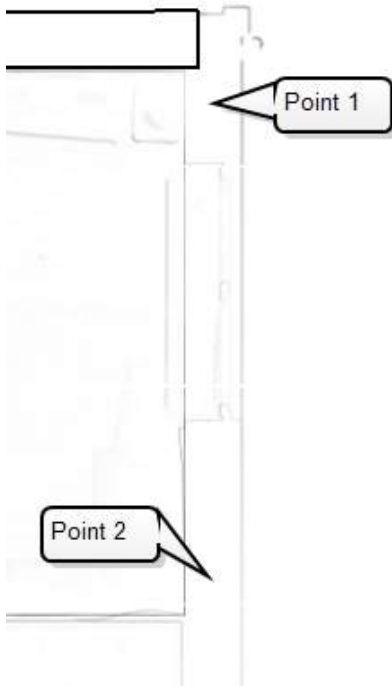
Purposely divide the exterior wall into two different wall sections, right and left of the partition wall to the stairs. The wall thickness in the area of the stairs changes by a few millimetres. In the course of this tutorial you will see how you can still create a continuous wall face on the outer face of the wall.

The dialogue box is once again displayed with the preview of the wall as well as the pre-selected wall type. In this case a wall thickness of 395 mm has been found. You can now decide if you want to use the pre-selected wall type "Generic 400.0 mm", or you can create a new wall type with a wall thickness equal to that found. To do this click on the button **[Insert Selected Type with Adjustment]**. The wall with the newly created wall type will be inserted into the project.
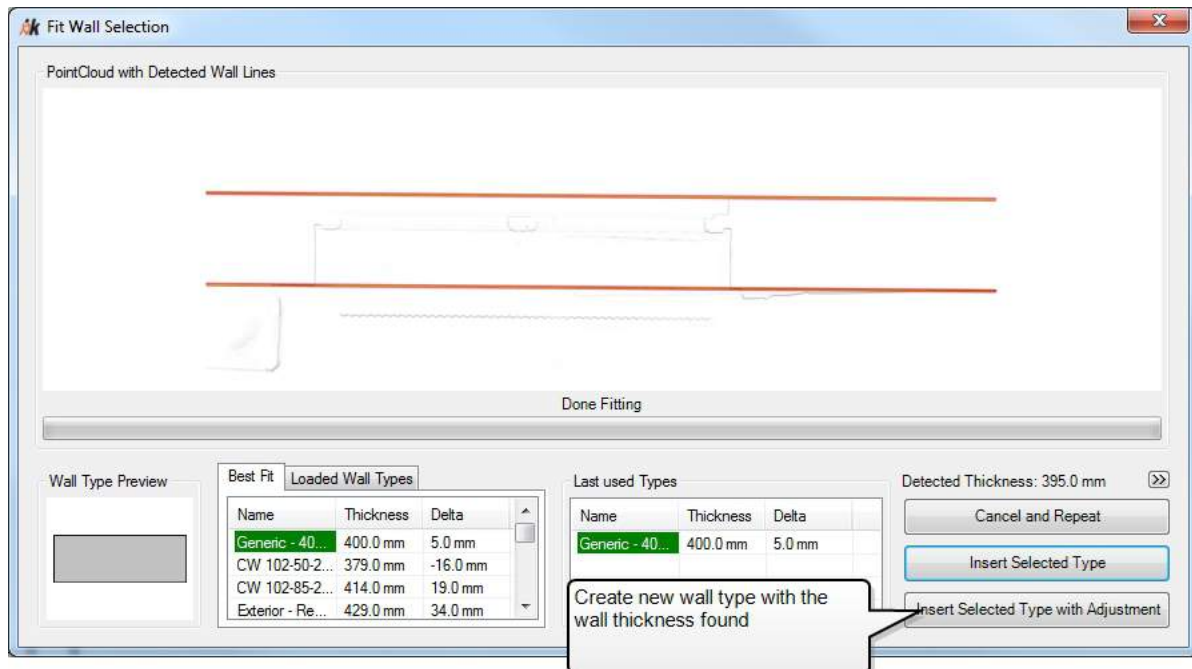
You will notice that As-Built for Revit has always rounded the wall thickness to a 5.0 mm accuracy. Naturally this value can be adjusted so that wall thicknesses can also be recognised with an accuracy of down to one millimetre. For more information, see the Command Reference for the command → **Object Tools - Fit Wall**.
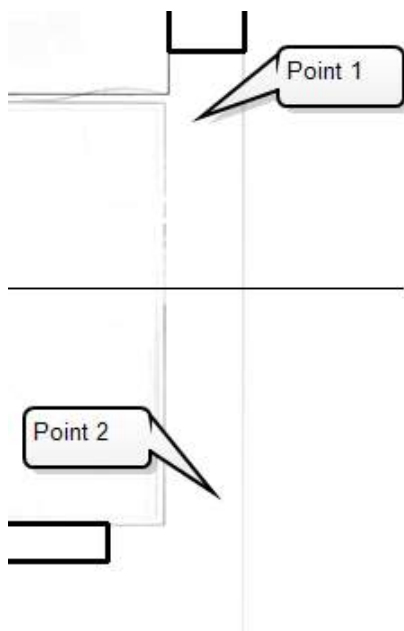

Now create the final piece of the exterior wall from the partition wall to the left end of the stairs. The detected wall thickness is 420 mm. From the list of loaded wall types As-Built for Revit suggests the type "Generic - 400.0 mm" as the most appropriate type, due to the previous wall thickness, because there is no wall type with a thickness of 420 mm. Therefore, we will again create a new wall type. Leave the pre-selection with "Generic - 400.0 mm", and insert the wall into the project using the button **[Insert Selected Type with Adjustment]**. A new wall type will be created with the correct wall thickness.

Now you can create, in just the same way as before, the partition wall to the stairs, and here also you can create a new wall type based on a different generic wall type. Exit the command with **[Esc]**. Your results should look like this:
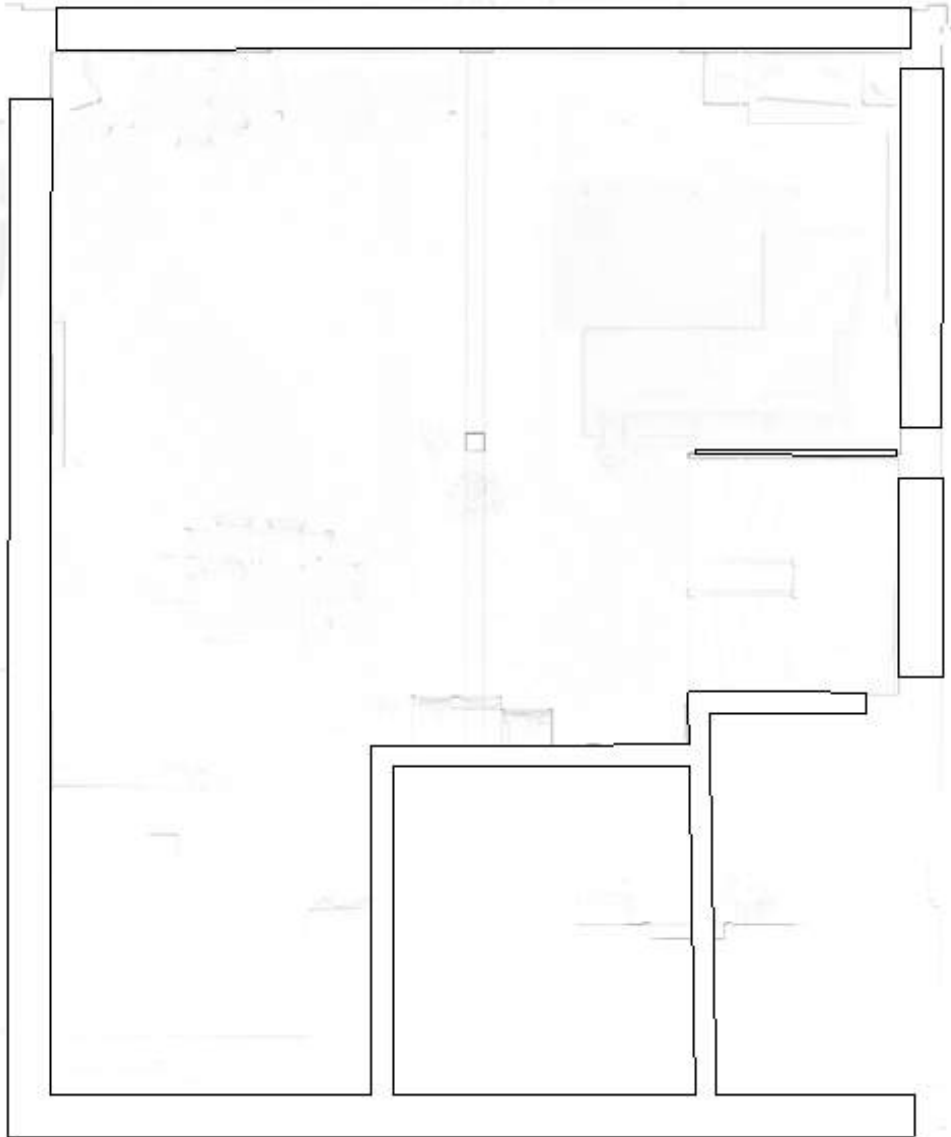
4.1.3 Automated alignment of walls

When fitting walls As-Built for Revit determines the best possible fit of the wall direction in the point cloud and at the same time fits the wall thickness. You can see that the walls do not stand exactly at an angle of 90° angles to each other. You can easily check this by inserting angular dimension between the walls (Ribbon, Annotate tab → Dimension panel → Angular △):
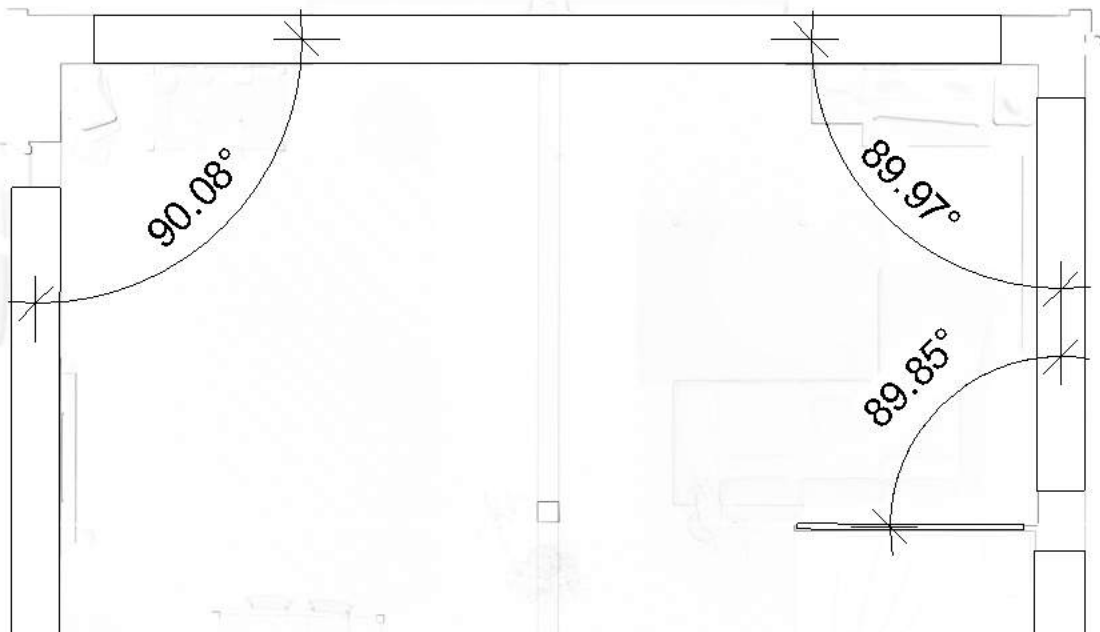
Depending which accuracy requirements have been placed on your model, it may be necessary, to adjust the walls appropriately, so that the walls are at 90° to each other and that they are also continuous and axially aligned. For this task As-Built for Revit provides you with a powerful tool with which you always keep the control over the accuracy to be achieved.

Launch the command → **Alignment - Align Walls** 🗔. The following prompt appears in the status line:

`Select multiple walls to align and apply orthogonality`	*Drag a selection window over all of the walls, and confirm the selection in the Revit Options Bar with* ***[Finish]***

The "Align walls" dialogue box is displayed in which you can adjust the parameters for the orientation of the walls. In this tutorial there will only be shown a fraction of the many possible settings. A detailed description can be found in the Command Reference → **Alignment - Align Walls**.

In the preview window you can see the selected walls displayed in colour. In the area "Display Colour Options" you can choose which properties should be displayed in which colours. Select here the option "Show reference systems":
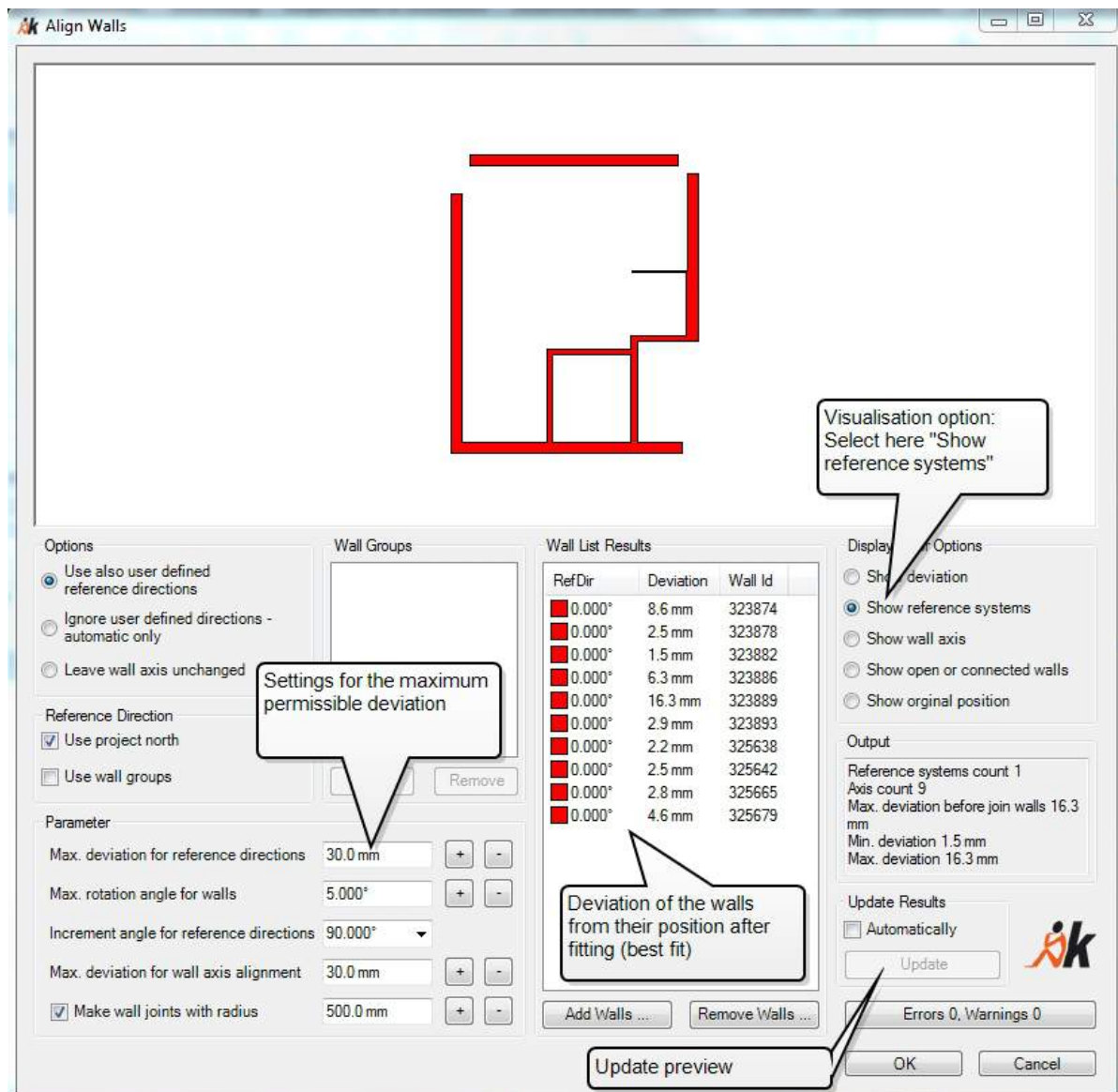
The colours in the preview window switch uniformly to red. Change the parameter "Max. deviation for reference directions" from 30.0 mm to 15.0 mm (this can be done quickest by clicking on the **[-]** button next to the text box), and then clicking the **[Update]** button in the "Update Results" area. You can see that a wall was recoloured.

Each of the colours represents a different reference system, in which the walls stand at a defined angle to each other (in the current example it is 90°). You can check the deviation for each wall in the list "Wall List Results", to see how they fit in the reference system after fitting the wall axes. In the current example, the wall with the ID 323874 was moved by about 8 mm, so that it is orthogonal to the reference system that has been found. If you click on an entry in the list, the corresponding wall will be highlighted in the preview window. With a double-click you can zoom in to the wall.

In the "Parameter" area you can set the values for the maximum allowable deviation of the walls. The value "Max. deviation for reference directions" is the value of the deviation that is allowed so that a wall can still be assigned to a particular reference system.

Change the value for "Max. deviation for reference directions" back to 30.0 mm again (this can be done quickest by clicking on the **[+]** button next to the text box) and click again on **[Update]**.
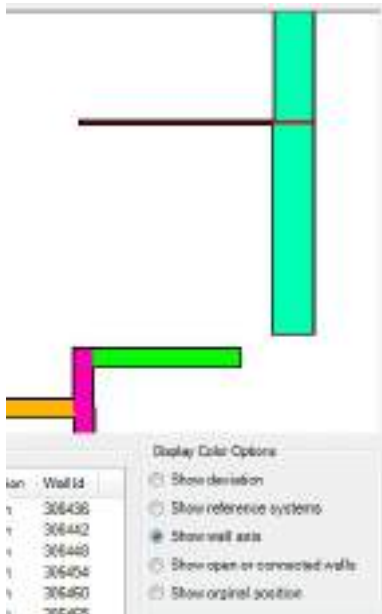
Change the visualisation option to "Show wall axis" and in the preview window zoom in on the walls in the staircase (to do this click in the preview window and scroll using your mouse wheel). Both of the walls to the left and right of the partition wall will be so moved that there will be a continuous axial alignment on the outer face of the wall. This common wall alignment will be signalled by the walls having the same colour.

Before you confirm the wall fitting, the radius for wall intersections should now be increased, so that the existing gaps can be closed. Here you need to change the value for "Make wall joints with radius" to 1000.0 mm (use the **[+]**) button again, and then click on **[Update]**. Exit the dialogue box by clicking **[OK]**.

Using the previously inserted angular dimension you can see that the command has fitted the walls so that they now stand at an angle of exactly 90° to each other. The existing gaps between the walls were also closed and at the stairs a continuous exterior wall now exists:
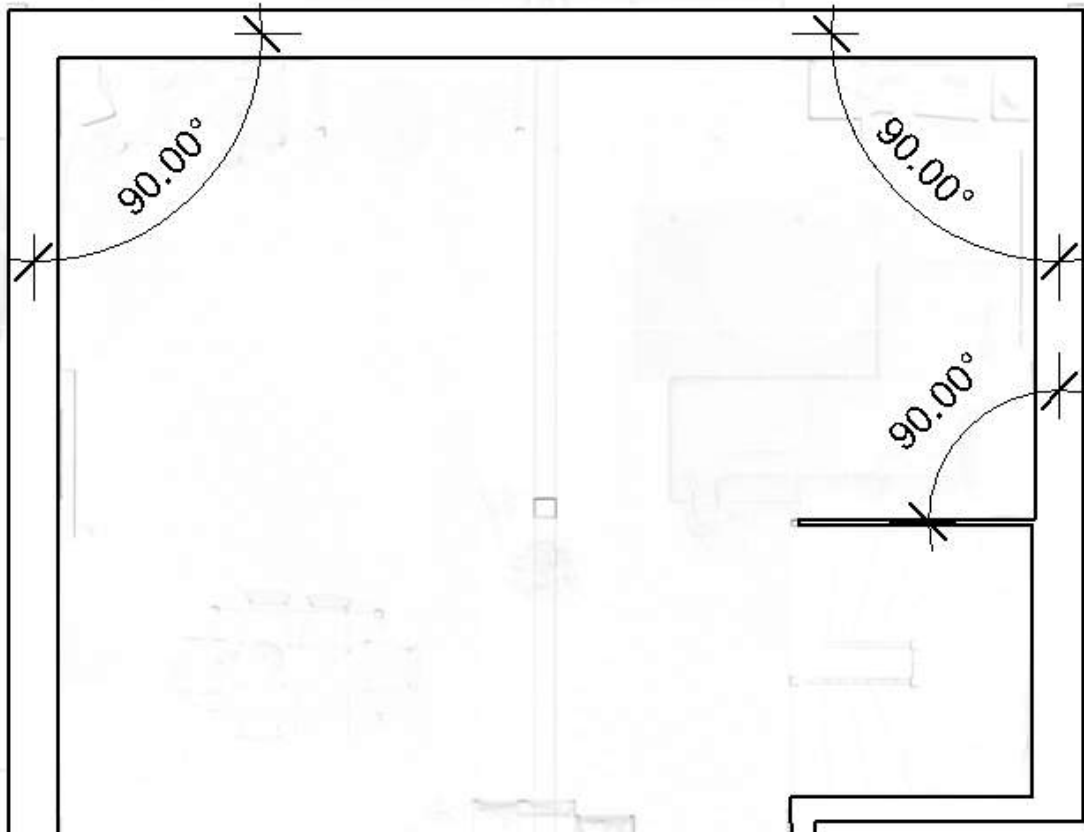
4.1.4 Insert windows and doors

In the next step a window and an exterior door will be added to the model. First change to the 3D view (Ribbon, View tab → Create panel → 3D View 🔲 or Project Browser → 3D Views →double click on **{3D}**).
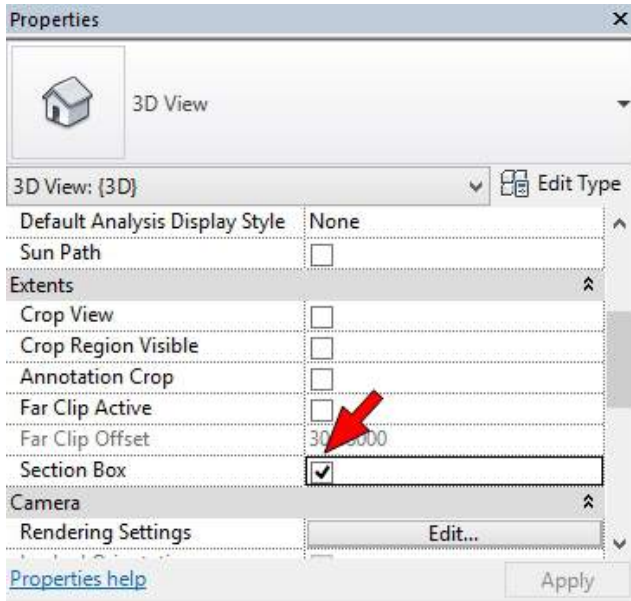
Activate the 3D section box for the current view on the Properties palette. In the drawing window a box appears around the model (zoom out if the box is not visible). When you select it (left click with the left mouse button) there appears handles on all six surfaces. Using these handles you can pull and push the section box so that the region shown below still remains visible. If necessary, change the perspective of the model using the Revit View Cube.
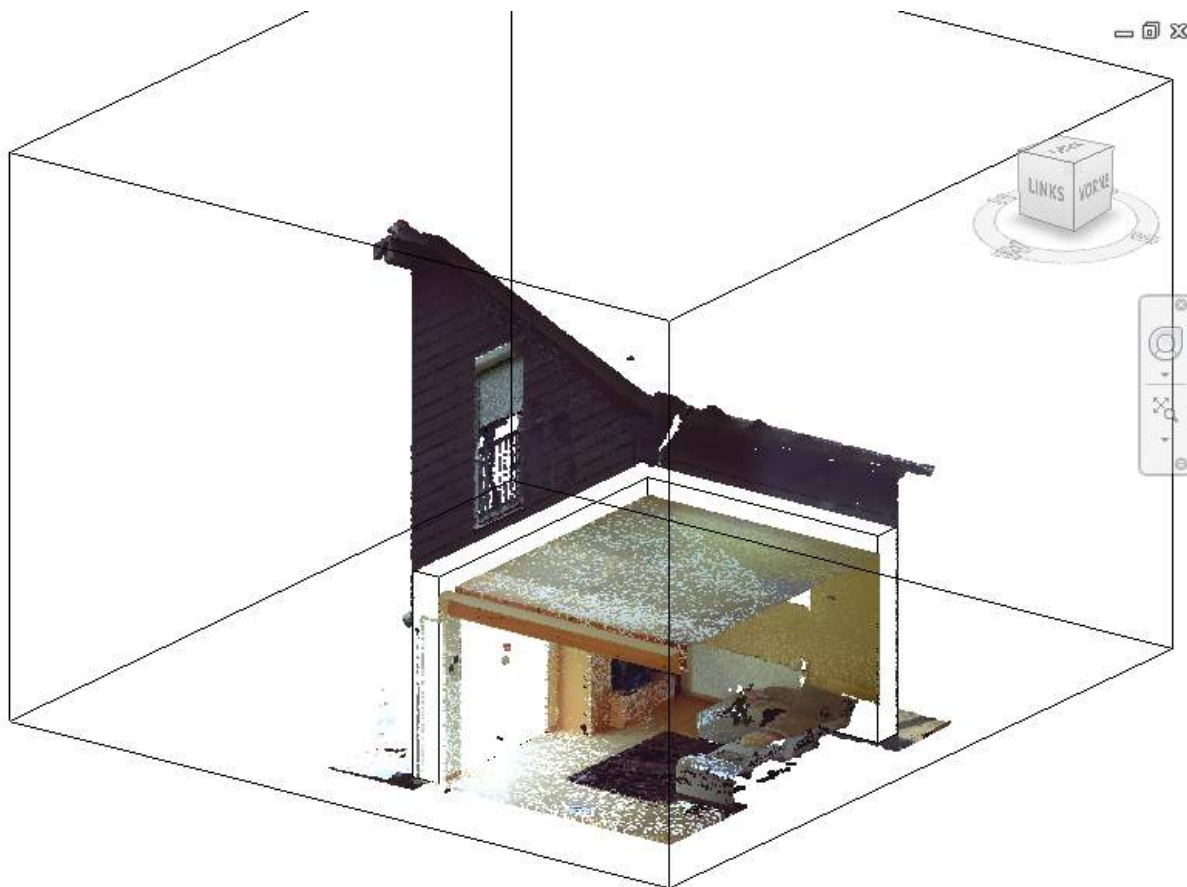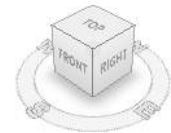
The current display style that has been selected for the 3D View is "Hidden Line", thereby the walls hide the point cloud in the region of the windows and doors. Change the Graphic Display Option so that the walls are displayed semi-transparent. To do this open the options dialogue box using the button at the bottom edge of the drawing window and set the transparency value to 50:
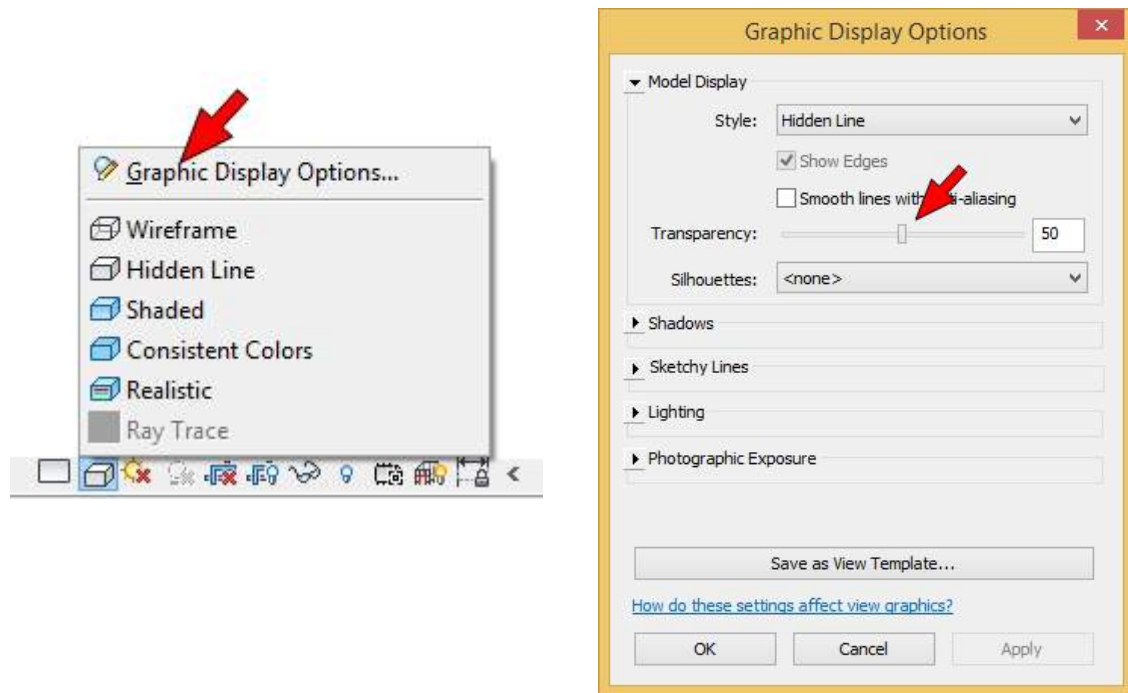
Now the walls are clearly visible and also the doors and windows in the point cloud.

Start by creating a window. Select in the 3D View above the ViewCube the view from the left and zoom in onto the window. Now start the command → **VirtuSurv Tools - Window** .
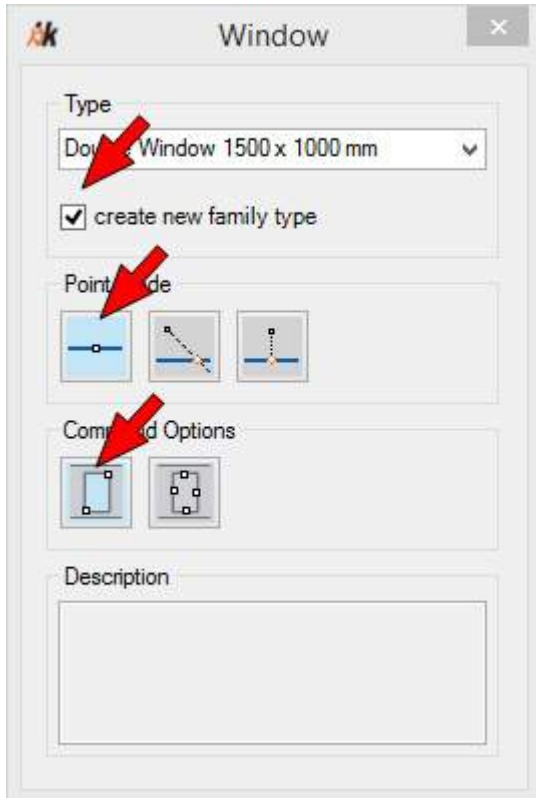
The VirtuSurv Options window appears.

Select from the Type dropdown list of the window family "Double Window 1500 x 1000 mm", as Command Option "Using diagonals" 🗔 and as Point Mode "Standard" ⊢⊣.

As we do not yet know the height and width of the window, As-Built for Revit should create a new type for the selected family with the precise dimensions. You need to activate the option "create new family type". Now click a point in the point cloud (without closing the dialogue box) in the lower left-hand corner of the window and another in the upper right-hand corner. Should an appropriate point be found in the point cloud a protocol point symbol is displayed in the form of a small cross.

Note: For this function As-Built for Revit uses its own 3D point snap. This works regardless of whether in Revit the point cloud snap (Ribbon, Manage tab, → Settings panel → Snaps 🧲) is activated or not. The Revit point cloud snap works only on the active work plane. The As-Built for Revit 3D point snap provides precise 3D coordinates from the point cloud. When you are working with the As-Built for Revit tools you can disable the Revit point cloud snap. If the Revit point cloud snap is enabled it can lead to image display delays.

After you have clicked the second point in the point cloud, the window is inserted into the wall. As-Built for Revit has created a new type for the window family, with the precise dimensions from the point cloud. Exit the command with [Esc].

Note: A rounding value (in the current example 5 mm) has been used for the opening height and width, just as for the wall thickness. This value can be set using the command → **As-Built - Settings** .

To finish this tutorial a door will be inserted into the exterior wall. Start the command → **VirtuSurv Tools - Door** . The VirtuSurv Options window appears.
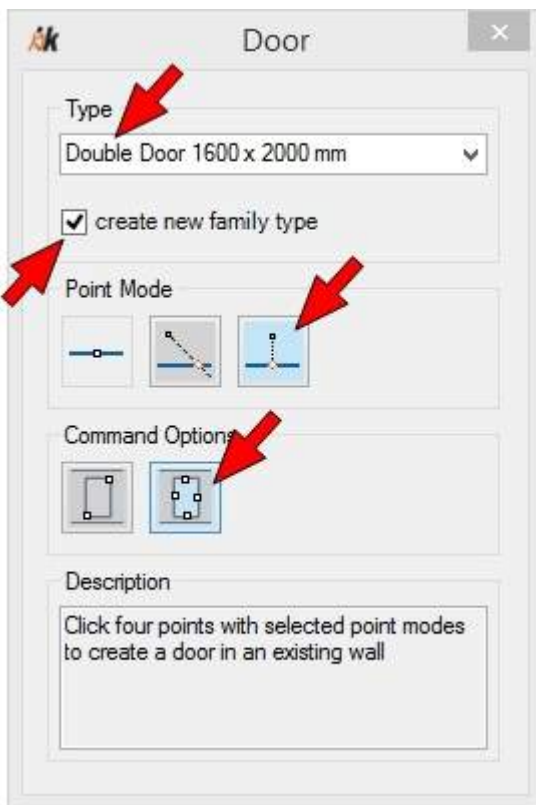
Select from the Type dropdown list of the door family "Double Door 1600 x 2000 mm" and activate the option "create new family type". Because of the skirting boards, we cannot precisely pick the diagonal points for the door in the point cloud. Instead select as Command Option "Using reveals" . In this case there are only the two point modes available "Intersection with the work plane" and "Plumb foot point" .

The option "Using reveals" expects that four points are picked in the reveals of the opening or on the edges. From the point modes it is ensured that the picked points are plumbed onto the wall surface or intersected with them. For more information on using the point modes, see Section **4.2.3** .

Rotate the 3D View, so that you can see the floor and the right hand reveal in the point cloud (to do that hold down the [Shift] key and at the same time the middle mouse button. Now you can move the mouse to rotate the view). If necessary, adjust the section box so that the door is easy to see.

Select the Point Mode "Plumb foot point" and click on a point on the floor in the doorway and on a second point in the right-hand reveal:
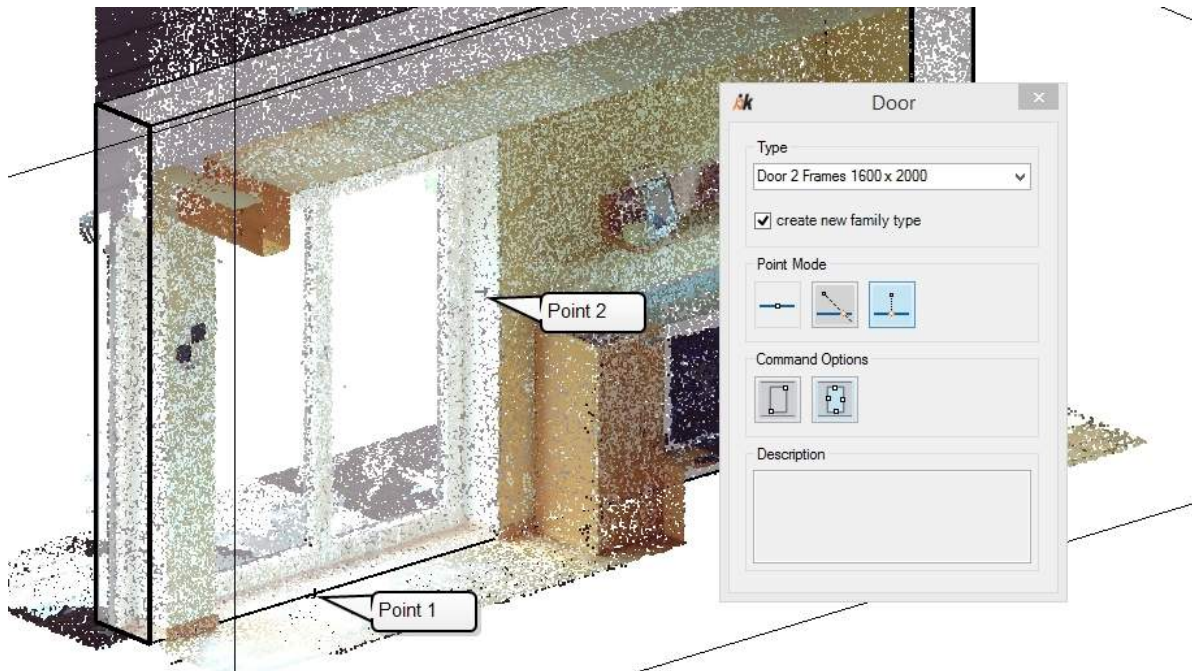
Rotate the view so that you can now see the left-hand reveal and the door lintel and pick a point in the reveal and then a point on the underside of the door lintel.
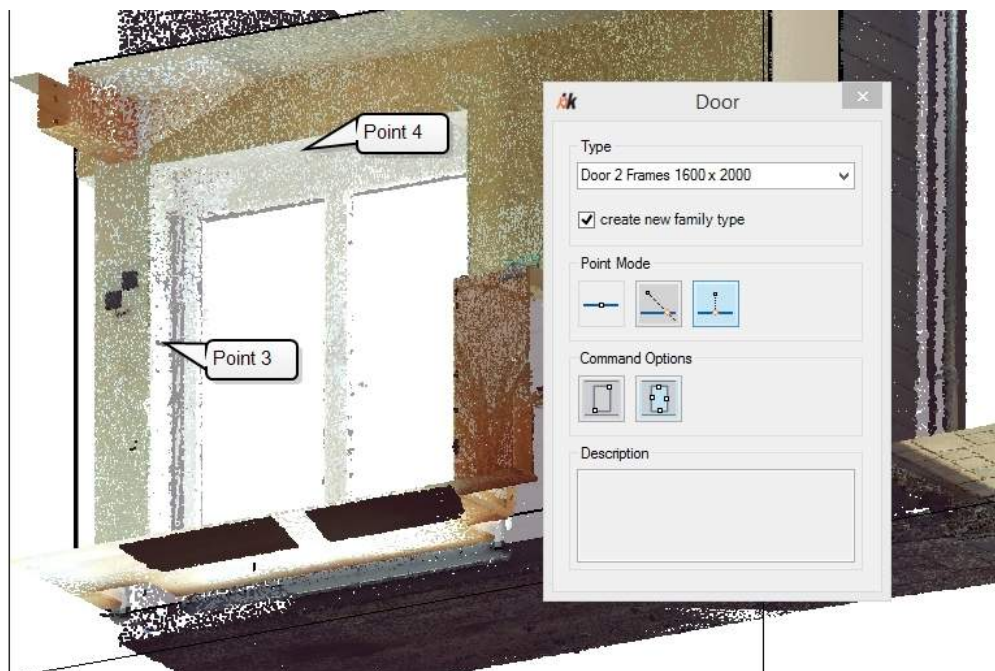
The door will be inserted. Exit the command with **[Esc]**. Should the door be inserted with the wrong orientation (with the roller blind housing on the inside), then change to the view "Level 1 prepared", select the door and click on the blue alignment arrows:
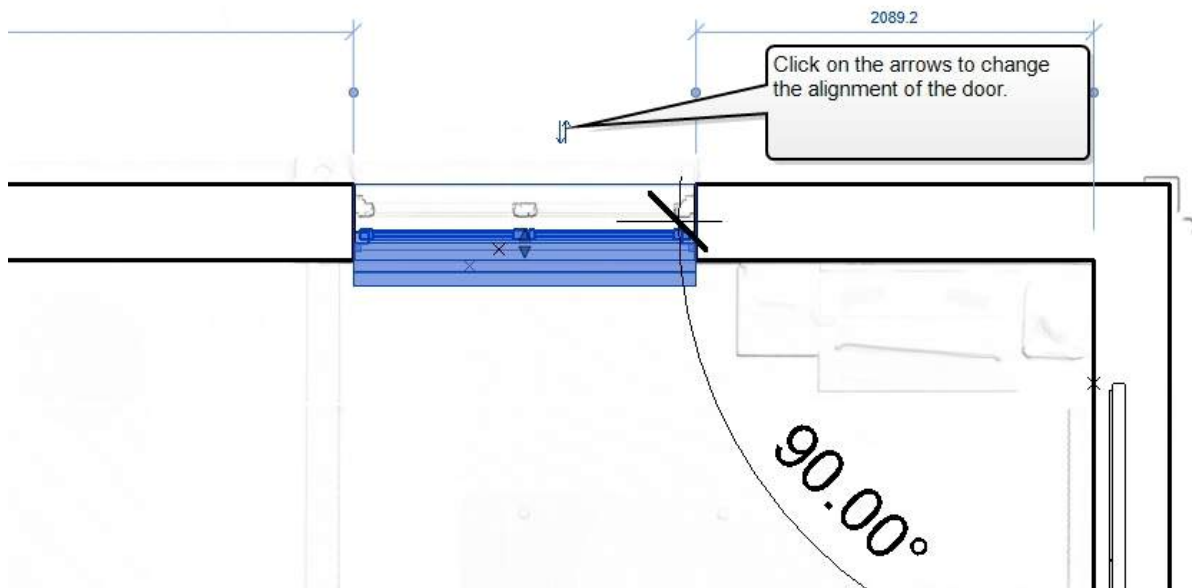
Change to the 3D view and disable the section box. Your model should now look like the following figure (in the Graphic Display Options change the Transparency value back to 0). To temporarily hide the point cloud in the view, you can use the command → **View - Hide or Show PC** .
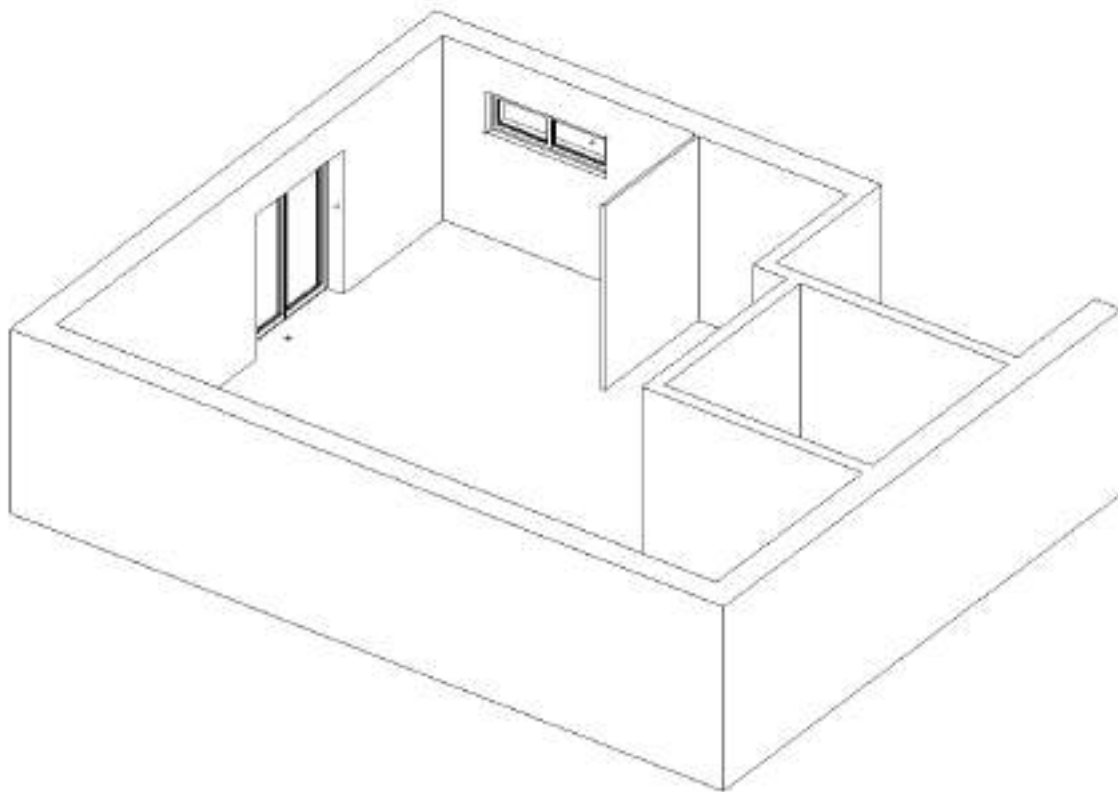
In this tutorial you learned, how to prepare a point cloud for fitting the walls and then creating the walls relative to the point cloud points. You have also become acquainted with how to use the VirtuSurv Tools in the Revit point cloud. The commands in the "VirtuSurv Tools" panel are primarily intended for working with VirtuSurv. Thanks to the 3D point snaps of As-Built for Revit you can also use the commands in the Revit point cloud. You will learn how to create a model with the aid of the Scan view in VirtuSurv in the next chapter.

4.2 Working with scan views in VirtuSurv

To be able to work with scan views As-Built for Revit uses the external program VirtuSurv. VirtuSurv provides a special tool palette for processing in Revit. Thereby, the Revit user can model particularly comfortably, both in the Revit project environment as well as in the Revit families editor. You may run the modelling commands directly from VirtuSurv and do not have to switch between VirtuSurv and Revit. Besides that, the connection to Revit is established automatically.

Note:

> Users of FARO SCENE can alternatively use the SCENE SendToRevit app. The app provides the same functions for working in SCENE Scan views (Planar View, Quick View and 3D View of a single scan) as the Revit command bar in VirtuSurv. The biggest advantage of working with FARO SCENE is that an additional data conversion to VirtuSurv is not required.

In the following, the special features of the Revit interface from VirtuSurv will be explained in more detail. For the basics of working with VirtuSurv please see the VirtuSurv manual. An English version of the manual can be opened directly from the user interface of VirtuSurv (Menu "Help" → "Manual" ⬚)

4.2.1 The Revit Command Bar

To send to Revit open the command bar and call the command "Send to Revit" ⬚ from the "Measure Mode" menu. The connection to Revit is established at the same time.

The Revit tool-bar is opened on the right hand edge of the VirtuSurv window. You can freely move the tool-bar around and dock it to another edge. In order to do so, click and hold down the left mouse button on the head of the bar whilst moving the bar to the desired position. The command bar may be closed by clicking the cross at the top right or by clicking on the ⬚ button in the tool-bar.

With the majority of the Revit commands you can select additional command options. As soon as you start such a command (e.g. "Wall" ⬚) a dialogue box opens with the applicable

command options. The options dialogue box is context sensitive, so that according to the command being used various options and choices will be offered. The dialogue box is always shown in the screen foreground and can be freely moved around:
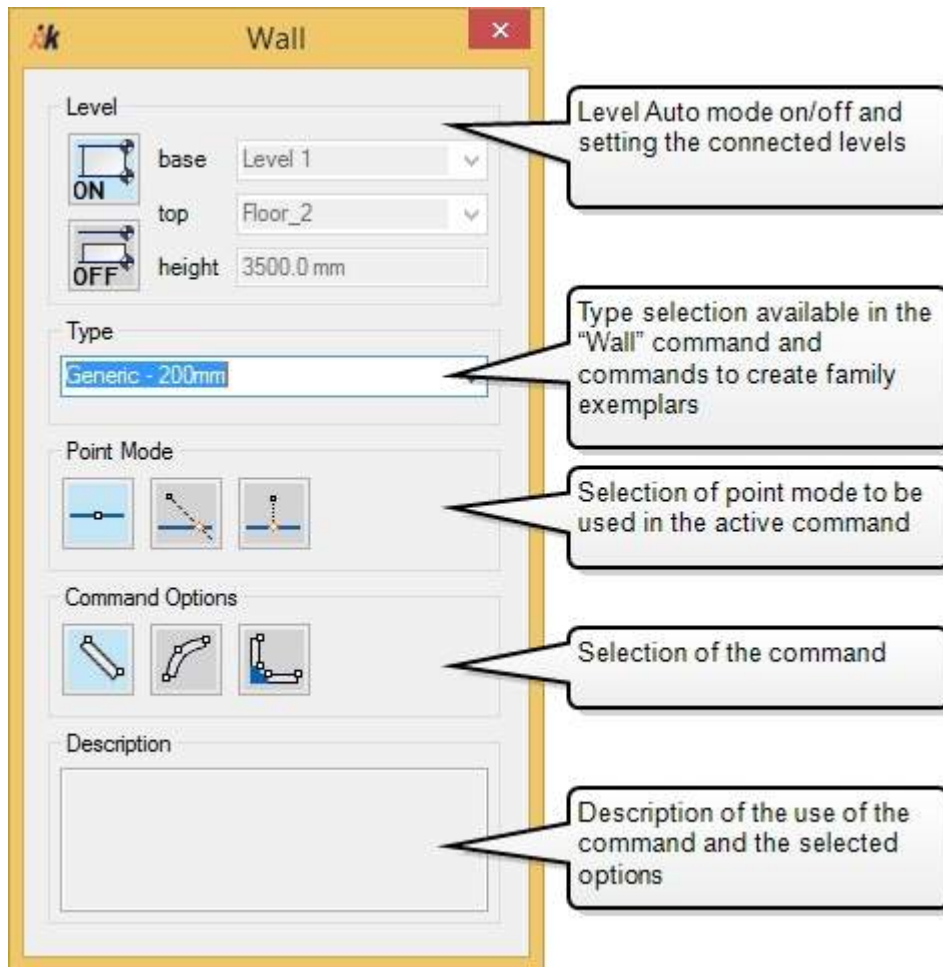
You can also call the majority of the commands in the VirtuSurv command bar in Revit from the ribbon, "As-Built" tab, "VirtuSurv Tools" panel. Therefore is irrelevant whether you start the command in Revit or in VirtuSurv. The commands, that can also be started from Revit, allow you to either click on points in the scan view or directly in the Revit point cloud. For this purpose As-Built for Revit makes available a 3D point snap. When one of these VirtuSurv commands is active, you can also switch between clicking in the scan view and clicking in the point cloud.

4.2.2 Level auto mode

The automatic level function is activated or deactivated with the "Auto Mode on/off" switch. This function influences how walls and family instances will be created with the functions → **VirtuSurv Tools - Wall** and → **VirtuSurv Tools - Column**.

If the automatic level function is activated the components will be connected on the base and upper constraints with both of the levels that are nearest to the points clicked in the scan or point cloud. The following sketch illustrates how it works.
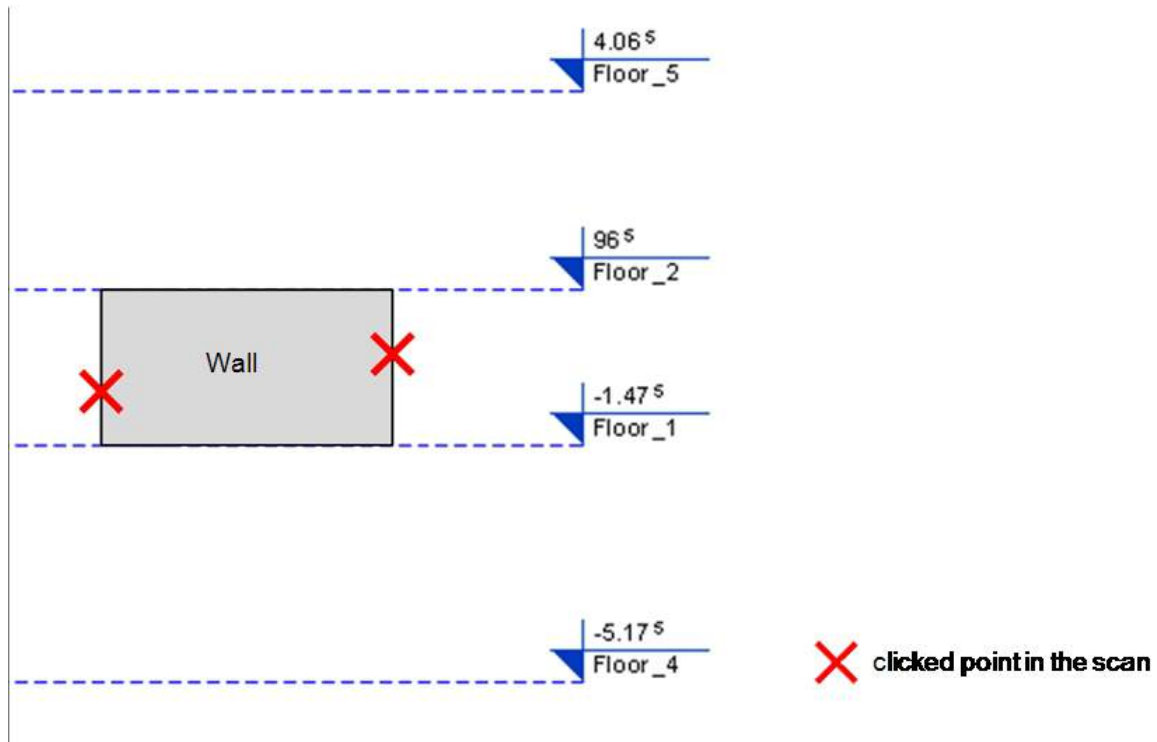
When the automatic level function is switched off As-Built for Revit uses the settings of the selection fields for the base level and the top constraint.

Selection field	Operation
base	Select the level for the lower component restraint. When <active view> is selected, the lower edge of the component will be set to the level of the current active view. Attention: In this situation a floor plan view must be active in Revit for the creation of walls or columns.
Top	Select the level for the upper component restraint. When <unconnected> is selected the upper edge of the component will not be connected, instead the given height will be used.
Height	Enter the height. This value is only used, when <unconnected> has been selected for the upper component restraint.

4.2.3 Using the point mode

Icon	Point mode	Operation
	Standard	The 3D coordinates of the selected point are used.
	Plumb foot point	The point that has been clicked on in the scan will be projected perpendicular onto active Revit work plane.
	Intersection with the work plane	The intersection point is established between the active Revit work plane and the line of sight between the scanner position and the point that has been clicked in the scan.

When using the point modes "Plumb foot point" and "Intersection point with work plane", the Revit add-in converts the coordinates of the clicked points onto Revit's active work plane according to the selected mode. You can thus, for the construction of Revit elements determine relevant points with significantly more accuracy, than by simply clicking on scanned points. If you use the point modes together with the opening commands (opening, door or window), the conversion refers to the plane of the wall side (and curved surfaces on a curved wall) closest to the scanner position. The following sketch illustrates the usage of the point modes when creating openings:
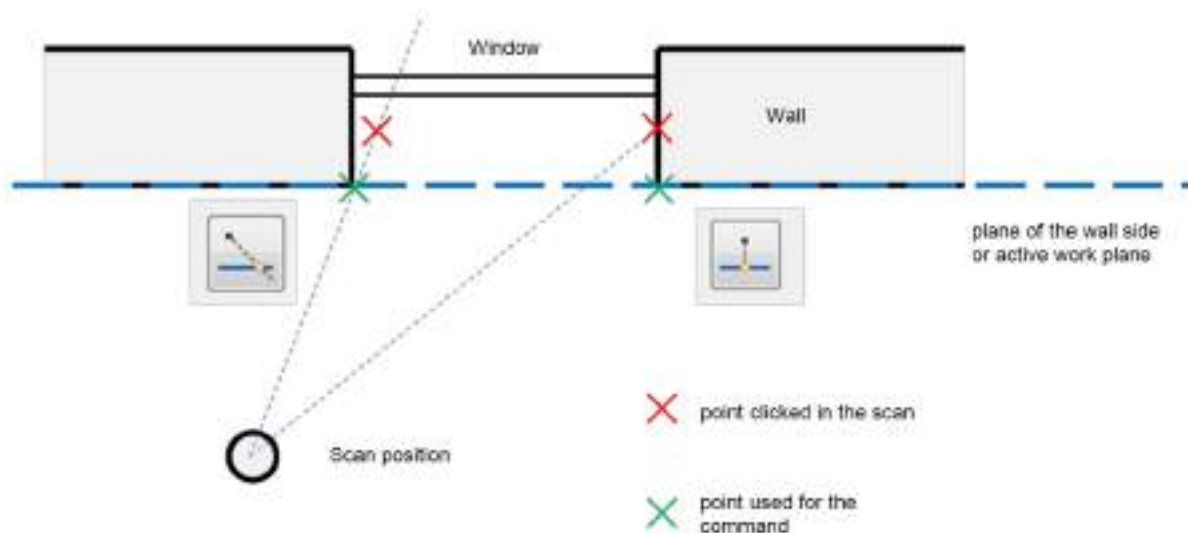
# 4.3 Tutorial 2 - Modelling a room with VirtuSurv

This tutorial demonstrates the typical workflow of how, with the assistance of the photo like scan view of VirtuSurv, you can create a building model. You can find this tutorial on the FARO web site for download:

http://download.faro-3d-software.com/VirtuSurv/Examples/index.php

Download the file EXAMPLE02_KUBIT_HOUSE.ZIP and unzip the archive.

4.3.1 Preparing the Project

Start VirtuSurv and open the project *Example02_kubit_house*. Make certain that the unit of measurement millimetres is selected. You can set this in VirtuSurv using the menu option "Configure measurement unit" 🐝.
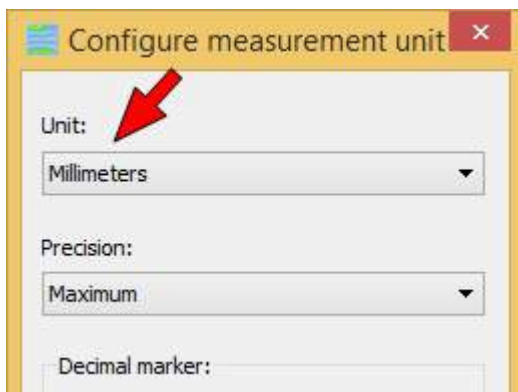
In the project *Example02_kubit_house* open the true colour views of scans 3 and 5 by double-clicking them. By selecting it in the structure view of the Project Explorer you must ensure scan 3 is at the front at first. With the mouse wheel zoom out, so that preferably the complete living room is visible in the scan window (scan_003).

Start Revit and open a new project with the "FARO_templates.rte" template. You can find this in the installations folder of As-Built for Revit in the sub folder Revit20XX\Templates (e.g.: C:\Program Files\FARO\As-Built for Revit v2018.0.0.27000\Revit2017\Templates).

In VirtuSurv open the tool-bar by selecting "Send to Revit" 🖈 from the "Measure mode" menu.

First the VirtuSurv coordinate system should be adapted to the Revit project coordinates. Select the "Project coordinates" command 🖾 and then in the options dialogue box "Project origin" 🖾. Then pick a point on the floor in scan 3. This will be the origin of the Revit project.

Confirm the message that appears. In the next step the orientation of the Revit project along the x-axis is defined. Select in the options dialogue box the "Project orientation" command 🔄 and pick two points in scan 3 along the outer wall on the left. Again confirm the Revit message

that appears. The project coordinates are now set and you can start creating the first Revit element. Select the "Level" command [icon] then pick a point on the ceiling in scan 3.

Switch to Revit and zoom to the whole project in the "East" view ("Zoom extents" [icon]). You will

see the default level "Level 1" with the elevation 0.0 and the new created level "Floor_2". By setting the project origin to the elevation of the floor the height of the floor now corresponds to the "Level 1" with the height 0.0. Together with the level "Floor_2" the corresponding floor plan view is created.

4.3.2 Wall creation

Switch to the floor plan view "Level1" in Revit.

In VirtuSurv now run the command "Wall" [icon] and in the options dialogue box select the option

"Continuous wall", and in the type selection the wall type "Generic - 200 mm". Make sure that in the options dialogue box that the automatic level mode is activated and that the point mode "Standard" is selected.
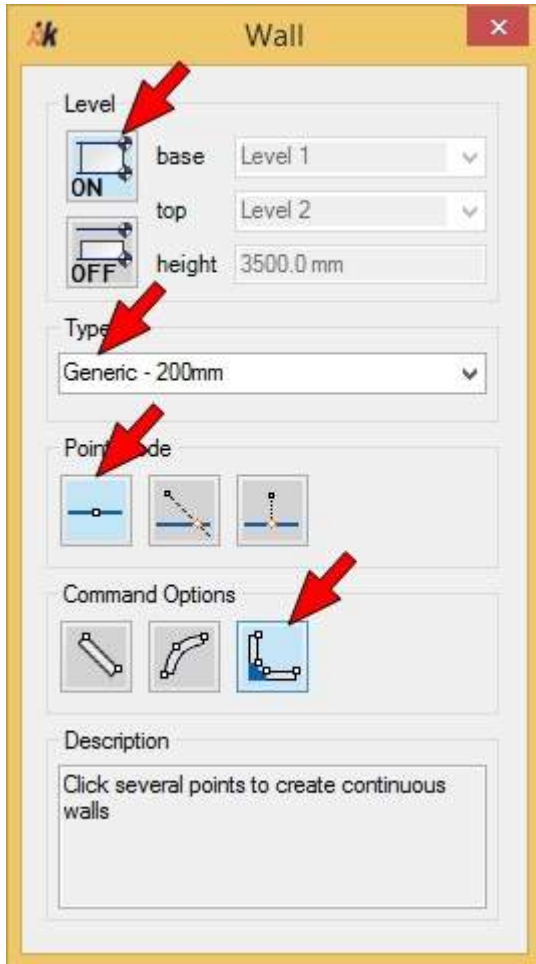
Now pick two points on each wall as shown below. It does not matter whether you pick the left point first or the right. As-Built for Revit automatically recognises the alignment of the wall face. Take care not to click directly on edges as such a measurement is not precise enough. The intersection of the walls follows automatically. Start with the wall with the small window.

Then continue with the separating wall at the stairs.

Pan the scan view to the left and continue with the left of the wall next to the left of the stairs and then clockwise around the room.
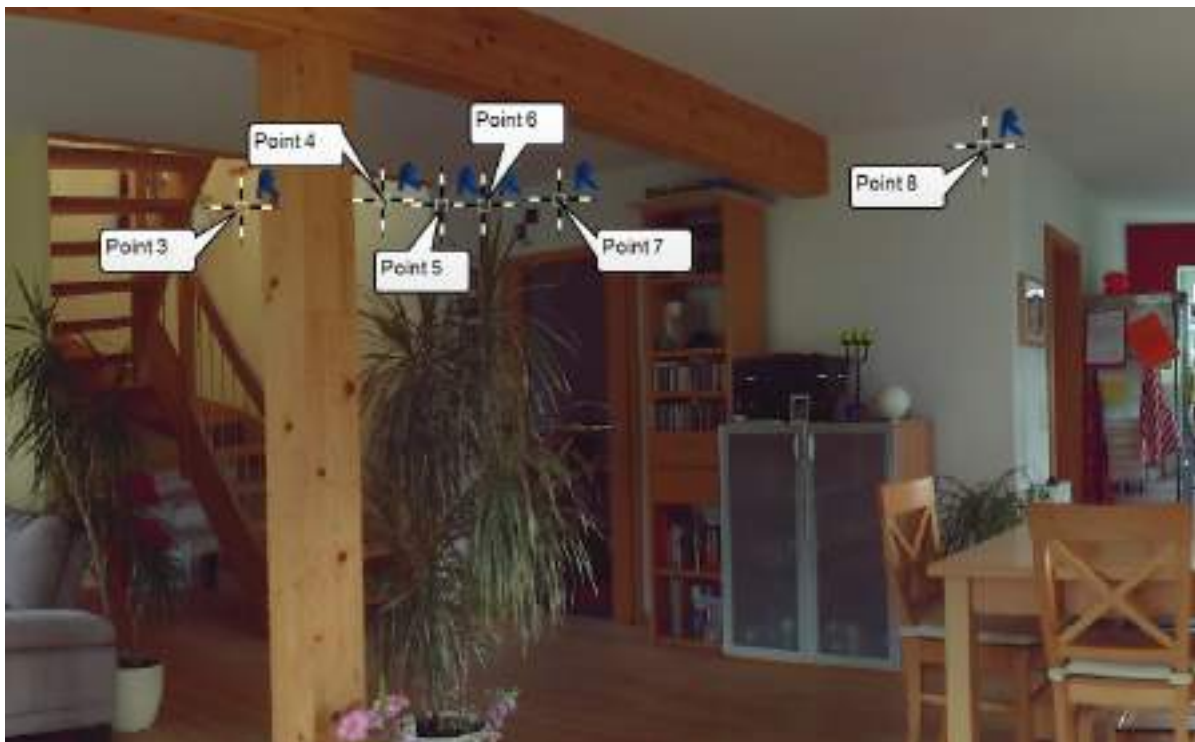
Point 15 is the first point on the last wall. The scan is an all around scan (360°) and it has been split on the last wall for creating the planar view. Therefore you have to zoom out, pan the view to the left part of the scan and zoom onto the corner with where you started to draw the outline of the living room in order to measure the second point on the last wall. Now pick point 16.

Exit the command either by right-clicking in the VirtuSurv scan window, or with [ESC] when the Revit program window is active.

Switch to Revit, open a 3D View ⬡ (Tab "View" → group "Create" → Dropdown list 3D view → Default 3D view) and zoom to extents 🔍.
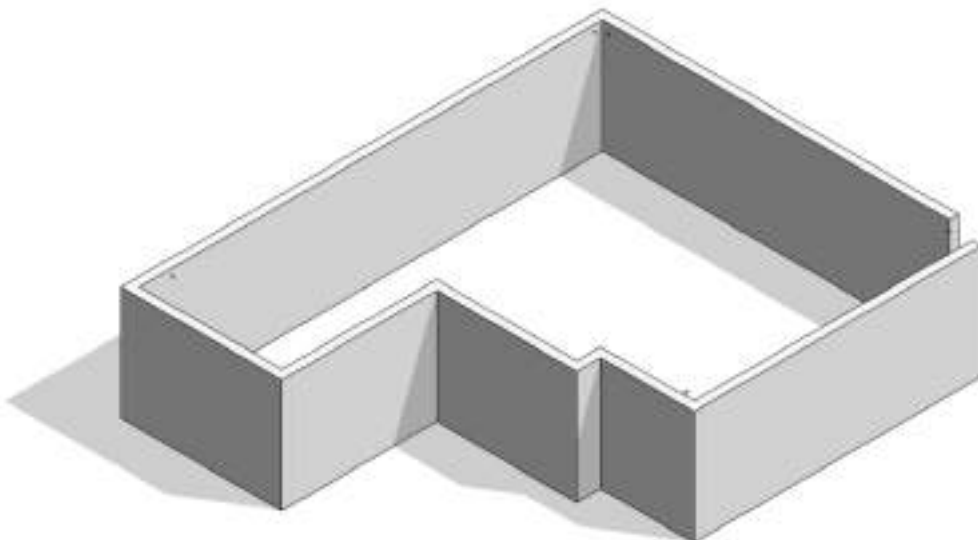
You will see the wall walls that you just created. The automatic level option of As-Built for Revit ensures that the top edge and the bottom edge of the walls are linked with the levels "Level 1" and "Floor_2", which you previously created.

Use the "Trim/Extend" command from the "Modify" tab for intersecting the first and last wall sections and select them one by one.

In the area of the stairs the room outline is not yet complete. Change to scan 3 in VirtuSurv and select the command "Wall" with the "Straight wall" option. Then pick two points on the narrow partition wall next to the stairs:
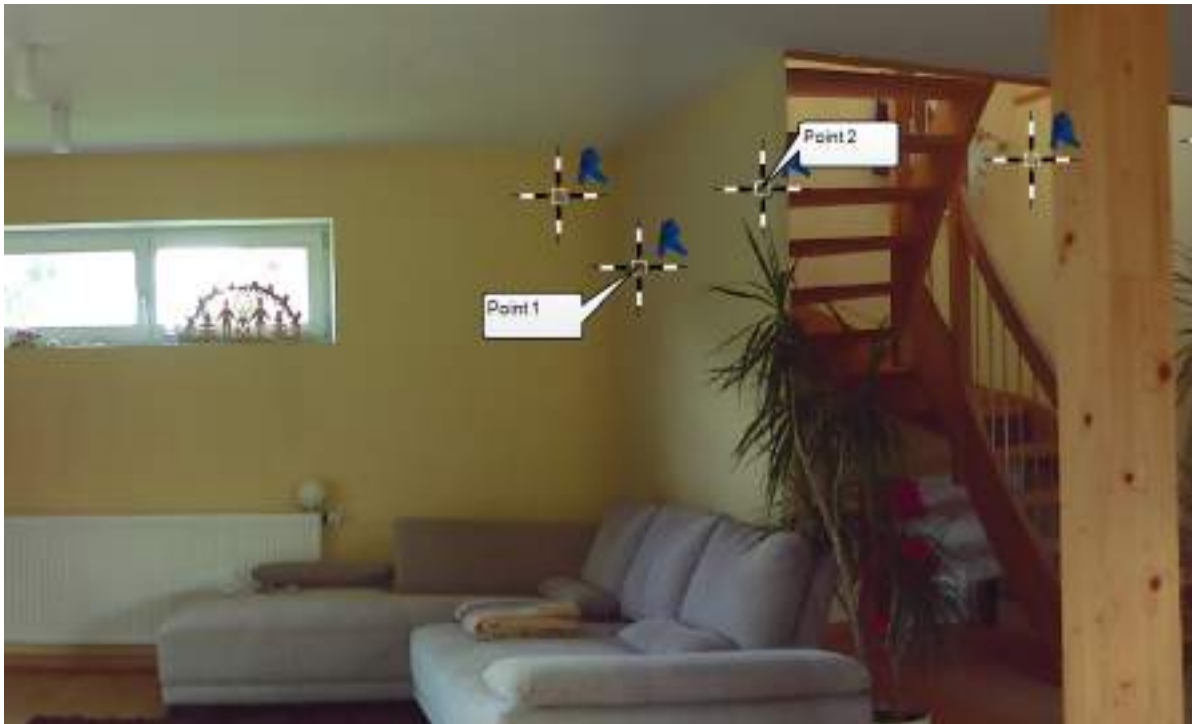
To adjust the wall thickness of the partition wall change to scan 5 and select the command "Wall side" with the option "Straight wall" and click on the second side of the partition wall to the stairs.
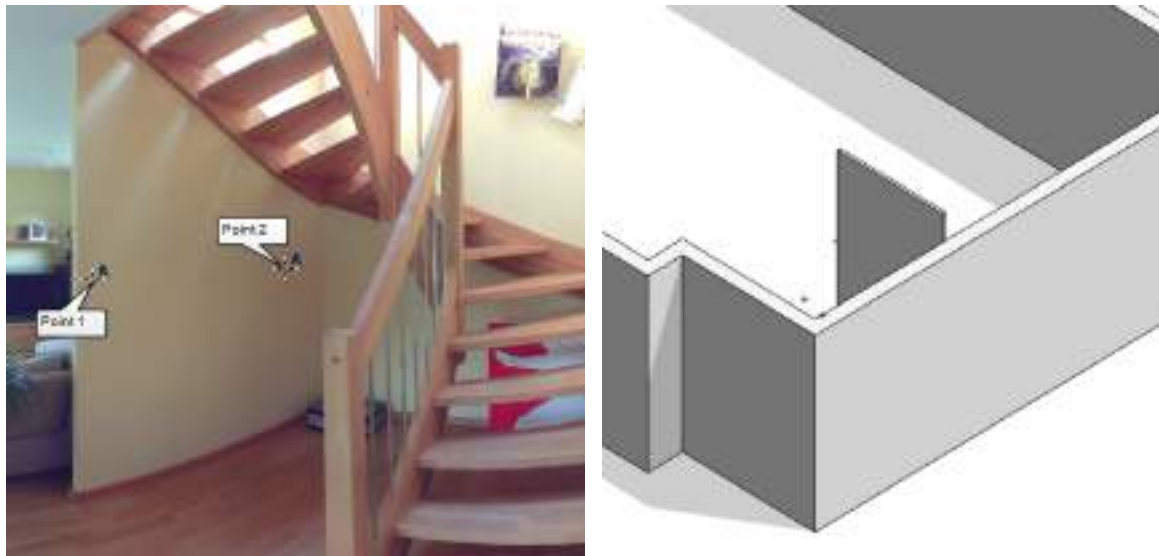
The wall width of the separating wall to the staircase is automatically adjusted to agree with the measurements. VirtuSurv creates a new wall type in Revit with the correct wall thickness. To connect the partition wall with the exterior wall, on the Revit tab "Modify" select the "Trim/extend single element" ⇥ (group "Modify"). First select the inner face of the exterior wall and then the partition wall.

Open scan 1 and scan 2 and repeat the process to adjust the wall thickness for both outside walls.

Again, VirtuSurv generates a new wall type with the measured wall thickness. This new wall type is assigned to both outside walls, since they both have the same thickness and the same wall type was selected as the source.

4.3.3 Creation of Doors and Windows

In VirtuSurv switch to scan 3 once more and zoom in on the small window. Select the "window" command [▦] and the option "Using diagonals" . [◻] Select the entry "Double Window 150 x 100 cm" from the type list. Select the option "create new family type" and make sure that the point mode "Standard" [▭] is active:
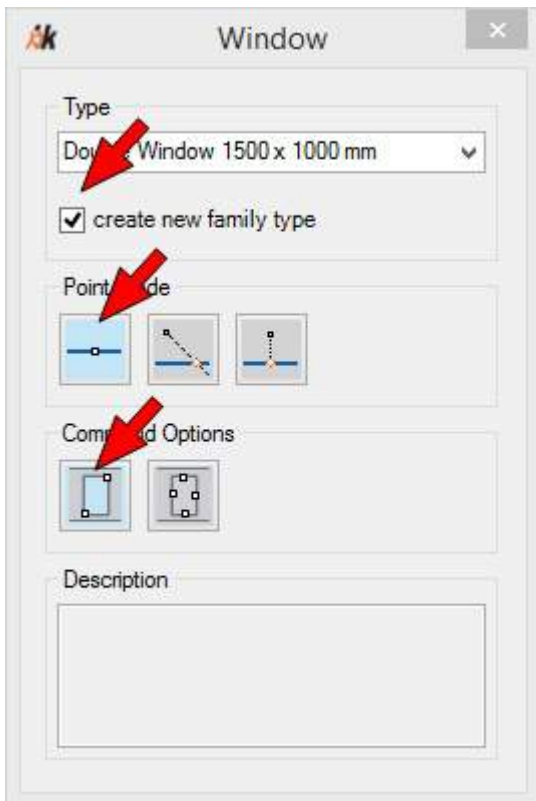
In the scan pick the lower left corner point and then the upper right corner of the window opening:
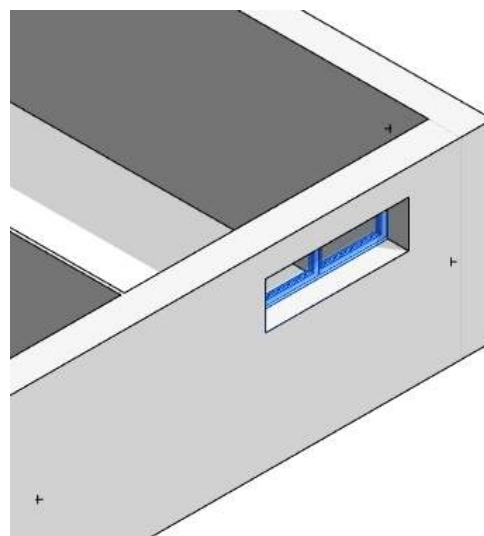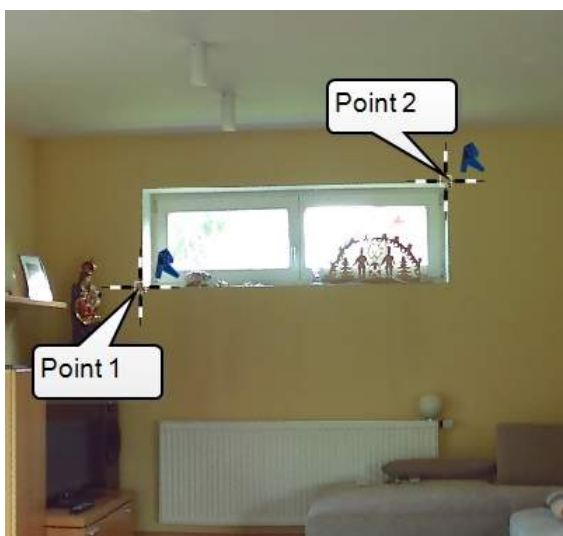
VirtuSurv creates an appropriate window in the existing external wall and at the same time creates a new family type with the correct measured values for the width and height of the window.

Now zoom to the two window doors on the right hand side of scan 3. Because of the baseboard it would be too in-accurate to measure the doors via the diagonal points. Therefore you make use of the point modes. Select the "Door" command 🚪 and the option "Using reveals" ⬚. Activate the "Plumb" point mode ⊥. Select the entry "Double Door 1600 x 2000" from the type list and activate the option "create new family type".
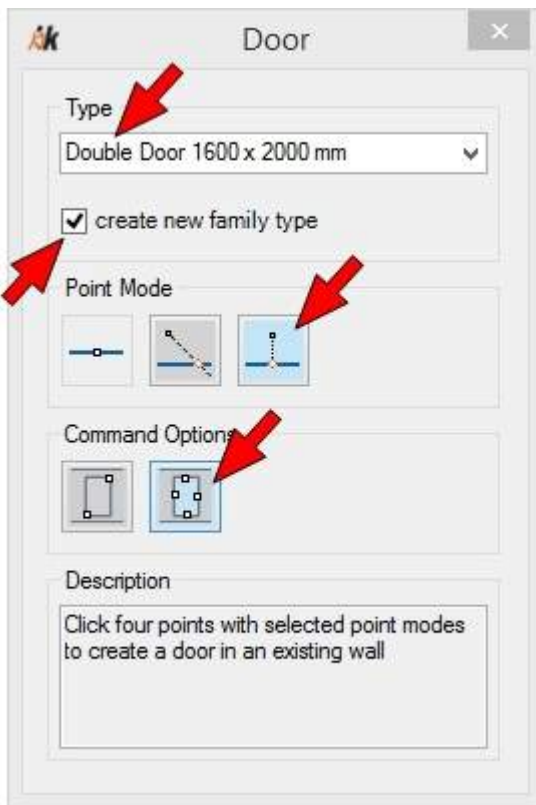
For the right door, pick two points on the reveals, a third one on the lintel and a fourth on the floor. Thanks to the selected point mode the clicked points are projected to the wall face and thus the correct dimensions for the opening are determined.
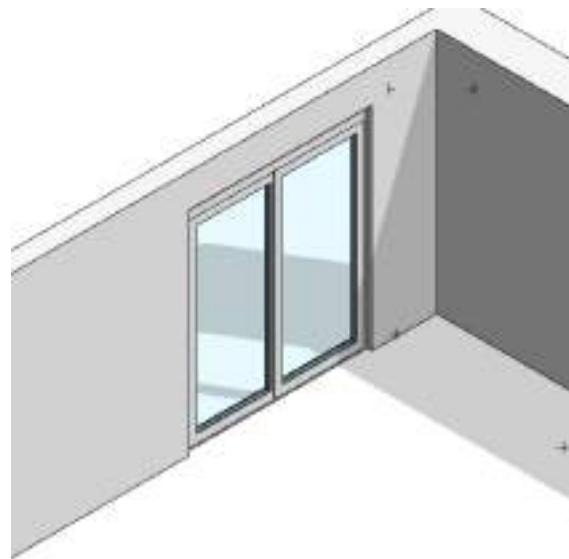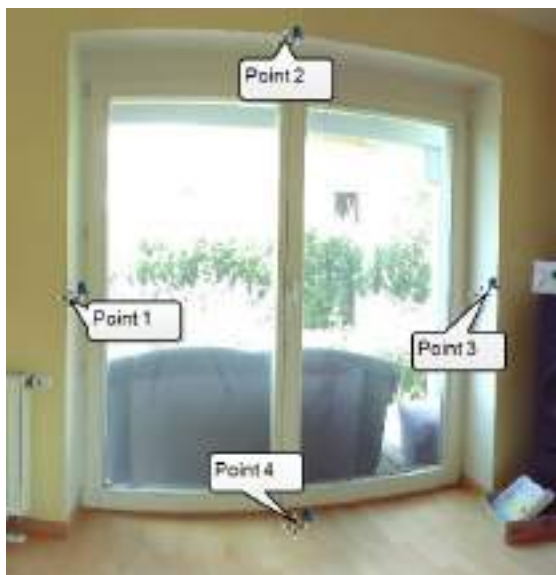
Because the option "Create new family type" was activated, an appropriate new family type" is again created.

Zoom to the left window door. Deactivate the option "Create new family type" and select the entry that has just been created "Double Door 1800.0 mm x 2115.0 mm" from the type list.

With the "Plumb" point mode activated click a point on the floor, one the left reveal and one on the lintel. Since the right reveal is obscured switch to the "Intersection point with work plane" point mode and click a point on the right edge of the door opening:
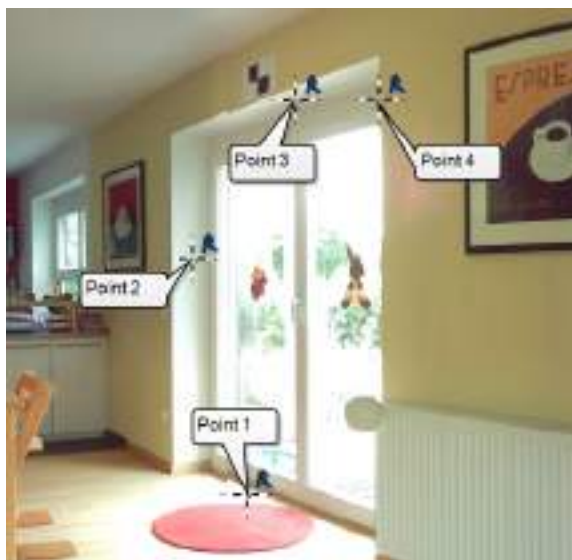
4.3.4 Working with construction aids

Next we want to generate reference lines for the construction of the staircase in Revit. Therefore switch to scan 5 and zoom thus the stairs are completely visible.

Select the command "Model line" ✏ with the option "Horizontal model line" ⊟. Make sure that the point mode "Standard" ⊟ is active and click two points on the front of each step of the stairs. Start with the bottom step and continue upwards as far as the steps are visible:
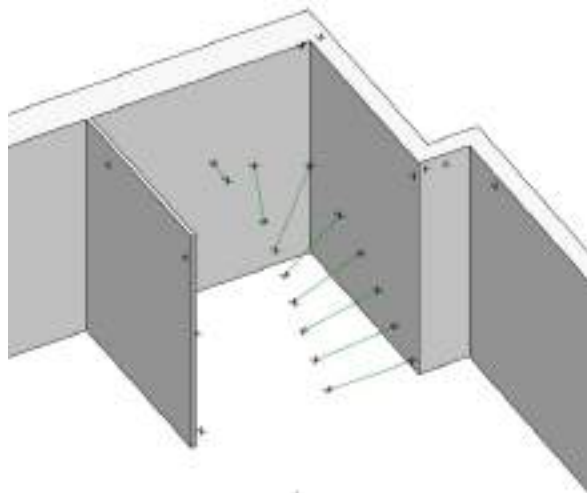
Switch to Revit and there to the view "Level 1". Zoom to the staircase thus you can see the model lines as well as the walls.

In the properties of floor plan "Level 1" under Extents you will find the "View Range" entry. Open the dialogue box via "Edit" and change the cut plane in the view range dialogue box to 1800 millimetres, thus all model lines are visible in the view:

In VirtuSurv select the "Detail line" command ⬜ and in scan 5 click two points on the left and on the middle stair string of the stair well. Take care that you actually get points on the stringer and not on the rail!
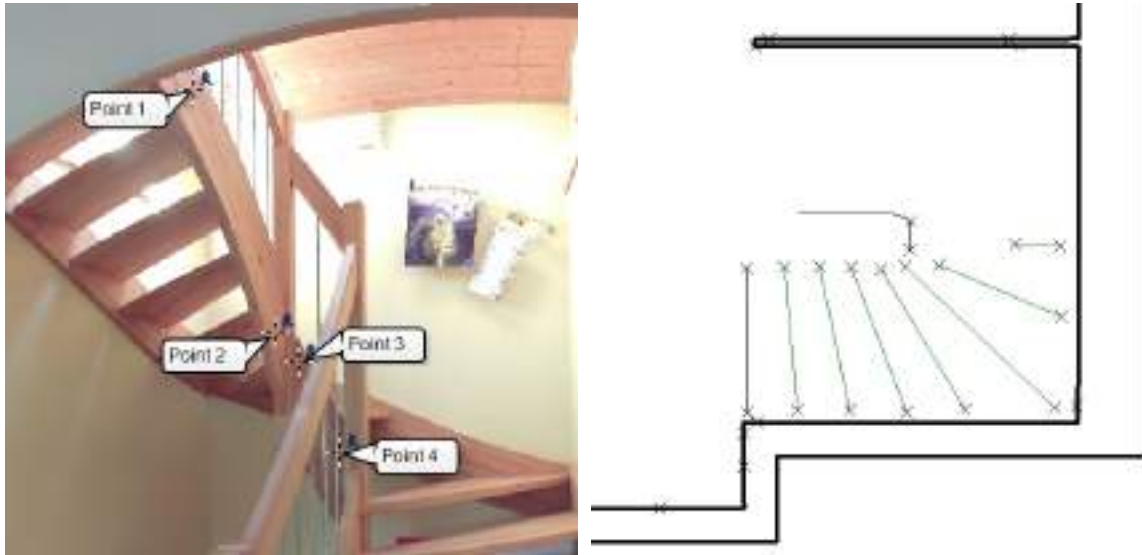
In Revit delete the spare line segment between both stringers.

Switch to VirtuSurv (Scan 3) and add the right stringer of the stairs by drawing another detail line.
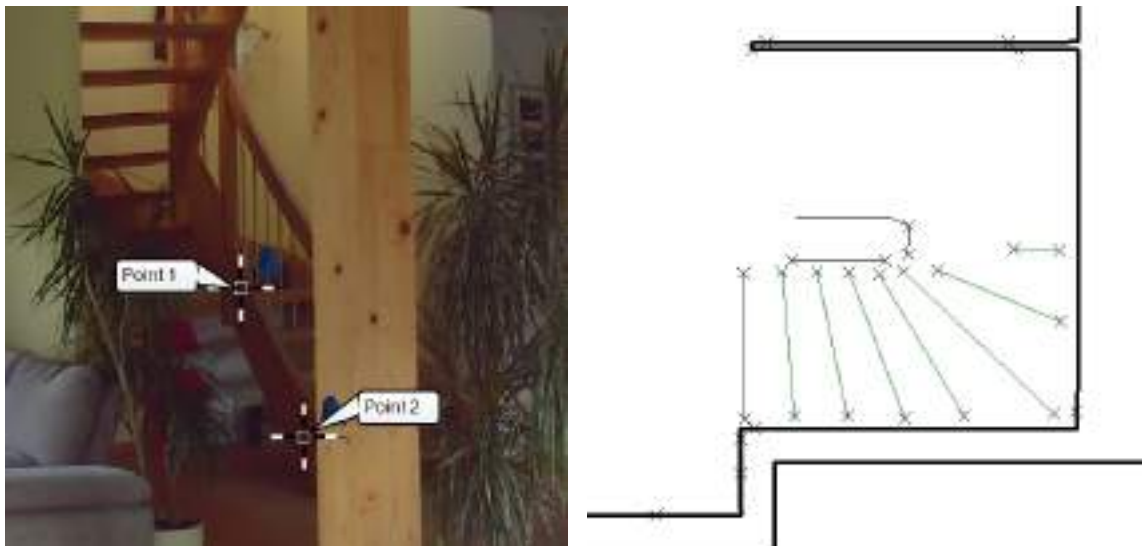
Then (in Revit) got to the As-Built tab in the ribbon, group view and select the "Hide PP" ⬜ command for hiding the protocol points in the current view for more clarity. Now you can start constructing the stairs. Use the command "Stair by sketch" ⬜ from the "Architecture" tab (panel "Circulation" → Dropdown list "Stair" → "Stair by Sketch"). Start drafting the boundaries of the stairs in the layout (tab "Modify" → group "Draw" → "Boundary" ⬜) by picking the lines

of the surrounding walls as well as the three detail lines of the stair strings using the "pick lines" command (group "Draw"). For the front closure of the stair boundary you may pick the line of the first step with the command "pick lines" and then intersect the other boundary lines with it:
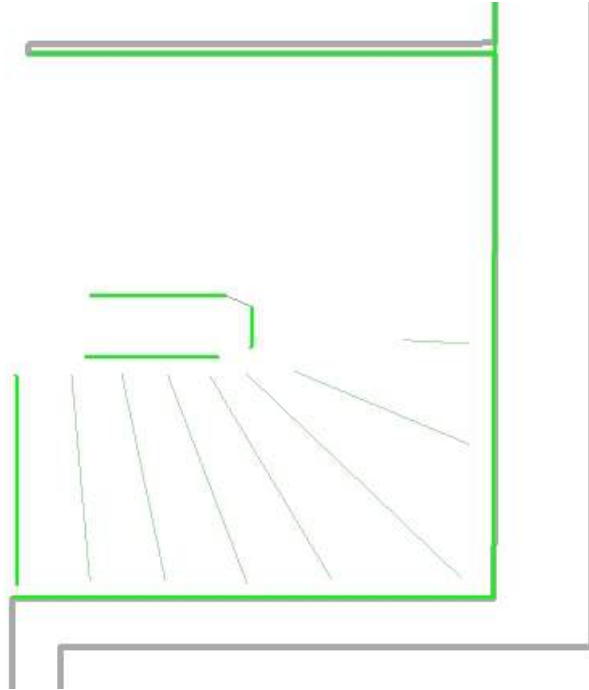
Complete the sketch using the "Trim/Extend to corner" command of Revit. Now delete the sketch line at the first step again. The final sketch should look like this:
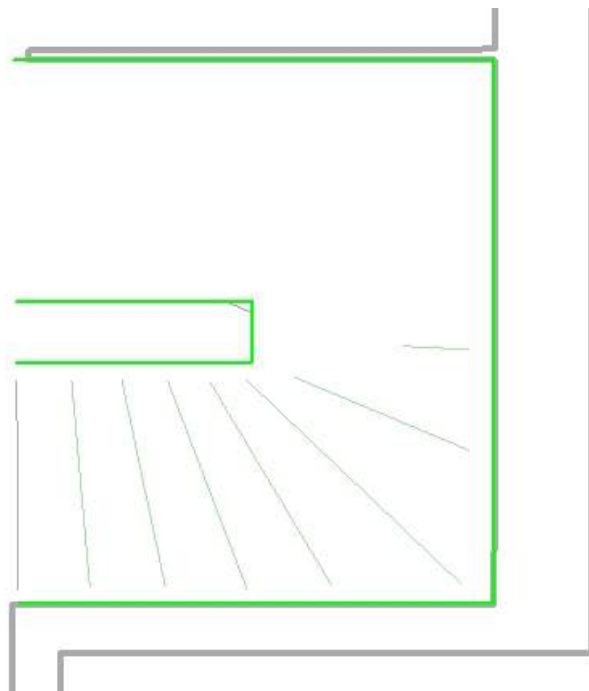
Now, draw the risers (tab "Modify" → panel "Draw" → "Riser") by picking the model lines earlier created in VirtuSurv using the "pick lines" command and extending the risers to the boundaries of the stairs using the "Trim/Extend multiple Elements" command. To do this first pick the stairs boundary line and then successively pick the individual lines of the steps:
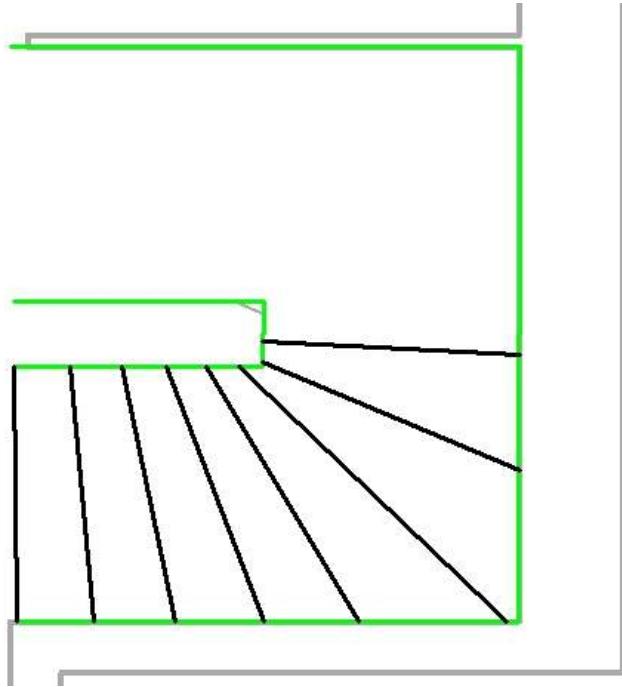
Finish the sketch of the stairs by confirming the green checkmark . If necessary change the direction of the stair run by clicking on the blue arrow at the lowest step, then switch to the 3D view. You may delete the two rails since they do not apply to the exiting stairs. To complete you have to change the parameters of the stairs so the steps get the correct height. Select the stairs and choose the stair type "standard" from the properties palette as well as following settings:

Parameter	Value
Constraints	
Base Level	Level 1
Base Offset	0.00
Top Level	none
Top Offset	2.895
Dimensions	
Desired Number of Risers	13

Your stairs should now look like this:
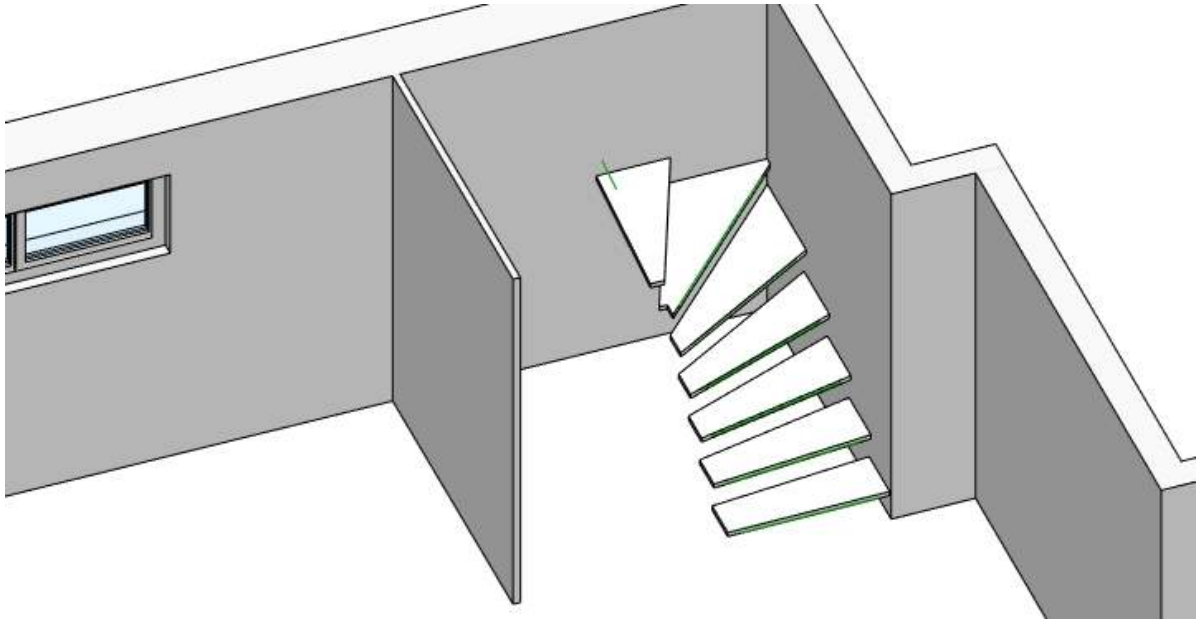
4.3.5 Beams and columns

In the next steps the beam and the column will be inserted. In scan 3 in VirtuSurv select the command "Line based structural family / beam" 🖫. In the type selection of the options window select the entry "Laminated wooden beam 170 x 287 mm" and the option "place horizontally" 🖫. Make sure that the point mode "Standard" 🖛 is active and click one after the other two points on the front face and two points on the underside of the beam:

To connect the beam with the exterior wall, on the Revit tab "Modify" select the "Trim/Extend Single Element" ⊿ (group "Modify"). First select the inner face of the exterior wall and then the beam. Then repeat the procedure on the opposite end.

Select the "2 level based family / column" ▯ command. In the type selection of the options window select the entry "Rectangular wooden column 170 x 170 mm" and the option "rectangle" ⊞. Make sure that the automatic level mode ▦ is activated and that the point mode "Standard" ⊶ is selected. Click one after the other two points on the front face and one point on the side face of the column:
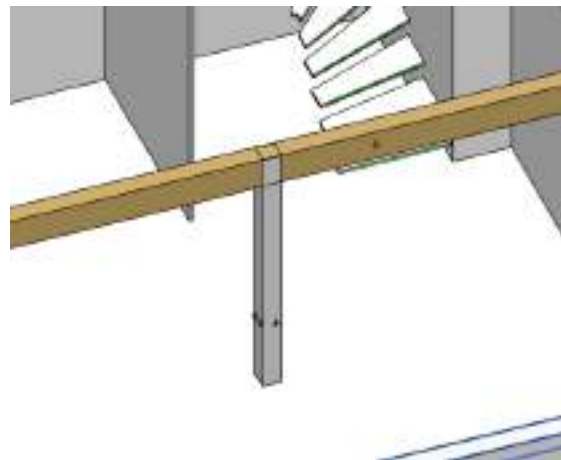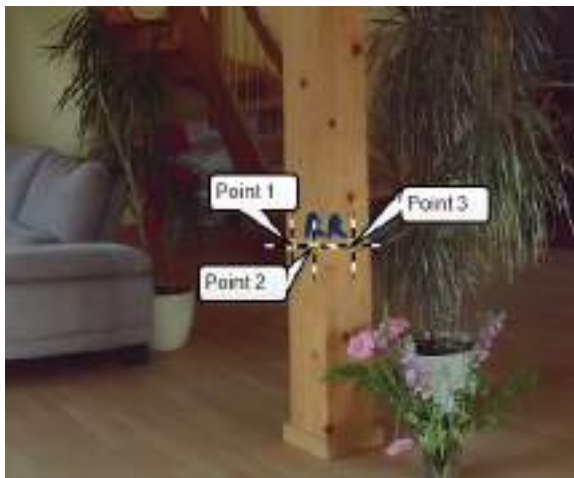
To connect the column to the beam, select it in Revit and activate the command "Fix top/base" ▯ from the "Modify | Trim" menu. Now pick the beam. The column and the beam are now connected.

The results of the exercise should look something like this (for the visualisation the "realistic" style has been selected and display of shadows switched on):
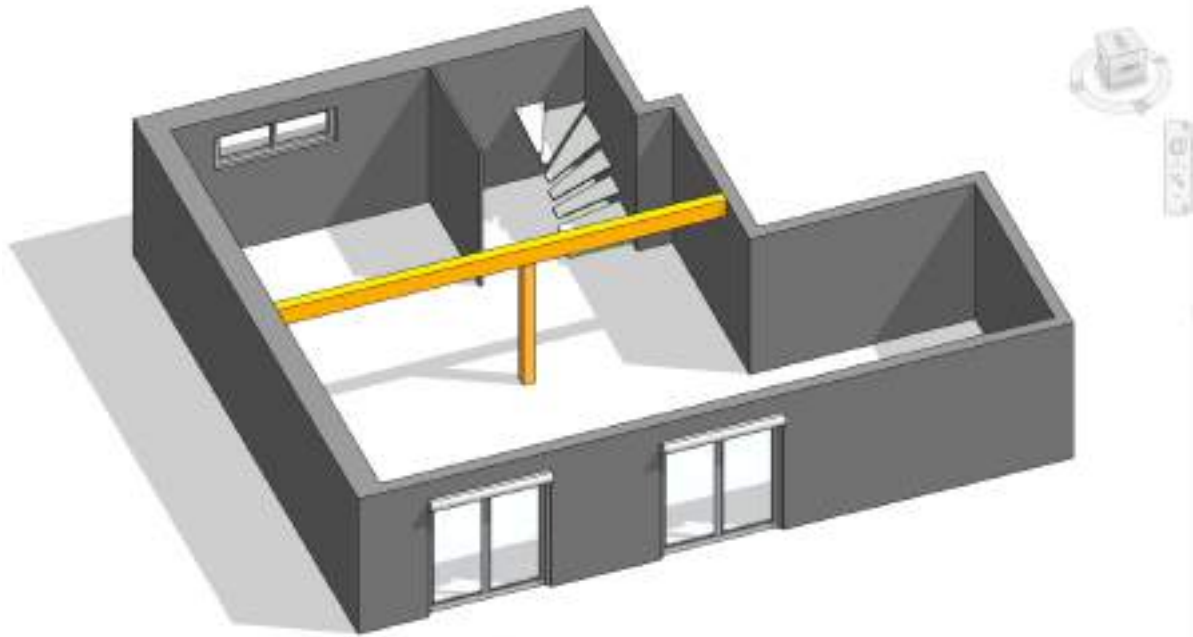
You may add further openings and elements as you like. For completing the stairs, we recommend to mark the bottom side of the steps by construction points and then defining the work plane of the construction point as active work plane (tab "Architecture" → group "Work plane" → "Set"). Now you may place the front edge of the step exactly on that plane by using the VirtuSurv "3D model line" command together with the "Intersection point with work plane" point mode in order to draw the risers in the stair sketch.

4.4 Tutorial 3 - Creating a pipe run

This tutorial demonstrates the typical workflow of how you can model a pipe run in a point cloud. You can find this tutorial on the FARO web site for download:

http://download.faro- 3d- software.com/all_ languages/Tutorials/As- Built/Tutorial_ As-BuiltRevit_1.zip

4.4.1 Preparing the Project

Unzip the ZIP archive. You will find therein the folder "Piping Tutorial" with the REVIT project file (RVT) as well as the point cloud data (RCS). Open the Revit project file.

The project file already contains a reference to a point cloud. Because of the changed saved path it cannot be found and the following message appears:

Select the option "Open manage links to resolve the problem". The Revit dialogue box "Manage Links" is opened. Select the tab "Point Clouds": [1]

---
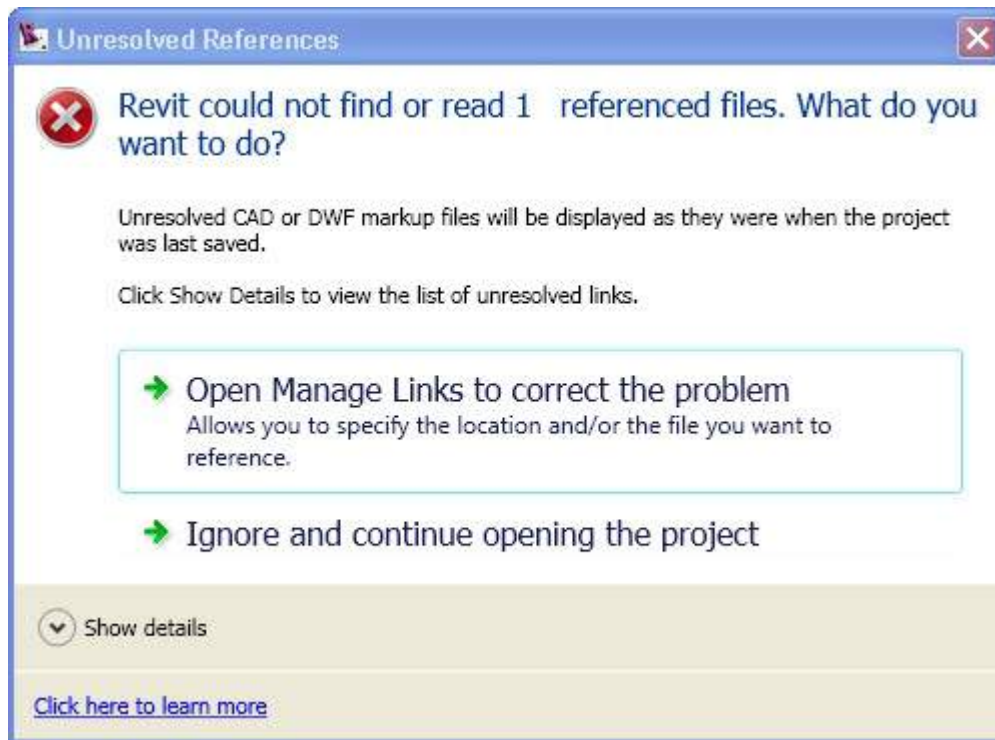
[1]Should the message not be displayed, then open the dialogue box "Manage Links" from: tab Manage → panel Manage Project → Manage Links.
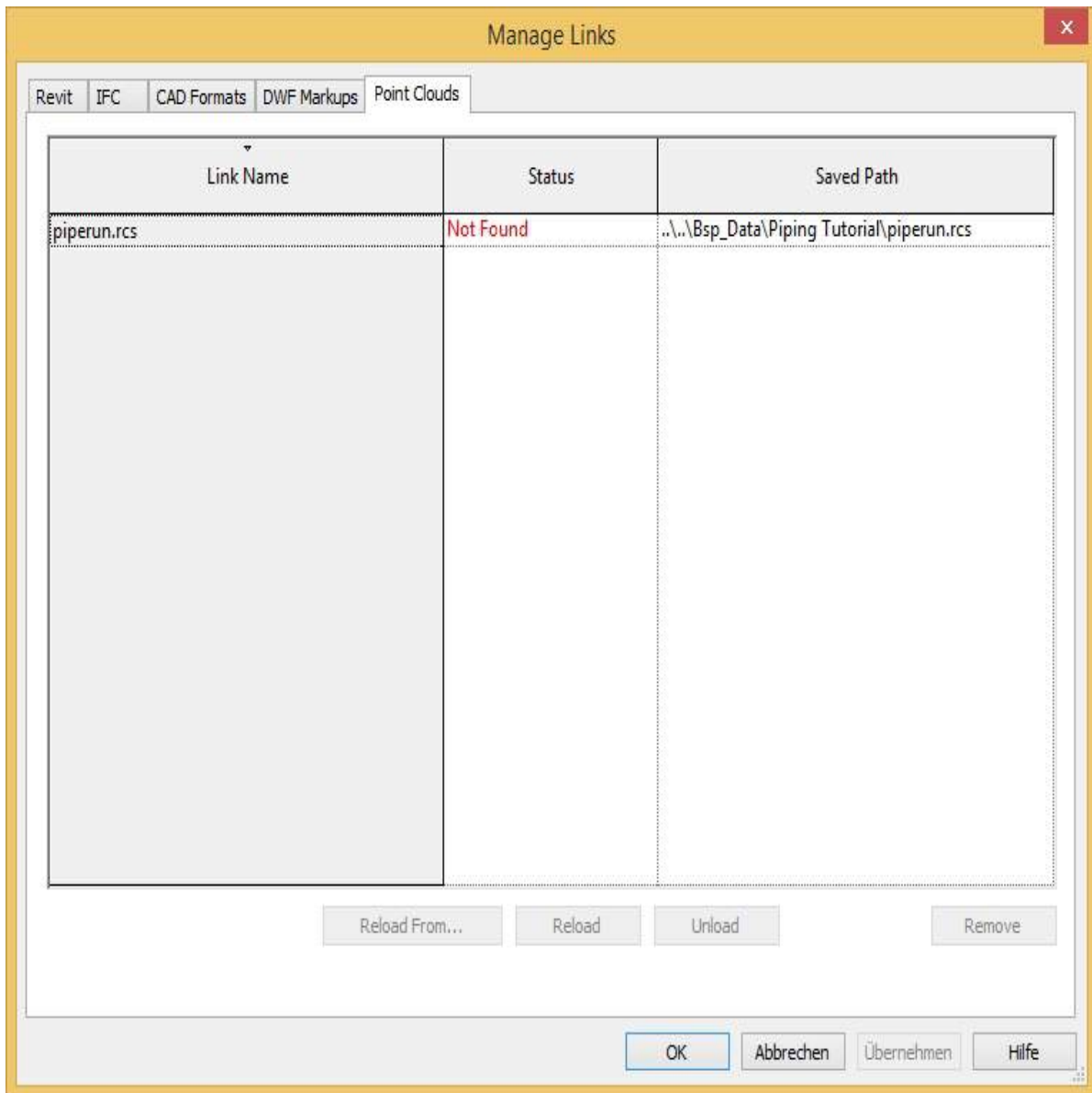
Left click on the entry "piperun.rcs" and then click **[Reload From ...]**. Navigate to the folder containing the tutorial data and then select the point cloud file "piperun.rcs". Exit the dialogue box by clicking **[OK]**. The point cloud is now correctly referenced in the project.

View the project in the default 3D view (Ribbon View → Create Group → 3D View). You see the part of a point cloud for the pipe run.
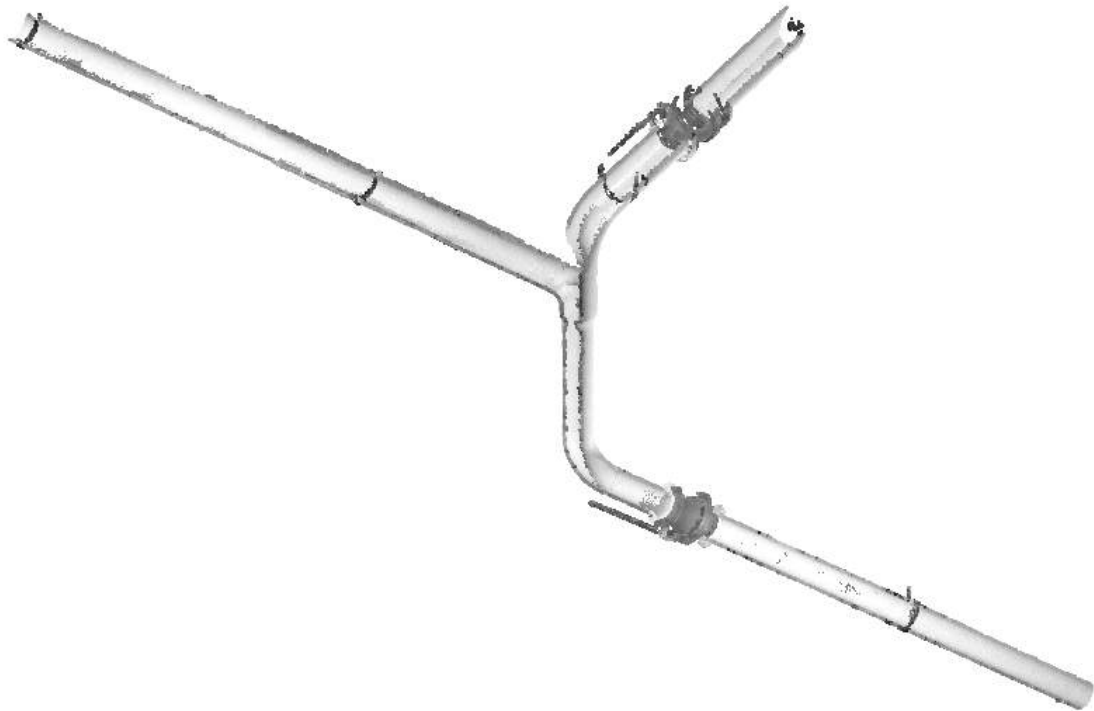
Note: The point cloud will be displayed highlighted whenever the mouse pointer hovers over it. This can sometimes be very irritating when you are working. To prevent this happening disable the option "Select links" on the icon below the drawing window or by unchecking the box in the "Select" drop down menu on the "Modify" panel in the "Architecture" tab on the Ribbon:

4.4.2 Fitting pipes

The sequence of steps for creating connected pipe runs using As-Built always follows the same method of working:

1. Fitting pipes and if applicable adapting pipe types to a new diameter.

2. Inserting pipe fittings between the pipes

3. Align the pipe run to observe consistency constraints (coaxial or coplanar axes)

So let's begin with the fitting of the pipe. When in the active 3D view start the command →
Object Tools - Pipe Fitting . The following dialogue box appears:
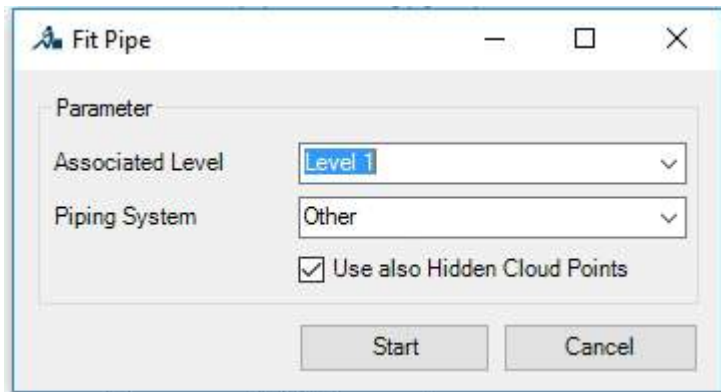
As the precise type of system for the pipe run is not known, please select "Other". Begin the fitting over [Start] and pick two points in the point cloud along the surface of the first pipe:
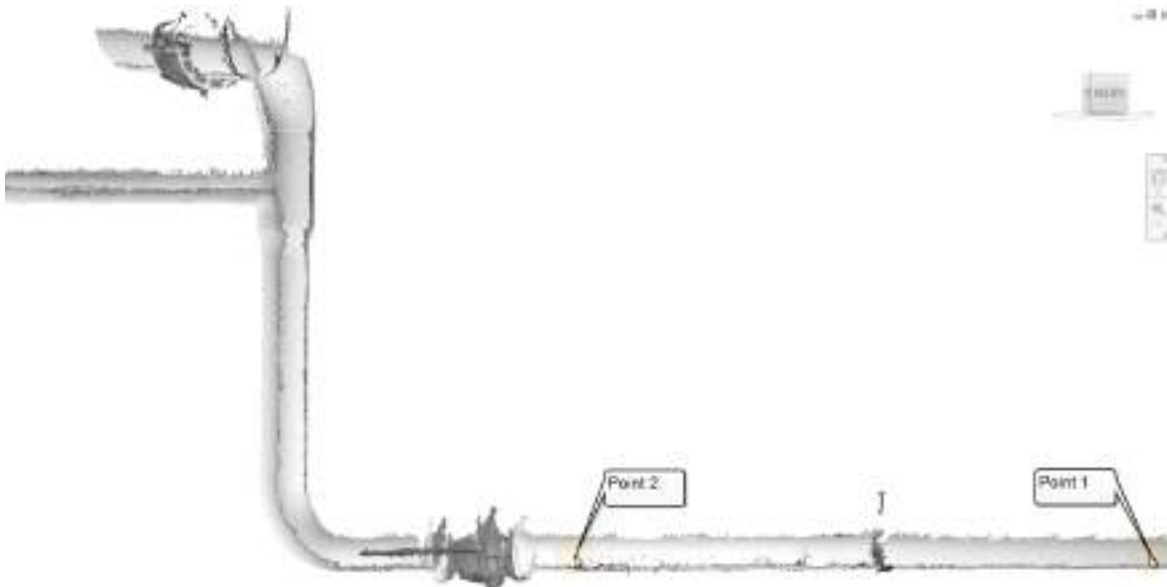
The points must not lie precisely on the start and end of the pipe. Connecting the individual pipes to each other using pipe accessories is carried out in the next step, whereby the pipes can then also be extended if required.

Based on the selected points As-Built determines the pipe diameter and suggests a suitable pipe segment:
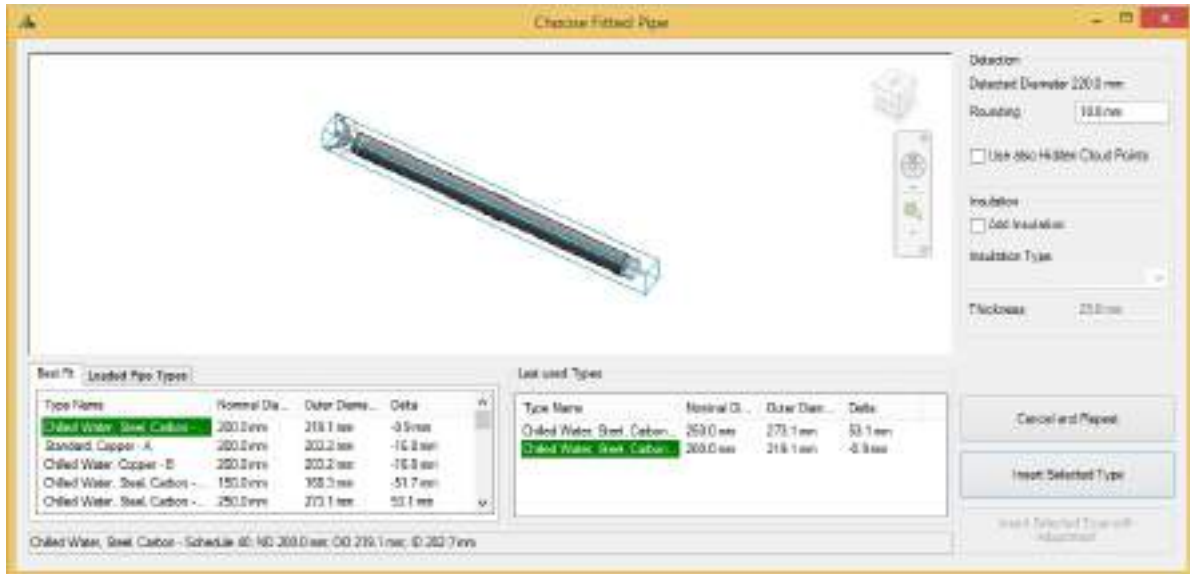
In the preview pane of the dialogue box you can visually check the fitting of the pipe onto the point cloud. In the list **"Best Fit"** you are shown at the top of the list the best suitable pipe segment with its nominal diameter, external diameter and delta to the diameter determined in the point cloud. If you want to use another pipe segment, you can at anytime select another entry in the list.

Note: If you have in addition to the point cloud some VirtuSurv data, then you can also pick the points for the pipe fitting in the VirtuSurv scan view. The fitting itself however takes place in the Revit point cloud. If the option **"Use hidden point cloud points"** is selected, then the fitting can also be carried out, even if you have picked the points in VirtuSurv and the point cloud in Revit is hidden.

Confirm the entry **"Chilled Water, Steel, Carbon - Schedule 40"** with the nominal diameter **"200 mm"** over the button **[Insert Selected Type]**. The pipe is then inserted into the project and you can seamlessly continue fitting the next pipe:
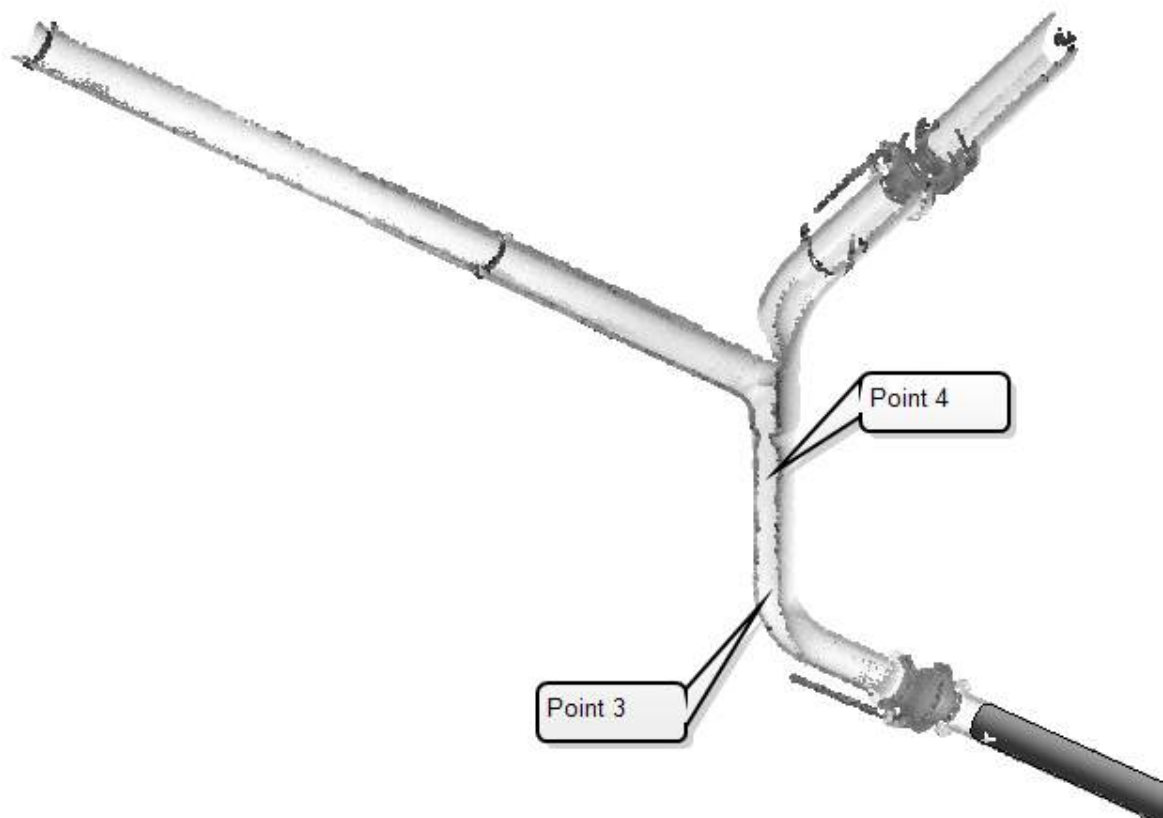
Select for this second pipe the entry **"Chilled Water, Steel, Carbon - Schedule 40"** with the nominal diameter **"200 mm"** and insert it with **[Insert Selected Type]** . Continue with the next pipe:
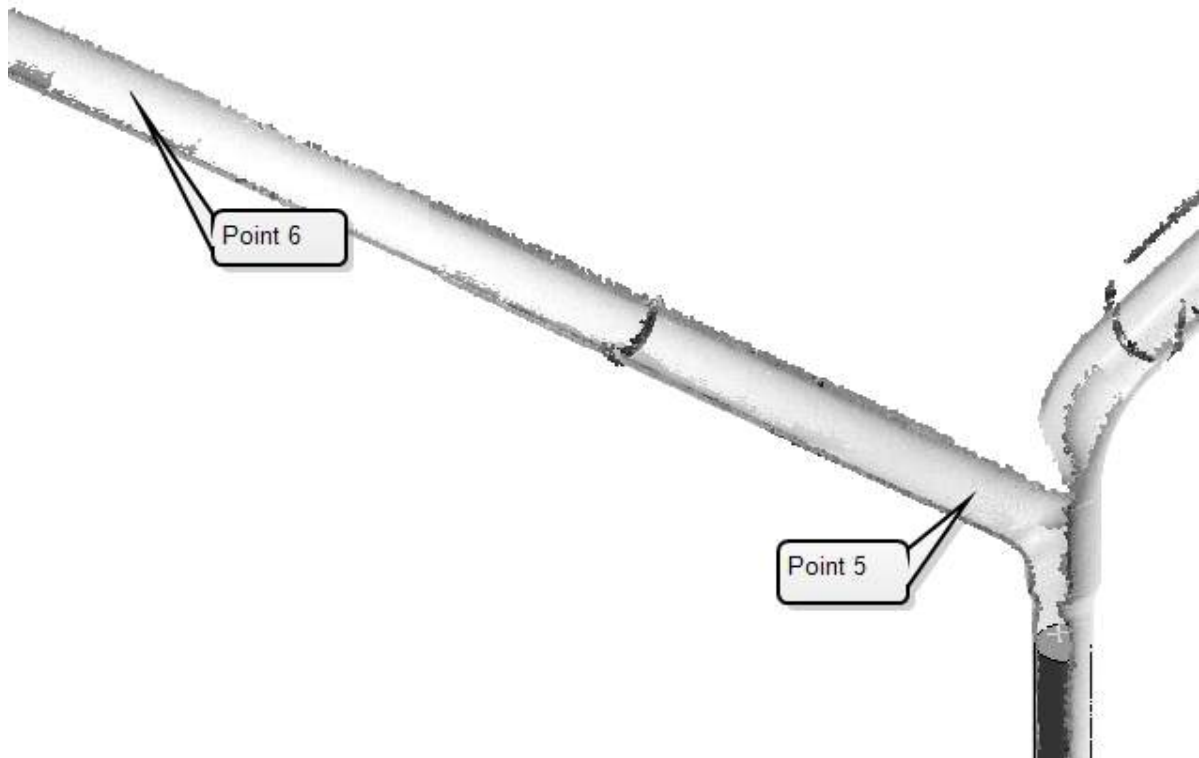
As-Built detects here a different diameter. Select the entry **"Chilled Water, Steel, Carbon - Schedule 40"** with the nominal diameter **"250 mm"** and insert the pipe. Continue with the last pipe:
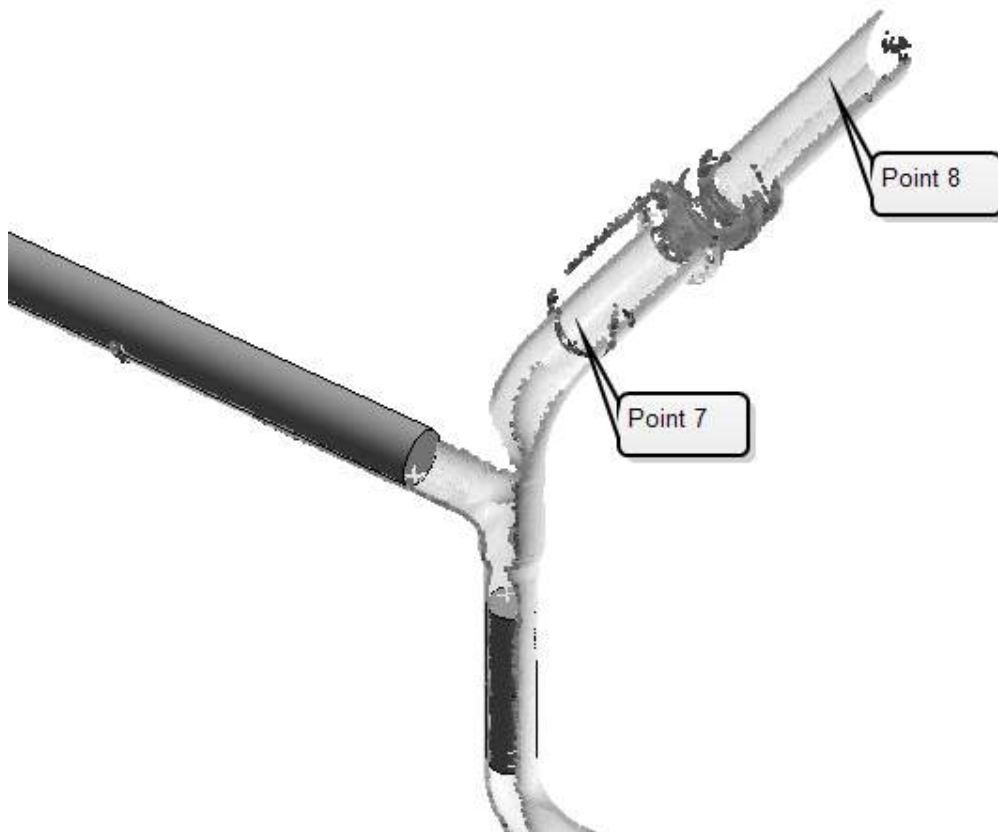
Again select the entry **"Chilled Water, Steel, Carbon - Schedule 40"** with the nominal diameter **"250 mm"** and insert the pipe.

Should at any point a suitable pipe segment not be found in your project, you can also create over the "Pipe Fitting" dialogue box a new pipe diameter. For this select an entry from the lists "Best Fit", "Loaded Pipe Types" or "Recently Used Types" and confirm with **"Insert Adapted Selected Type"**. A dialogue box is displayed where you can create a new pipe diameter for the selected pipe segment based on the detected diameter of the pipe:
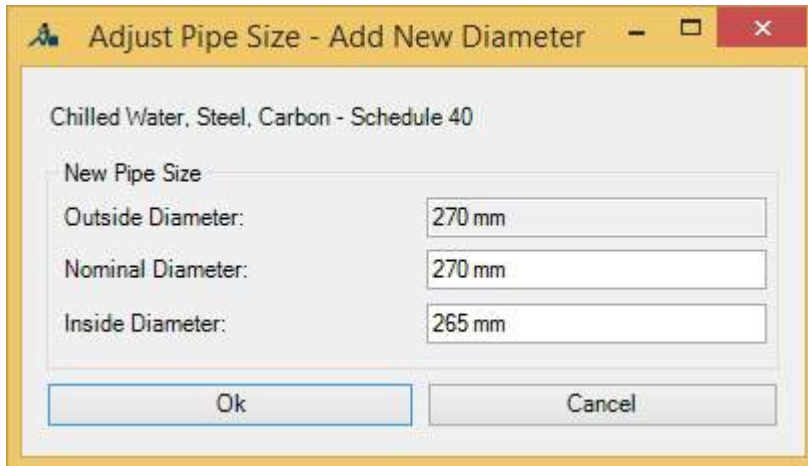
4.4.3 Insert pipe fittings

In the next step suitable pipe fittings have to be inserted between the pipes. From your work with Revit you will be familiar with the tools "Trim/Extend to corner" or "Trim/Extend Single Element" . You will ascertain that it is not possible to use them when fitting pipes, because the axes are not coaxial or coplanar. These consistency constraints will be first dealt with for the whole of the pipe run at the end of the modelling process.

Furthermore we must in this example insert three pipe fittings one after the other at the branch. A reducer, a tee and a elbow.

Start the command → **Object Tools - Placement of Pipe Fittings** and click after each other both of the lower pipes in the vicinity of the pipe ends and finally end the input by pressing the **[Esc]** key:
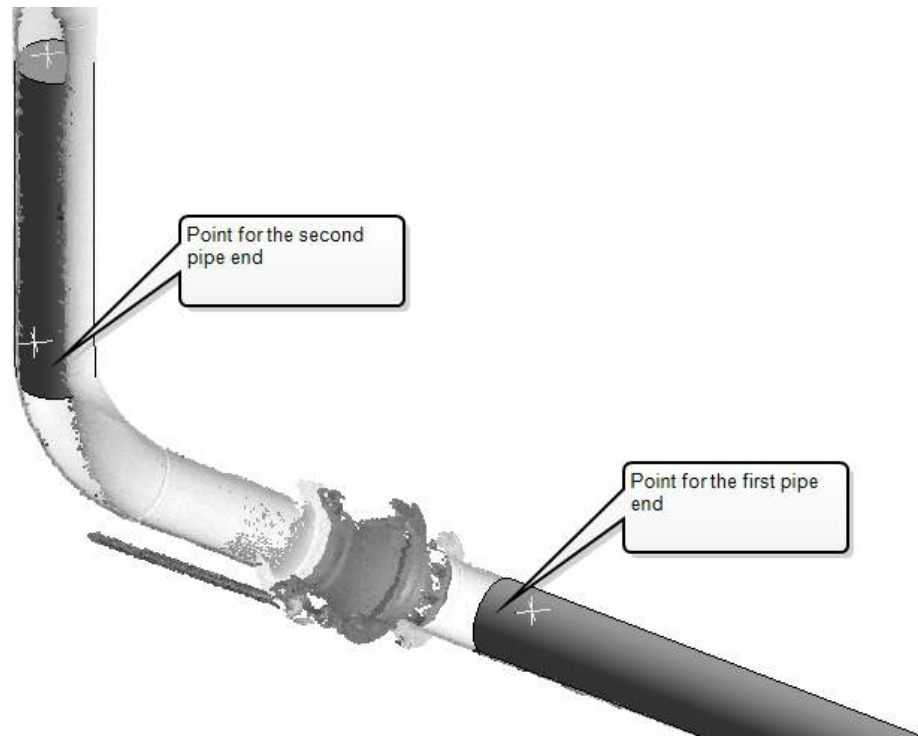
The following dialogue box appears for selecting the pipe fitting:
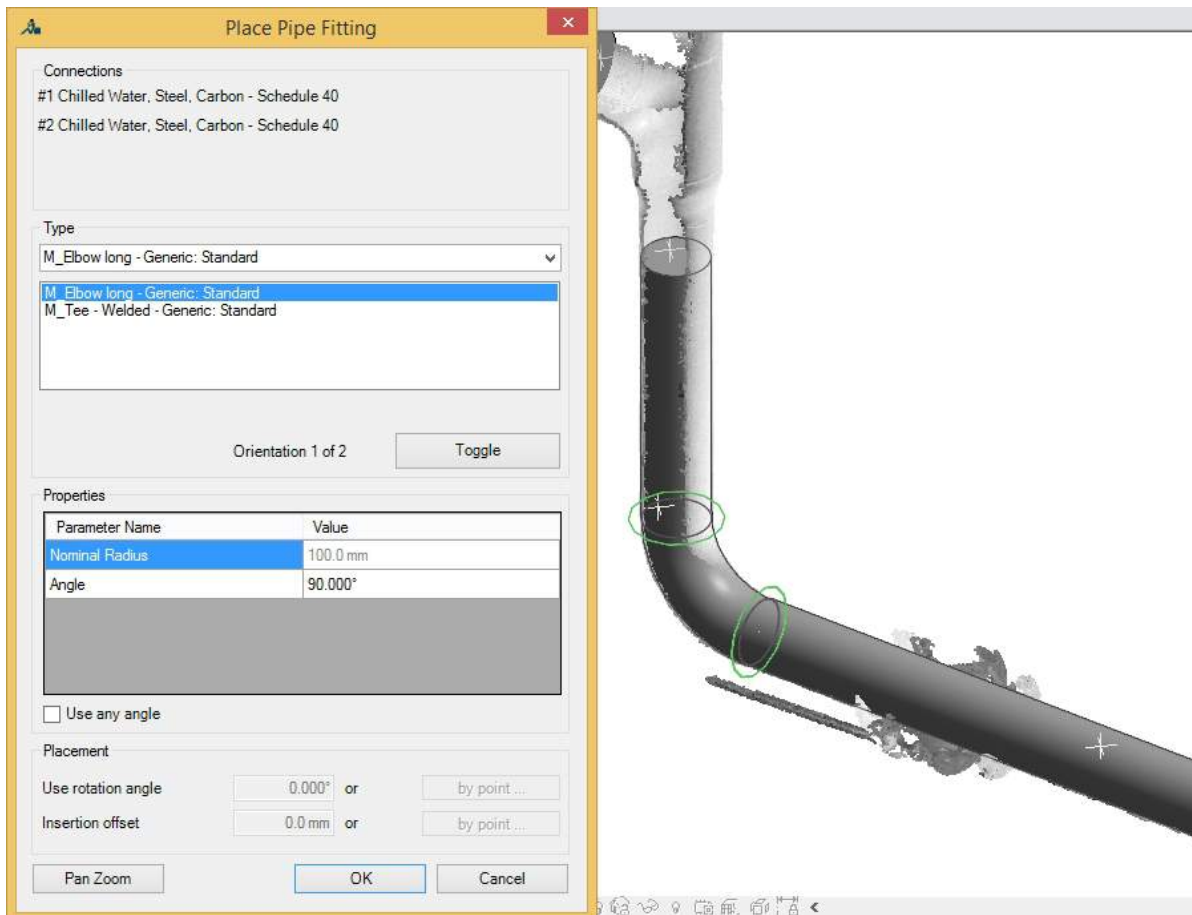
In the upper part of the dialogue box you are shown, as information, the selected pipe joint. In the pick list below you can select the component which should be inserted. The only components listed are those that are suitable for the selected pipe joint. The component that is actually selected is displayed in the project for checking. When you use the command a lot, the most recently used components will be displayed in the list below the type selection box. You can also select the type from here.

Select the component **"M_Elbow long - Generic: Standard"**. In the **"Properties"** group, you can see the As-Built detected angle (Parameter "Angle") rounded to 90°. When you have activated the option **"Use any angle"**, the actual precise angular value between the position of the two pipe axes will be shown. You can control here, whether As-Built should use components with their angle taken from the fitting of the pipes, or if one of the pre-defined angular values of (90°/ 60°/ 45°/ 30°/ 22,5°/ 11.25°) from Revit should be used.

In the **"Properties"** group you can see each of the variable parameters of the selected component. If the parameter value is not defined in the selected pipe joint, it can also be set manually.

When you want to more closely inspect the component before it is inserted, you can temporarily leave the dialogue box using **"Pan Zoom"**. Press **[Esc]** and you will return to the dialogue box.

Confirm the selected component with **[OK]**. As-Built inserts the elbow and extends the connecting pipes. Thereafter you can immediately continue to insert the next component.

For the branch there must be in total three pipe fittings inserted. Start with the tee on the branch. Pick the pipe end, and end the input by pressing the **[ESC]** key:

From the list of the fitting types select the entry **"M_Tee - Welded - Generic: Standard"**. If necessary, click the **[Toggle]** button, to bring the component into the correct position.

Depending on how you selected the points for the fitting of the pipes, it can be that the tee does not yet sit close enough to the branch. The correction of the position is not absolutely necessary at this point, as the following modelling step ("Align pipes") will now bring the components into the correct position. Nonetheless the amount of movement should be shown here.

Point for the pipe end
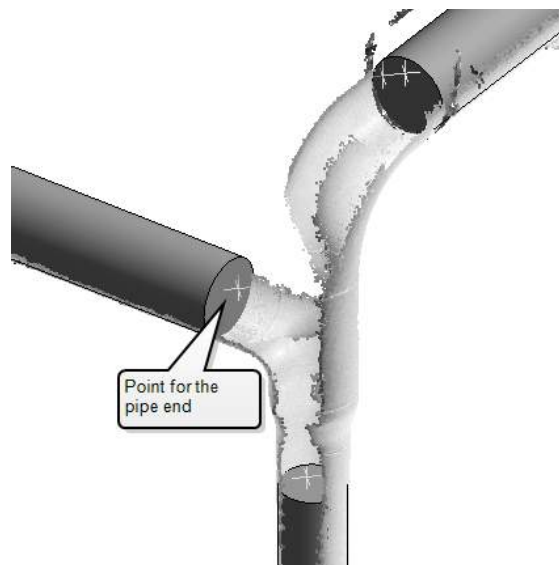
In the **"Placement"** group you find the entry **"Insertion offset"** and the button **[by point ...]**:
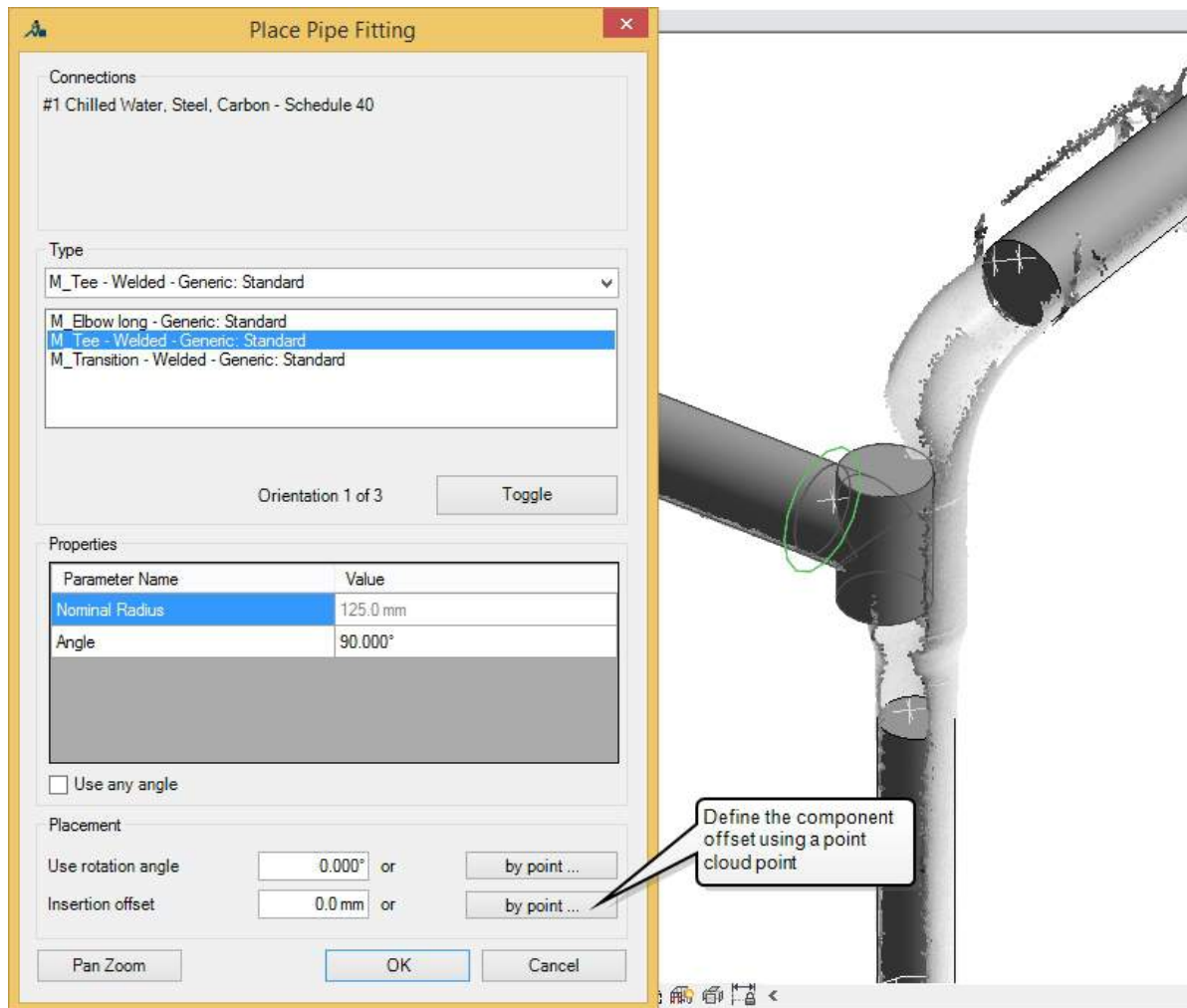
Click the button **[by point ...]** for **"Insertion offset"**. The dialogue box disappears, and you can now pick a point in the point cloud, to which position the component should be offset:
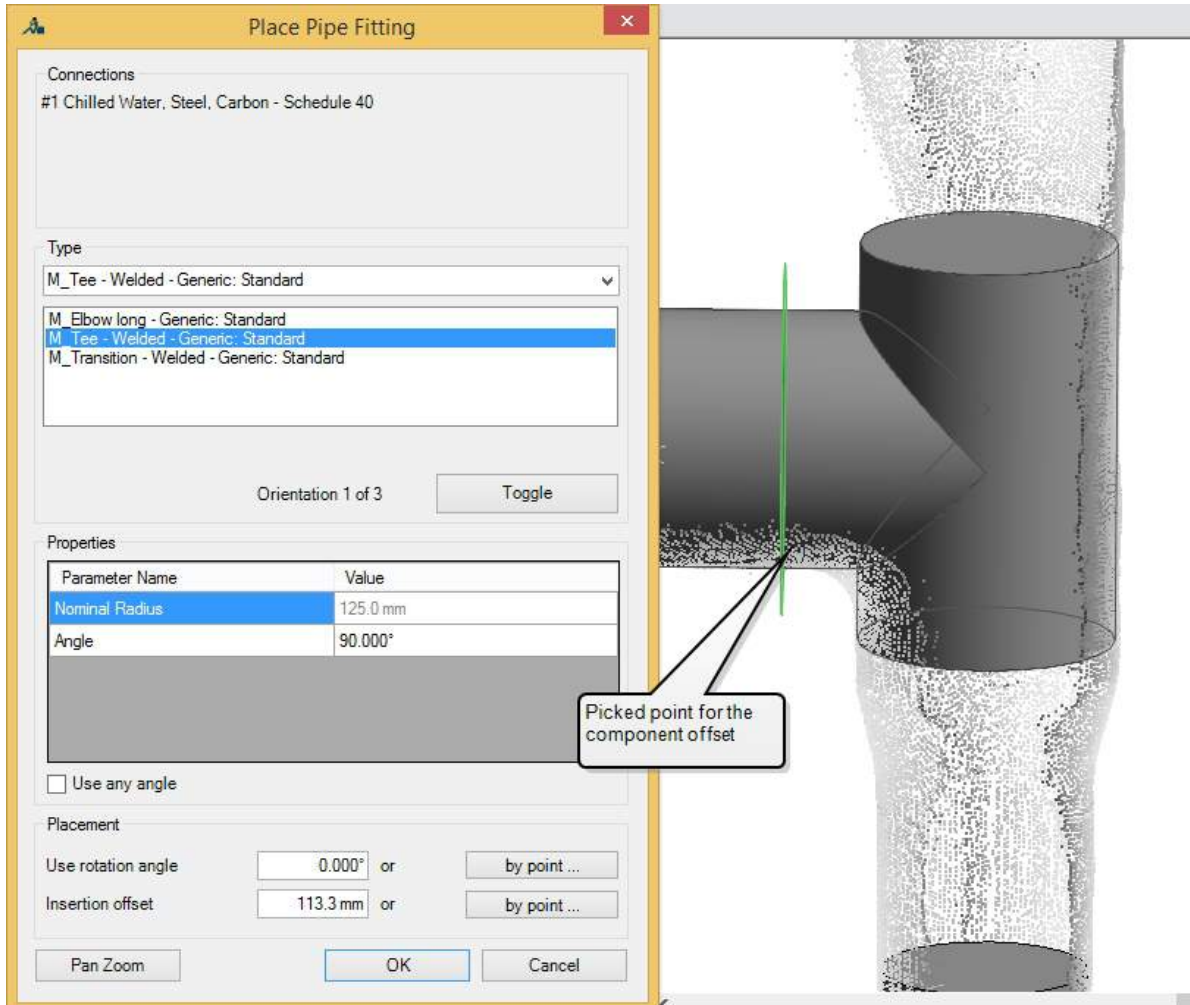
As already mentioned, the offset must not be absolutely correct, as the complete pipe run will be later aligned. Insert the component with **[OK]**.

For both of the two remaining pipe fittings there are again for each of them two pipe joints available. Begin with the reducer and click on the appropriate components (tee and pipe), each in the vicinity of the pipe ends:
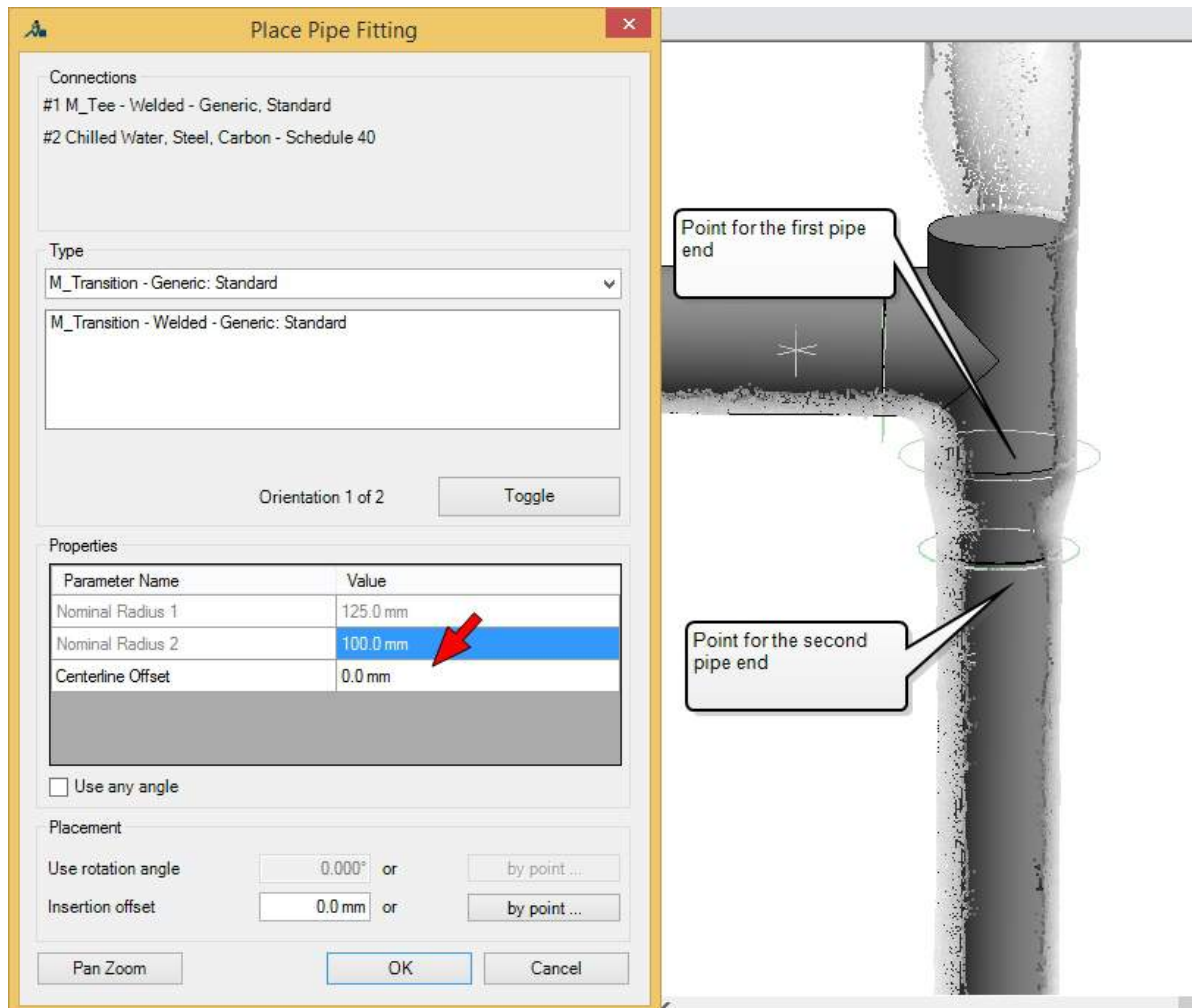
Select from the list the fitting type **"M_Transition - Welded - Generic: Standard"** and in the **"Properties"** group for the parameter **"Centreline offset"** enter the value 0.0 mm. This parameter controls the eccentricity of the component. In this example however a symmetrical component should be used.

Insert the component using **[OK]**, and then create the missing elbow. For this use the component **"M_Elbow log - Generic: Standard"**. Your results should look like this:

4.4.4 Alignment of the Pipe Run

When you take a close look at the components, you will notice small gaps between the individual parts. The reason for this, lies in the fact that the axes of the pipes, through the precise fitting of the pipes onto the point cloud, do not run coaxially and coplanar to each other. Even though we have inserted components with a precise 90° angle. To achieve a connected pipe run in Revit, the consistency constraints must be first met. To achieve this use the command → **Alignment - Align Pipe Runs** .
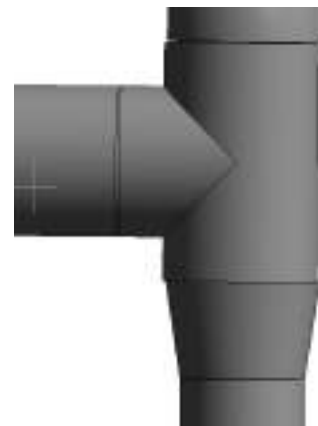
Start the command and click on any one of the components of the pipe run. A message is displayed, saying how many connected components for this pipe run have been selected:
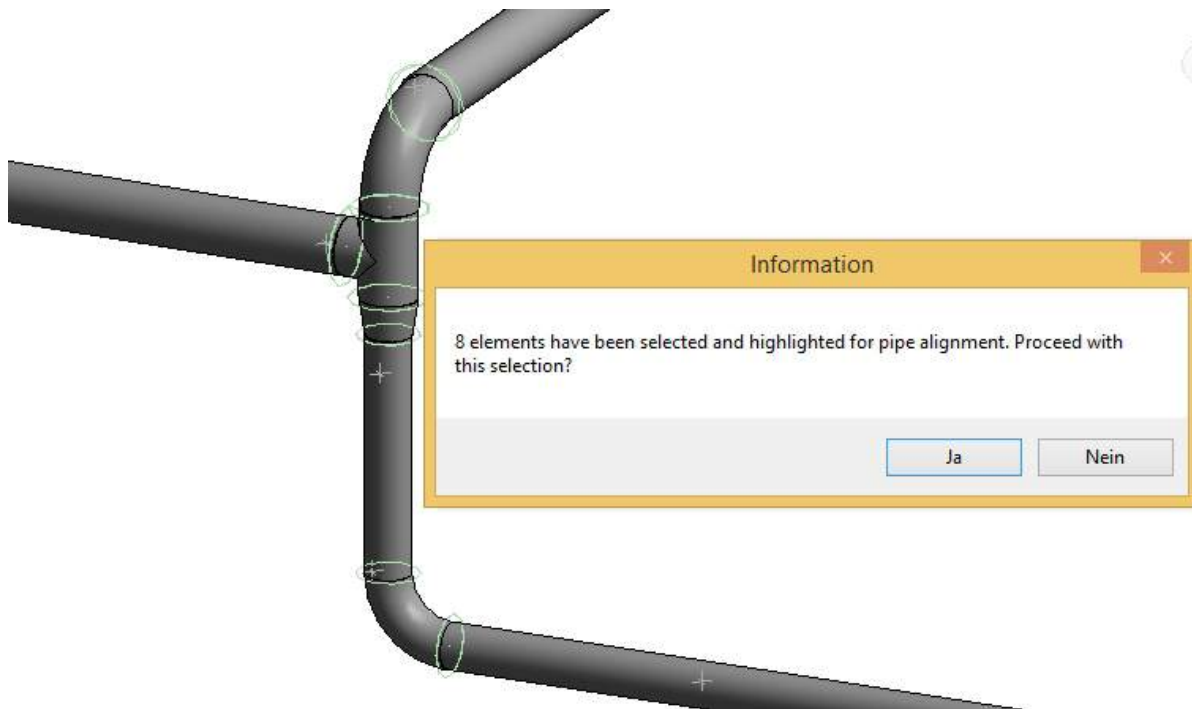
As-Built uses its own connections between the components to recognise a connected pipe run. Alongside of the message window these connections are also shown in the project as circles around the pipe joints. These As-Built connections are automatically created when using the command → **Object Tools - Placement of Pipe Fittings** .

Confirm the selection with **[Yes]**. The components are now aligned with each other and Revit compliant connected to each other. You can check this by showing the disconnects in the pipe run (Ribbon: Analyze → Panel: Check Systems → Show Disconnects (Pipe)):
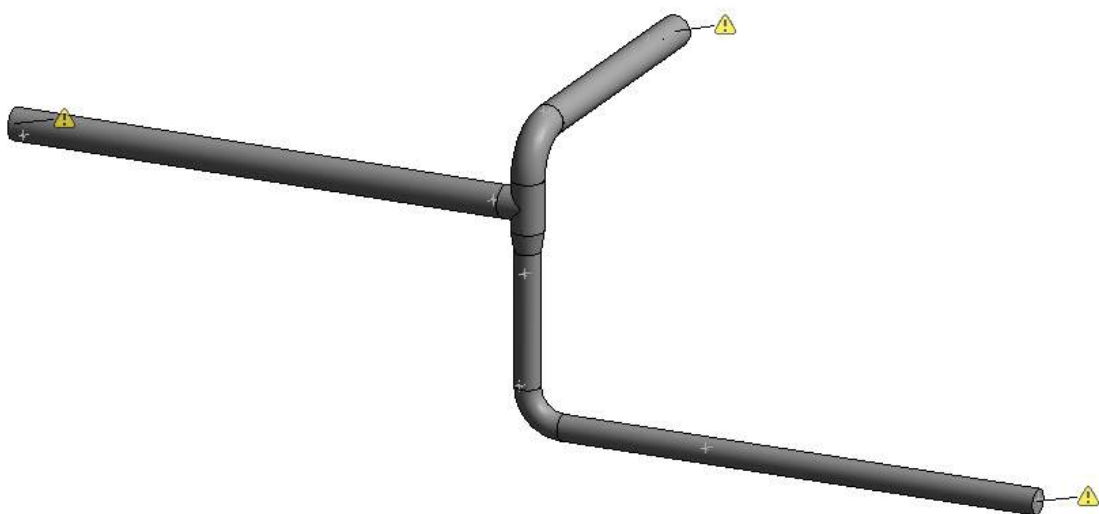
4.4.5 Insert pipe accessories

To complete the pipe run, both of the valves should now be inserted. Here the default Revit functions will first be used. Start the command **"Pipe Accessory"** 🔧 (Ribbon: Systems →

Panel: Plumbing & Piping). Select in the Revit Properties window the component **"Globe Valve - 200-250 mm"** and the type **"200 mm"** and click on the pipe on the position of the valve in the point cloud:
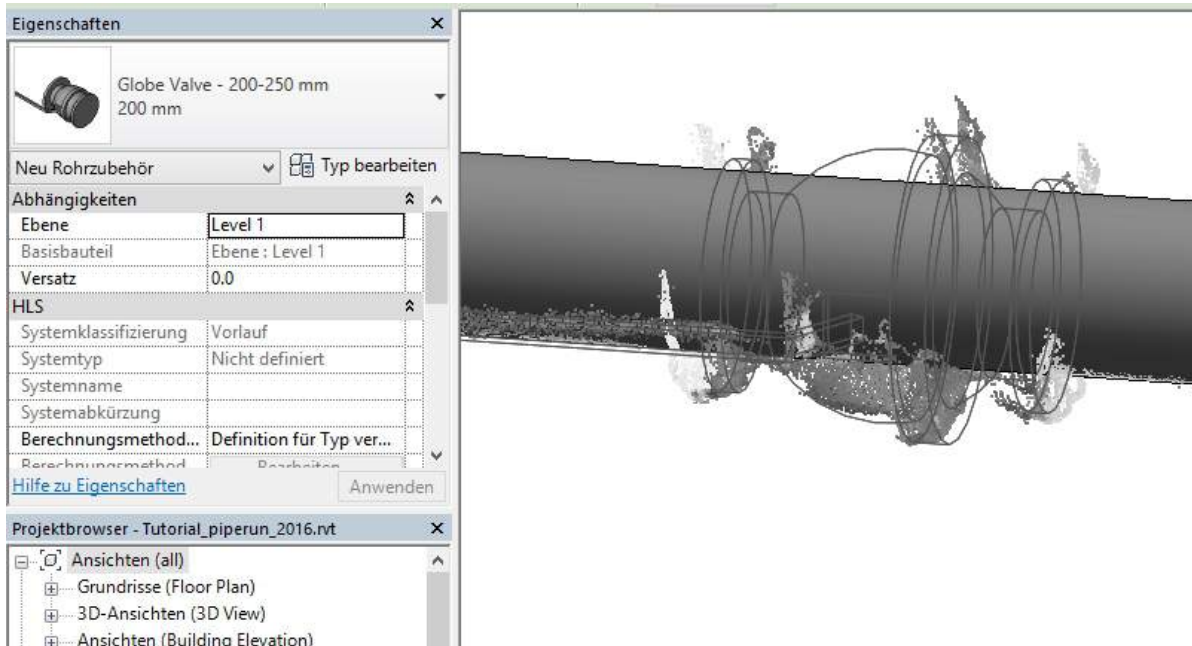
A As-Built construction point should be used for the precise alignment of the valve. Start the command → **VirtuSurv Tools - CPoint**, and in the point cloud pick a point on the flange of the valve.
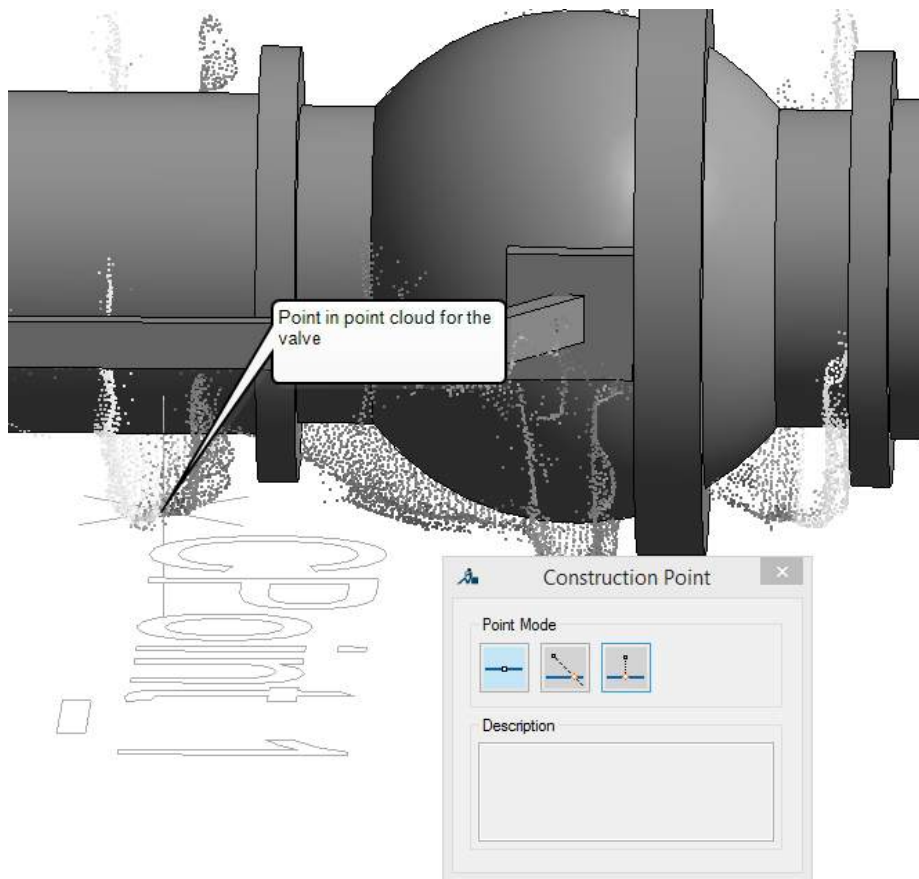
With the Revit command **"Align"** (Ribbon: Modify → Panel: Modify) you can now move the valve onto the position of the construction point. Start the command and click first on the vertical line of the construction point and then on the valve flange, that should be aligned:

Line as reference for the alignment

Valve face that should be aligned
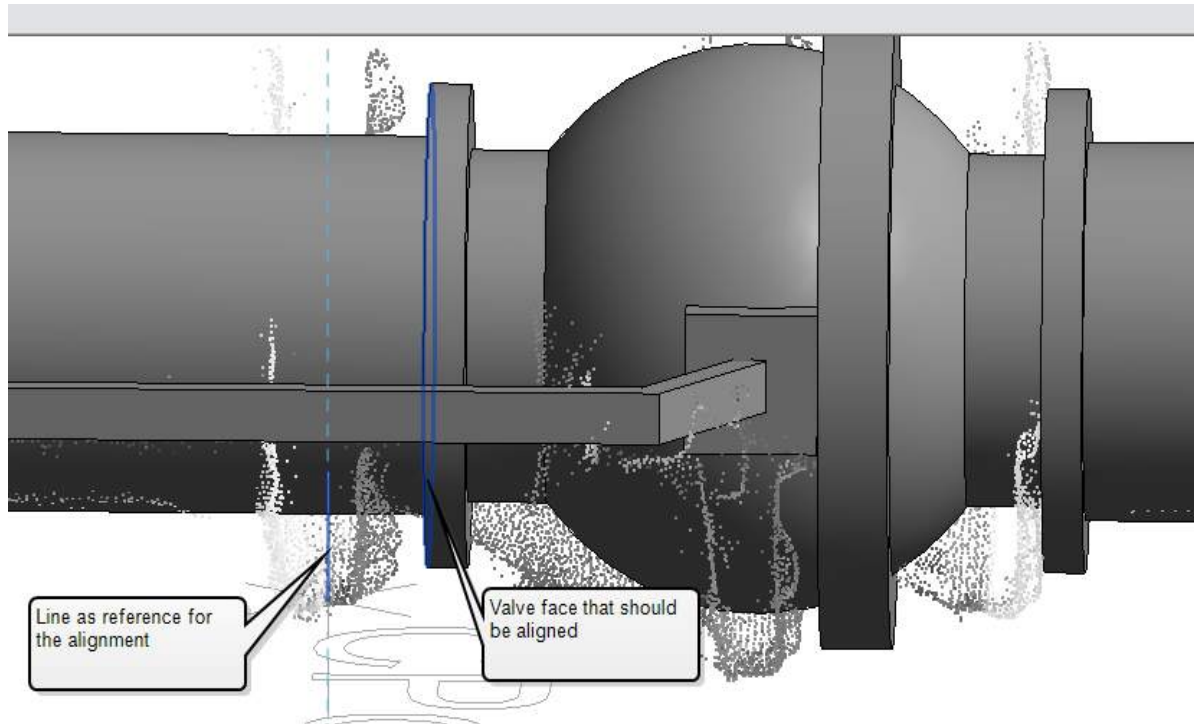
Repeat this procedure with the second valve. Here insert the component **"Globe Valve - 200-250 mm"** and the type **"250 mm"** .

The results of the tutorial should look like this:
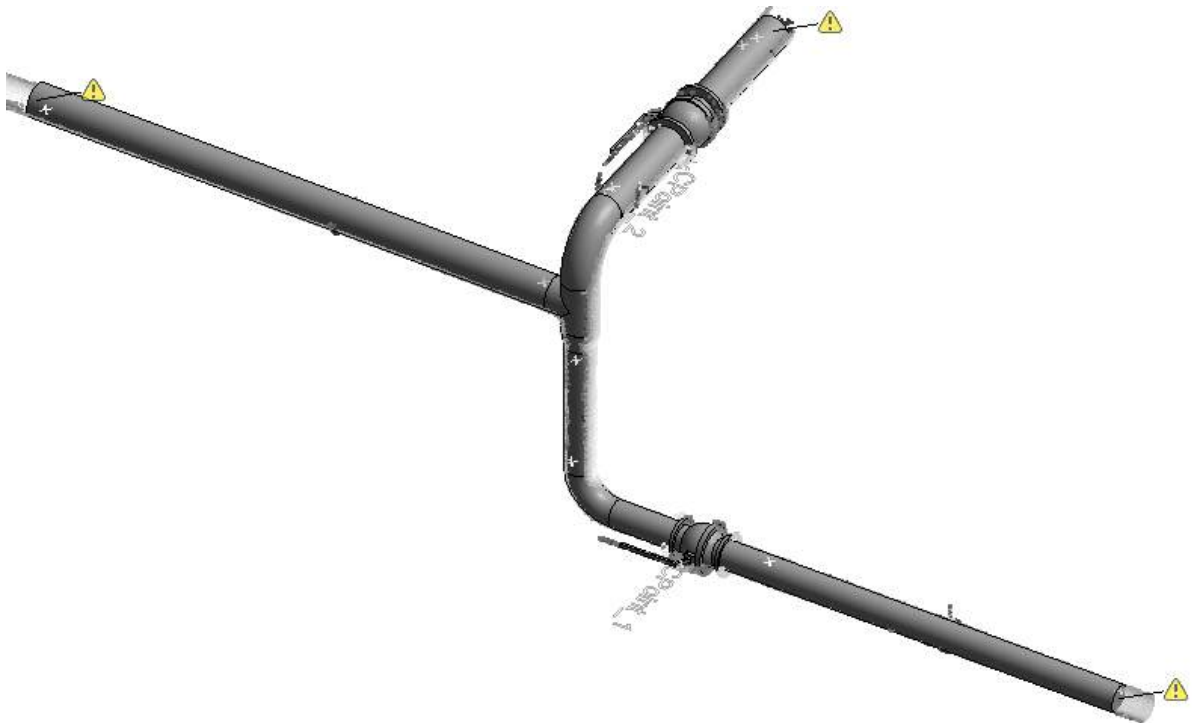
5 As-Built for Revit Command Reference

All of the commands that are relevant for managing and configuring your As-Built for Revit installation, are listed in the Command Reference. The commands are to be found on the ribbon. The description of every command starts with a small header block with the following structure:

Ribbon tab text

Button	Used in:		
	Project	Family	VirtuSurv

Ribbon tab text

You will find the command under this heading in the As-Built for Revit tab on the ribbon.

Button

You can call the command by clicking the symbol shown here in either the tool bar or the As-Built for Revit tab in the ribbon.

Used in

Here you will find the areas in which the command can be used:

Project	The command is available in the Revit project environment
Family	The command is available in the Revit Families Editor
VirtuSurv	The command is available in the Revit Toolbar for VirtuSurv

Object Tools - Prepare Point Cloud

Used in:

Project

This command is used for the pre-computation of a point cloud in the floor plan view for the fast use of the command → **Object Tools - Fit Wall**. Hereto is the point cloud, in the active plan view, from the visible top edge to the bottom edge, as well as the currently defined cropped region, converted into a dense point image with a threshold value. This converted image can then be inserted in the floor plan view and serves for better visualisation of the point cloud and also the acceleration of the "Fit Wall" command. If a point cloud exists in the project, the following dialogue box opens:
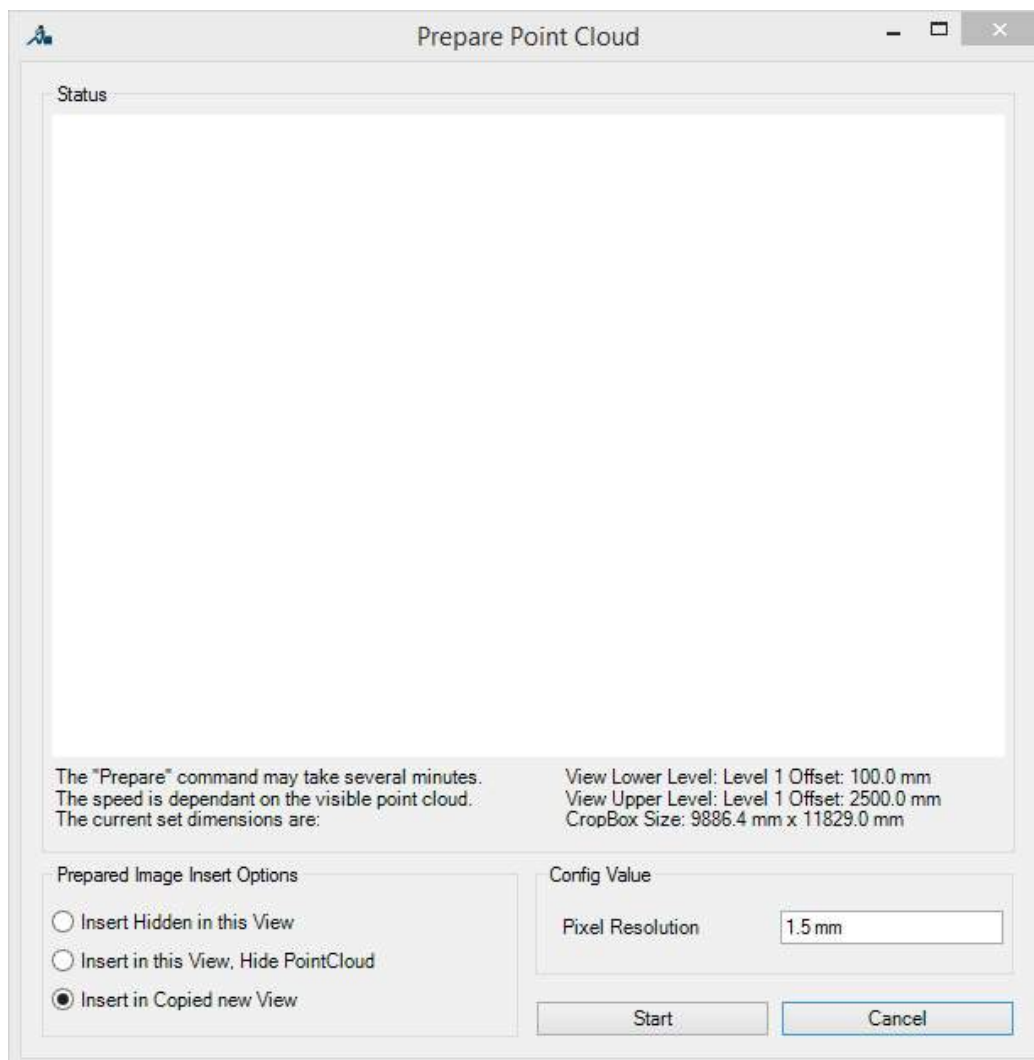
Beneath the currently white preview pane, the dimensions of the current point cloud region are displayed. Click on the button **[Start]**, to start the pre-rendering.
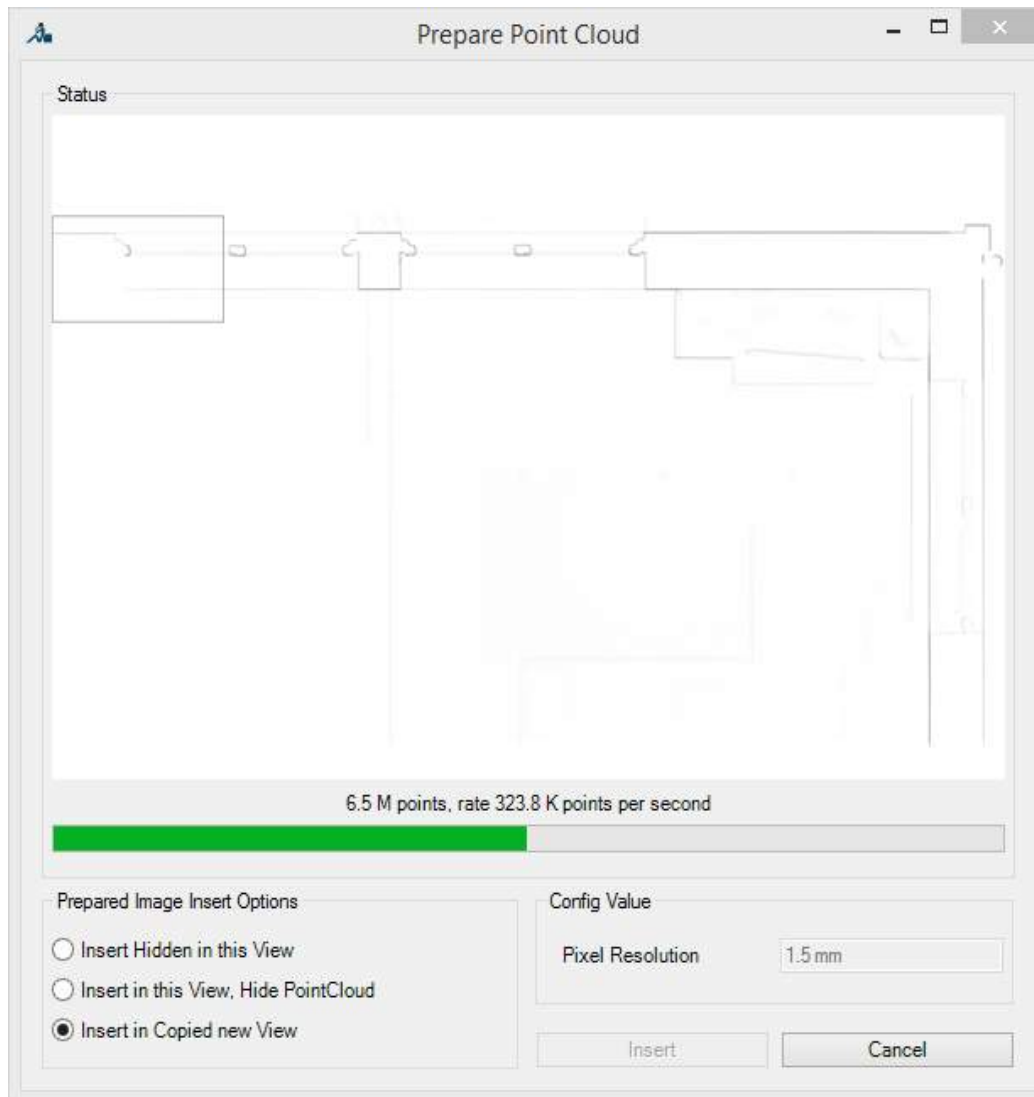
The pre-rendering begins. You will see the progress in the preview window, as well as the processing speed. The moving black border in the preview window indicates the position in the image that is currently being computed.

The "Contrast" parameter can be used to control the image intensity. The higher this value is, the more contrasty and darker appears the pre-computation. The default value is 1.
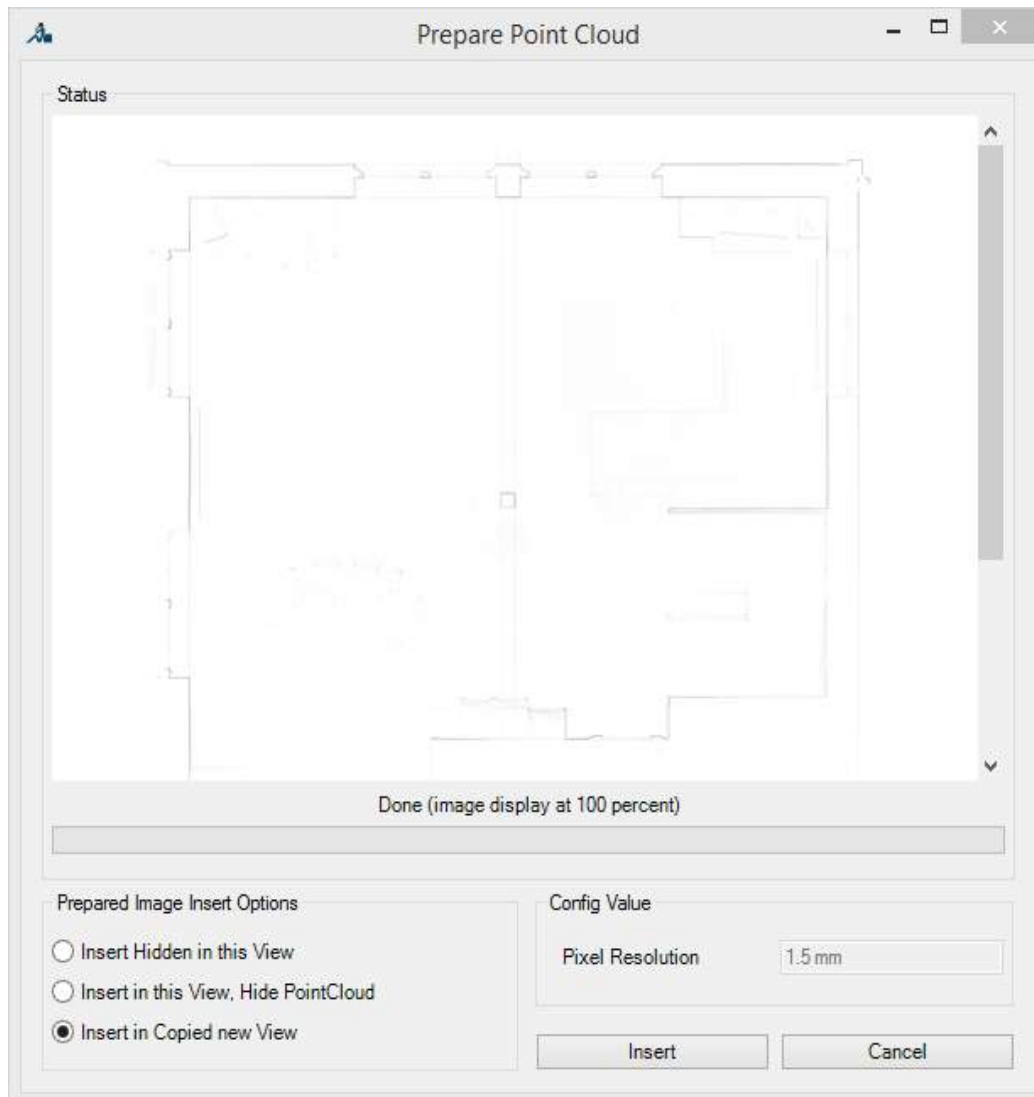
The pre-rendering is complete. You now have three options for inserting the rendered image into your project using the button **[Insert]**:

- "Insert Hidden in this View" - Inserts the rendered image hidden in the current view. It can also be used hidden for to the command "Fit Wall".

- "Insert in this View, Hide PointCloud" - Inserts the rendered image into this view, makes it visible and hides the point cloud.

- "Insert in Copied new View" - Inserts the rendered image visible into a duplicated view, in which the point cloud is hidden.

Using the Detail Level of the view you can also control the display of the rendered image from "Coarse" to "Medium" to "Fine", depending on the zoom level and in which level of contrast one wants to work.

Using the parameter "Pixel Resolution", one can if necessary set the resolution of the rendered image manually. The default value is set to 1.5 mm.

Object Tools - Ortho Image

| Used in: |
| Project |

The command "Ortho Image" creates a dense, continuous view of the point cloud in the current Revit view. It can be used in orthogonal 3D views, floor plans, sections, elevations and reflected ceiling plans. In plan views (floor plans, sections, elevations and reflected ceiling plans) such generated images can be directly inserted into the project, in their correct position and true to scale. Furthermore, such images can also be exported, then, for example, imported into the families editor with their correct size by using the command → **Connect Point Cloud - Insert Ortho Image**. Detailed family components can be modelled with the ortho images in the background.

When you start the command "Ortho Image", the following dialogue box will appear:
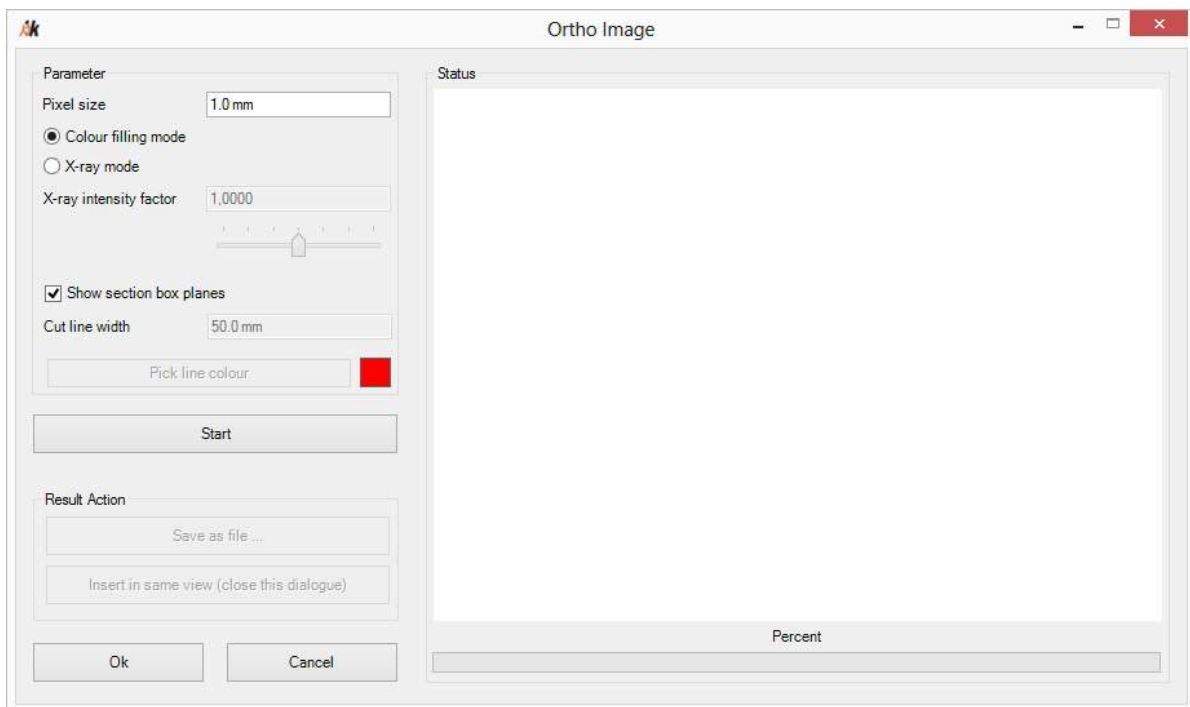
The first adjustable parameter is the **"Pixel size"**. It is the physical size of an image pixel in project units. It defines the resolution of the image and thus also the relevant required computation time. The default value for typical indoor surveys is 1 mm.

Then comes the drawing mode of the ortho image. Here the **"Colour filling mode"** or the **"X-ray mode"** can be chosen. The faster "X-ray mode" uses the same imaging method that is also used in the → **Object Tools - Prepare Point Cloud** command. Thereby the colour information of the point cloud is not taken into consideration, on the contrary a greyscale dense

point cloud image is created. With the parameter **"X-ray intensity factor"** the image intensity can be controlled. The higher this value is, the more contrasty and darker appears the X-ray image. The default value is 1.

In the following figure you can see four ortho images of the same window. On the left is the view "Colour filling mode", in the middle, on the right and below is X-ray mode in elevation, section and floor plan.
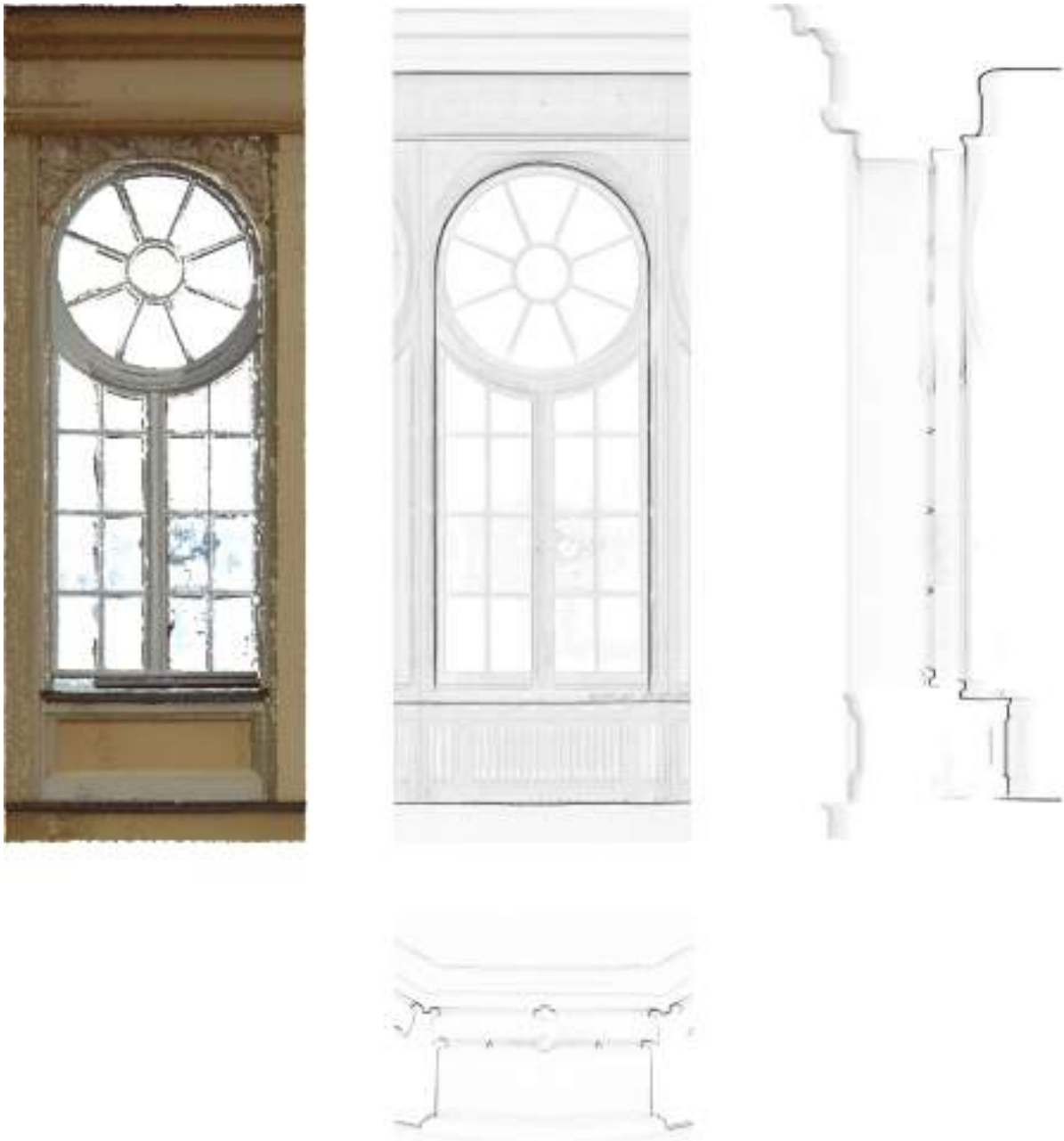
In the drawing mode "Colour filling mode" the point cloud will be seen as a continuous surface. Thereby, amongst other things, the individual point cloud points will be enlarged and re-

orientated, in relation to each other, and appear as a small discs in the image. It appears to the viewer to be a dense, natural surface with the coloured texture of the point cloud.
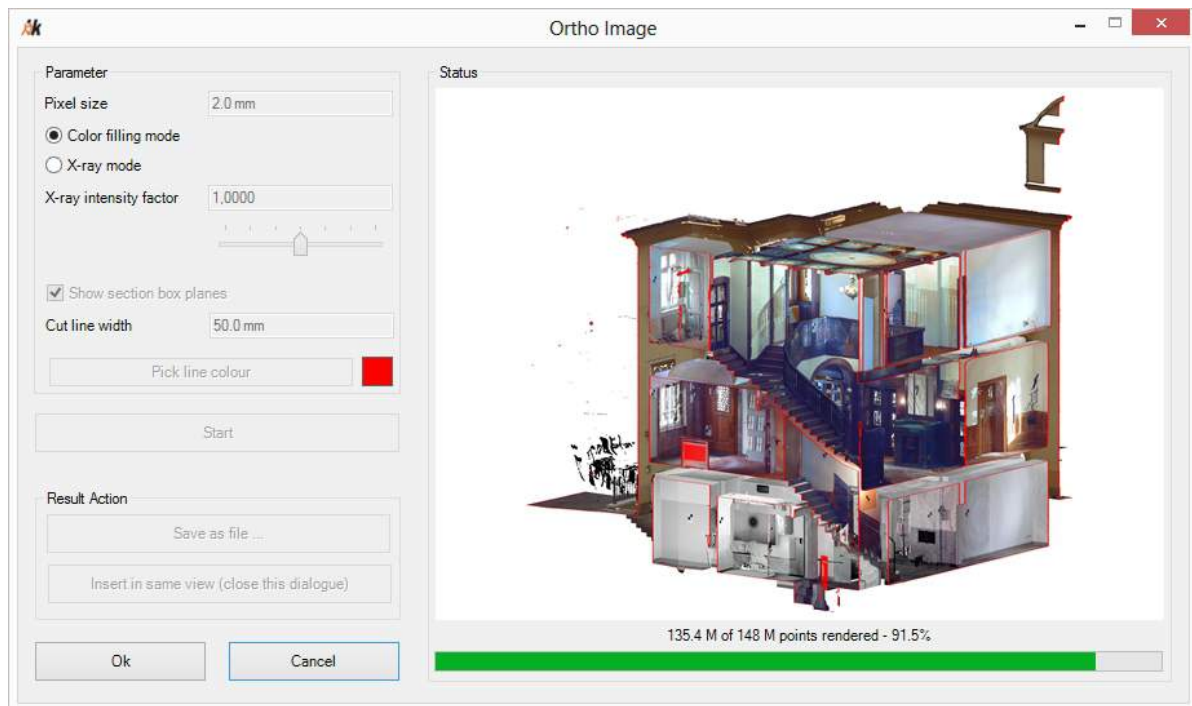
A further option is with the tick box **"Show section box planes"**, to switch them on or off. In a 3D view this allows the limits of the selected section box to be displayed in colour. In plan views, however, it is the front section plane that is coloured. The text box **"Cut line width"** determines the width of the strip that is to be coloured and the colour of the strip is defined using the button **[Pick line colour]**.

After you have set all of the parameters, you can click the **[Start]** button to start the image rendering. The **[Cancel]** button interrupts the rendering currently being carried out and the parameters can be reset. When the image rendering is complete, two more buttons are enabled. The button **[Save as file ...]** saves the rendered image, in full resolution, in an image format of your choice. Additionally, an *.xmlscale file is created, which contains size information and the scaling of the image. If you use the command in a planar view, you can with the button **[Insert in same view (close this dialog)]** insert the image directly into your current Revit view and then close the dialogue box.

Object Tools - Fit Wall

Used in:

Project

The command "Fit Wall" is used for the fast and precise fitting of vertical, straight walls in a point cloud. The command can be used in floor-plan views. Initially the following small dialogue box appears:

It serves to define the upper edge of the new walls that are to be drawn. The default preset value is the next level above the level of the active floor plan. It is possible to set the value of "Top Constraint " to "<manual>", in order to be able to define a wall height in the now enabled "Wall Height" text box. If you select the option "Fit only one wall face", the wall recognition assumes that there exists in the point cloud just observed date from just one side of the wall.

After you have clicked on the **[Start Wall Fitting]** button, the dialogue box closes and the actual user input begins. In the Revit status bar appears the message:

```
Click to enter start point for wall
fitting
```
Click the first point on the inside of the wall

```
Click to enter end point for wall fit-
ting
```
Click the second point on the inside of the wall

Both of the clicks should lie approximately on the centreline of the wall to be drawn.
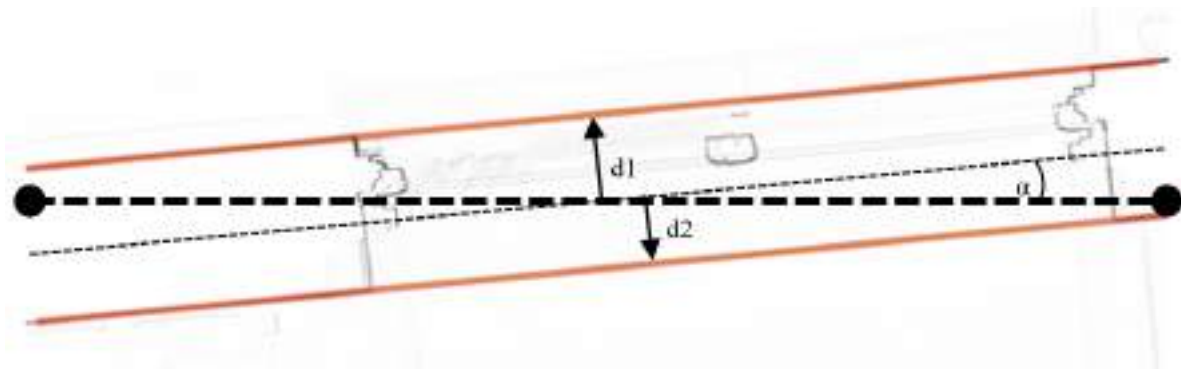
Based on the initial click-line, the correct rotation and extent of the wall will now be calculated. Based on the determined and appropriately rounded wall thickness, the compatible Revit wall types will be displayed for you in the dialogue box which appears.
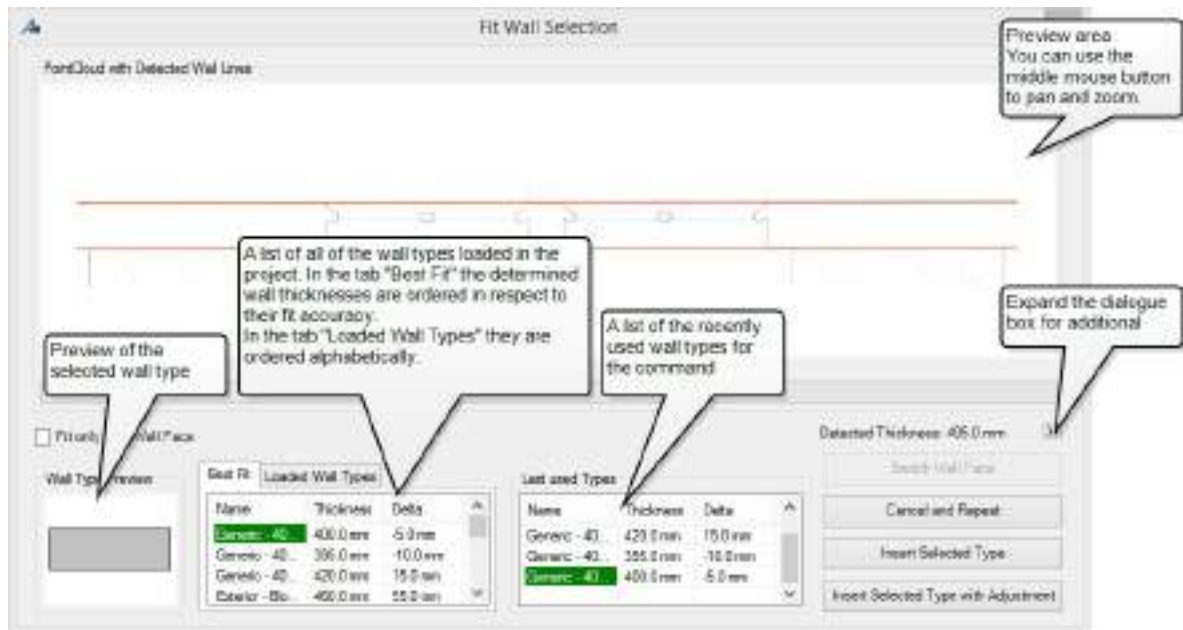
The best matching wall type is highlighted in the "Best Fit" list. You can also manually select a different wall type.

Using the button **[Insert Selected Type]** a wall of the selected wall type will be inserted. Should a wall type not be found with the determined wall thickness, you can automatically create a new wall type using the button **[Insert Selected Type with Adjustment]**. Thereby the currently selected wall type will be copied and its thickness adapted. With multilayered wall types, you can define which layer of the wall should have its thickness adapted. To do this click on the button **[>>]**. The dialogue box is expanded:
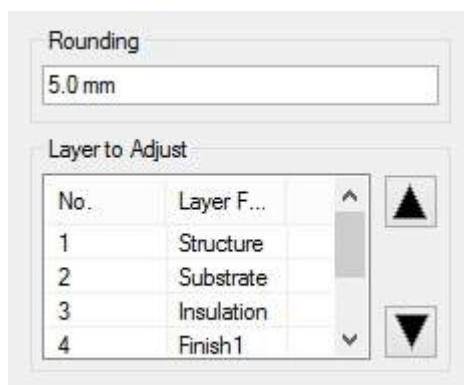
When you select a wall type for the action "Insert Selected Type with Adjustment", the layer of the wall type which is at the top of the "Layers to Adjust" list will have its thickness adjusted. By default, it is the "Structure" layer that is selected. Using the up and down arrow buttons priorities can be re-ordered. The value "Rounding" defines the accuracy to which the determined wall thickness should be rounded. Here the default is 5 mm.

In the case where the wall cannot be correctly determined, two different procedures are proposed. In the case where you have selected the wrong starting points, simply repeat the input of the initial click-line, whereby you need to select the **[Cancel and Repeat]** button.

The second possibility is to assist the algorithm by giving it some correction points. This can be necessary if there are a lot of interference objects that can influence determining the wall, or there are large openings in the wall (for example a large window). It allows you to define up to four correction points which, in the search for the local optimum, will define the start position of the wall. To do this click with the left mouse button in the preview area, in the vicinity of one of the two red lines. This will now be thicker highlighted. Now click a point in the preview window, through which the red line should approximately run. The correction points are shown as small blue crosses in the floor plan graphic, if applicable with an additional line to their associated wall side. After a correction point has been defined, a new search is started for the optimum local wall. Up to two correction points can be defined for each side of the wall.
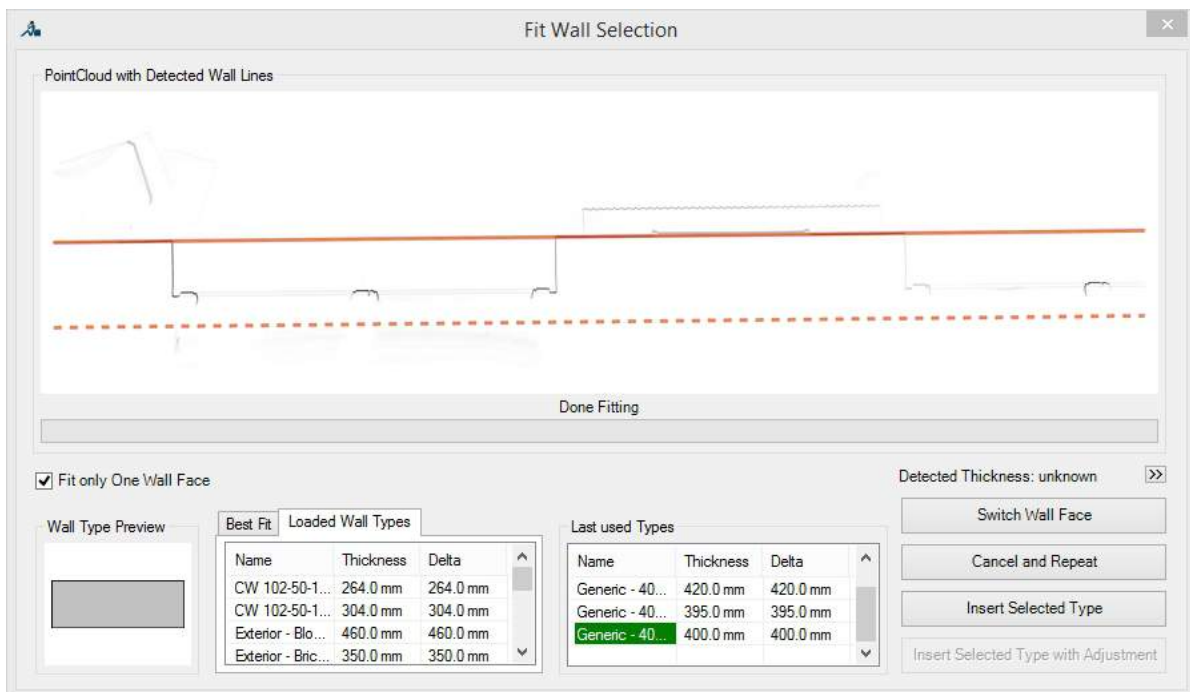
If you have only scanned the wall on one side, click the button "Fit only one wall face". Based on the line that has been found, the wall position is now determined. The red dashed line in the preview pane symbolises, using the selected wall thickness, the other wall face. The button "Switch wall face" changes the notional side of the non-scanned wall face. In this method of observation there are in total two possible correction points.
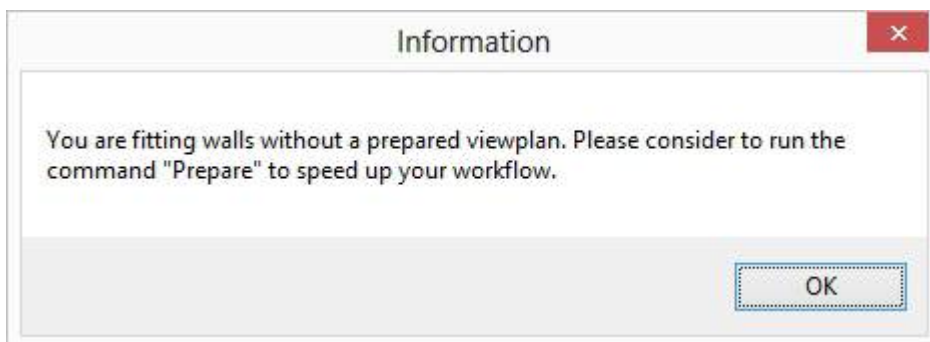
In the situation where during the command the above message box appears, you should initially run the pre-computation command → **Object Tools - Prepare Point Cloud** for your floor plan. The pre-computation significantly accelerates the fitting of the walls.

Object Tools - Toposurface

Used in:

Project

With this command you can create a Revit toposurface from multiple model lines and construction points. To achieve this, proceed as follows:

- Create an unlimited number of model lines or construction points in the point cloud using the commands → **VirtuSurv Tools - Model Line** and → **VirtuSurv Tools - CPoint**

- Then run the toposurface command. In the Revit status bar appears the message:

  ```
  Select multiple construction points
  or model lines
  ```
  Select them as shown in the following figure.

- Confirm the selection in the Revit Options Bar with **[Finish]**.

As result you can see the new created toposurface.

In combination with the commands for surface analysis you can create very quickly and very detailed terrain models. Please proceed as follows:

■ First create some auxiliary geometry (e.g. a floor or a project frame) in the size of the terrain to be created

■ Compute the deviation of this auxiliary geometry to the point cloud using the command → **Surface Analysis Tools - Calculate**. Here select the surface that is facing the point cloud.

■ Create model lines based on the computed deviation by using the command → **Surface Analysis Tools - Export Results**. Here it is sufficient if the profile lines are only created in one direction (U or V). It is recommended to use the option "Minimum" as the export value.

■ Use the command "Toposurface" on the newly created profile lines. If necessary you can beforehand carry out the manual creation of additional construction points or model lines, or corrections to the run of the profile lines.

Object Tools - Pipe Fitting

Used in:

Project

The command fits a pipe into the point cloud.

After starting the command you are prompted to select the reference plane and the type of system of the desired pipe.
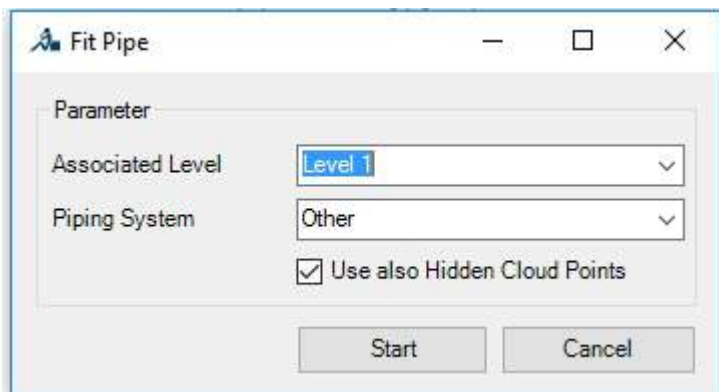
As soon as you have selected the level and the pipe system, confirm by clicking **[Start]**. You are now prompted to pick two points in the point cloud along the pipe.
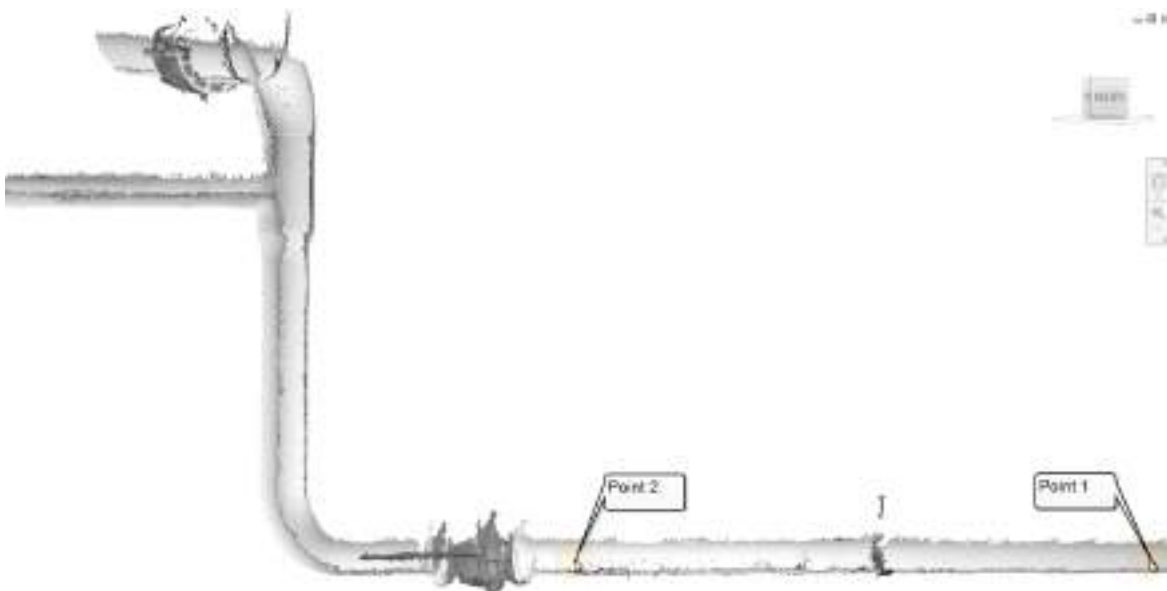
`Pick a point in the point cloud`	*Pick the first point on the scanned pipe.*
`Pick a point in the point cloud`	*Pick the second point on the scanned*

pipe.

You can either pick the point cloud points direct in Revit or also from a planar view in VirtuSurv.

The option "Use hidden point cloud points" is particularly important when using planar views of the point cloud from VirtuSurv. For detection purposes it is not only the visible point cloud points that are used, but all of the points in the point clouds that are loaded in Revit.

After you have picked both of the points a dialogue box is displayed, where the recognised pipe geometry is shown and where you are prompted to select which pipe that you definitely want to insert.
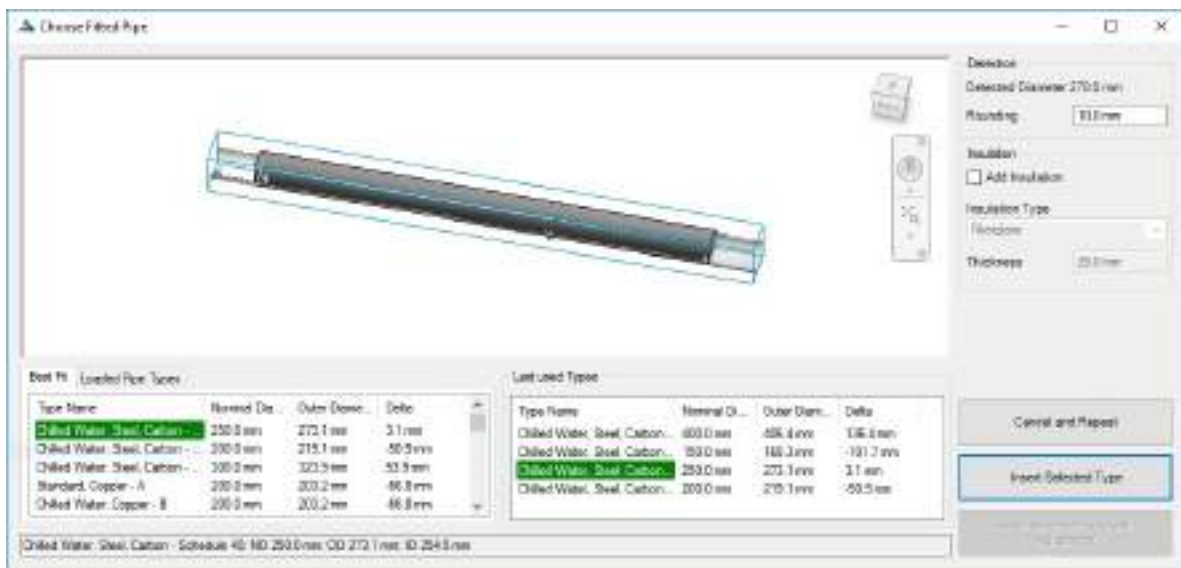
You can select the pipe that you want to insert into the project from three lists [Best fit], [Loaded pipe types] and [Recently used types]. In the preview you can see the geometry of the selected pipe together with the surrounding point cloud region. The pipe is simultaneously displayed as a preview in the project.

You can more precisely define the pipe that is to be inserted through dialogue elements:

Dialogue Element	Explanation
[Best fit] list	Here are all of the pipes displayed that are in your project together with their predefined segments and diameters. This list is sorted in respect to the deviation of the actually recognised pipe diameter to that of the selected pipe. Entries at the top of the list have a relatively better fit to the point cloud as those further down in the list.
[Loaded pipe types] list	In the same way as in the **[Best fit]** list, here all of the pipes are shown that are defined in the project. The pipes are sorted relative

Dialogue Element	Explanation
	to their nominal diameter.
[Recently used types] list	In this list all of the pipes are shown that you have most recently inserted.
[Nominal diameter] column	In this column of the three lists for selecting a pipe, it is the nominal diameter of each of the pipes that is shown.
[External diameter] column	In this column of the three lists for selecting a pipe, it is the external diameter of each of the pipes that is shown.
[Delta] column	In this column of the three lists for selecting a pipe, it is the variance of each of the pipes to the pipe geometry that has been detected in the point cloud that is shown.
[Detected diameter]	It is the detected diameter of the pipe surface in the point cloud that is shown.
[Rounding] text box	This value shows the range to which the detected pipe surface should be rounded.
[Insert isolation] option	If this option has been selected, then the pipe will have insulation around it.
[Insulation type] select	With this selection you can choose the type of insulation.
[Thickness] field	In this field you can enter the thickness of the insulation. The difference of the external diameter to the detected diameter of the pipe surface is shown in the [Delta] column and it changes to this value.

After you have made your choice, you can insert the selected pipe into the project by selecting **[Insert selected type]**. Should a desired pipe segment with the desired size not appear in any of the pick lists, then you can define a new pipe size for a selected pipe segment by selecting **[Insert fitted selected type]**. In this case a dialogue box is displayed that prompts you to define the new pipe size for the pipe segment.
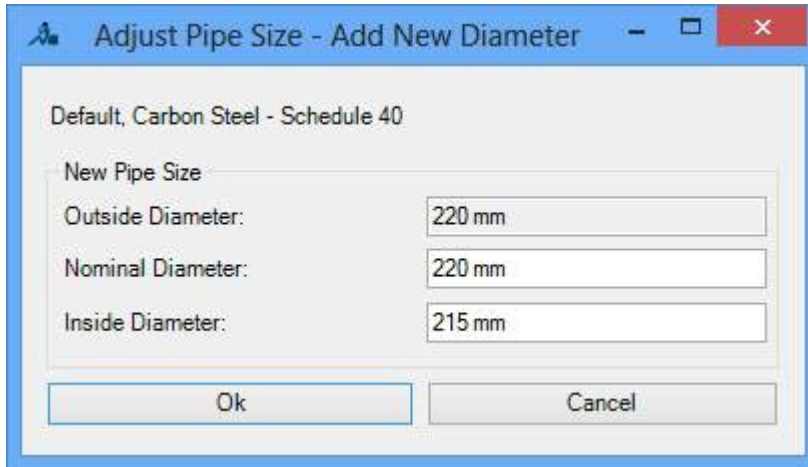
After confirming with **[Ok]** the pipe will be inserted with the new pipe size.

In a few cases it can happen that the pipe geometry is falsely detected. You can see this in the preview, when for example, the detected pipe axis does not lay central to the scanned pipe surface. In such cases you can by selecting **[Cancel and repeat]** repeat the detection of the pipe by re-picking two points on the surface of the pipe.

As soon as you have inserted a pipe, you will be prompted again to pick two points on a new pipe in your pipe run. If you want to end the pipe detection procedure then press the [ESC] key.

Object Tools - Placement of Pipe Fittings

Used in:

Project

After you have fitted all of the necessary pipes of a pipe run in your project, you can with this command insert pipe accessories in between the detected pipes.

By the first start of this command in a new project, As-Built for Revit first carries out an analysis of all of the loaded pipe accessories families. The analysis determines the variable parameters in the families, which can be subsequently fitted with the command "Placement of Pipe Fittings". The results of the analysis are saved in the project, so that by a further start of this command this step can be omitted. Should you have in the meantime loaded further families into the project, then the analysis will only be carried out for these new families.

After starting the command you will be prompted to select the pipe ends between which you want to place the pipe fitting:

`Pick the first pipe end to place the pipe fitting`	*Then select the first pipe end*
`Pick the next pipe end or end the input by pressing the [ESC] key`	*Select here up to two further pipe ends. Exit the selection with [ESC].*
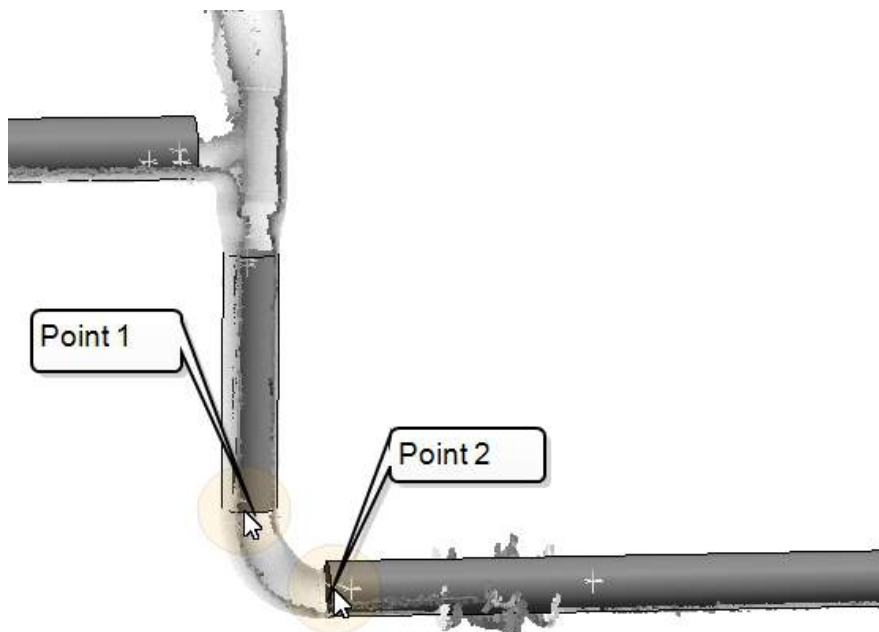
As soon as you have selected all of the pipe ends, a dialogue box is displayed, in which you are supplied with all of the suitable fittings that can be fitted between the pipe ends:
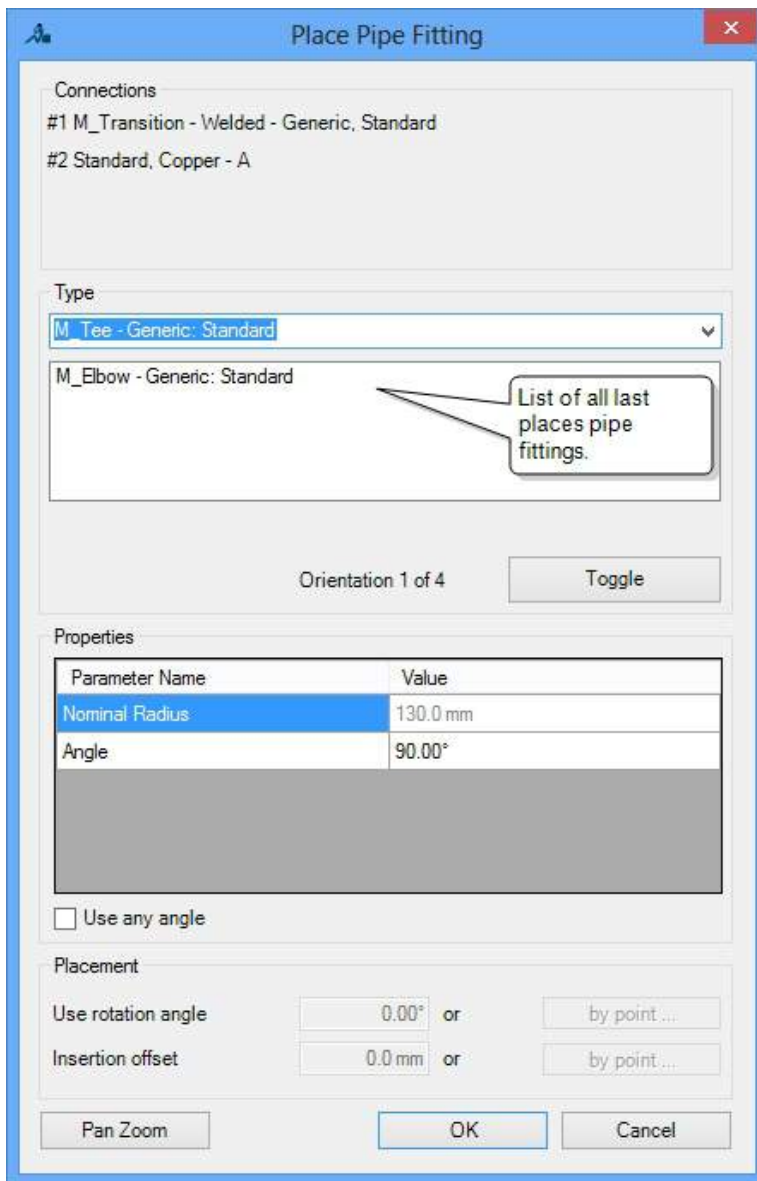
Through dialogue elements you can now select the desired pipe fitting and define its settings:

Dialogue Element	Explanation
[Pipe ends] group	Here are listed the names of the pipe types and pipe segments of the selected pipe ends. Should a pipe end of a pipe fitting be selected, then the respective names of the families and types will be shown.
[Type] select	The pick list shows all of the pipe accessories in the current project that would fit on the selected joints, with respect to the nominal diameter. The selected type will be inserted into the

Dialogue Element	Explanation
	project by confirming with [OK].
List of the recently used pipe fitting types	In this list all of the pipe fitting types are shown that you have recently used.
[Change] button	By clicking this button the fitted position of the component will be changed. An appropriate preview can be seen in the current project view.
Orientation ... of ...	Here are shown how many possible fitted positions the selected pipe fitting has for the selected pipe ends and which there from is currently selected.
[Properties] list	In this list are shown all of the exemplar parameters that are relevant to the geometry of the component. If the value is greyed out, then this value cannot be manually changed. If the value is black then it can be changed by editing the field.
[Use any angle] option	If this option has not been selected then the angle in the properties list will always be rounded to one of the Revit default angles (90°, 60°, 45°, 30°, 22.5°, 11.25°). If the option has been selected you can set here any angle you like.
[Rotation angle] text box	You can select with this value the rotation of the component around its own axis. If the rotation has already been defined by the selected pipe joint, then this option is disabled.
[From points ...] define rotation angle	By clicking this button you can define the rotation angle by picking a point in the point cloud, to which a pipe joint of the component should point (also see: → **[From points ...] define rotation angle**).
[Insertion offset] text box	With this value the insertion offset can be defined of the component to first selected pipe joint. If the offset has already been defined by the selected pipe joint, then this option is disabled.
[From points ...] define insertion offset	By clicking this button you can define the insertion offset by picking a point in the point cloud (also see: → **[From points ...] define insertion offset**).
[Pan Zoom] button	By clicking this button you can temporarily examine the selected component in the current view. The mode is ended by pressing [Esc] and the dialogue box again displayed.

During your selection, the chosen component with its chosen properties will be displayed in the drawing:
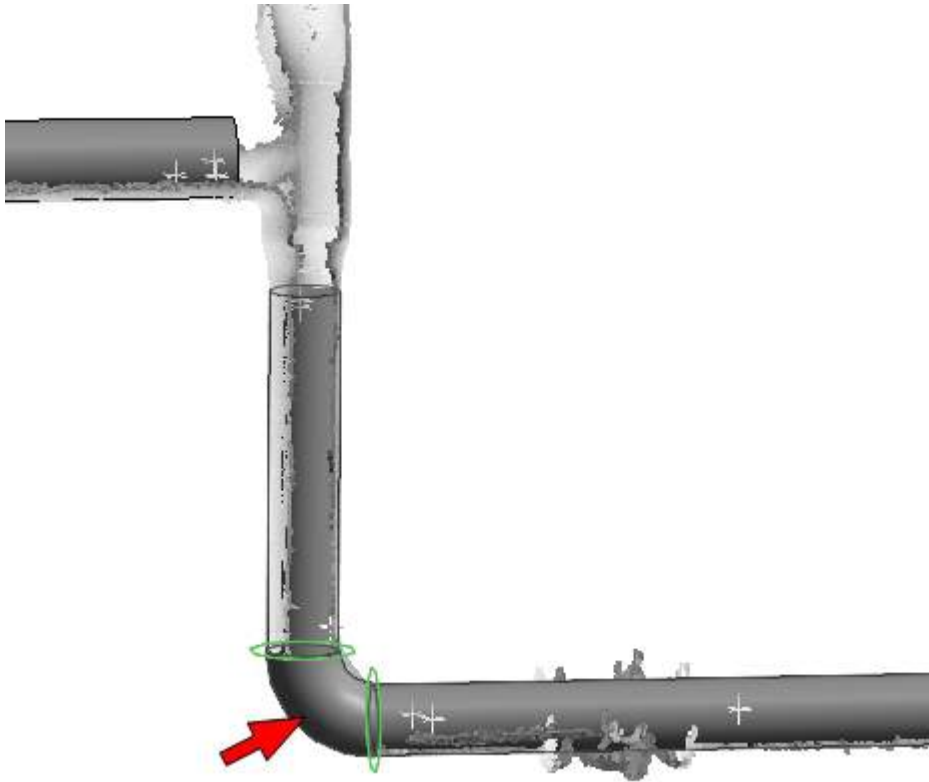
As soon as you have selected the correct pipe fitting and its associated properties, confirm your choice with **[OK]**. The component will then be inserted into your project.
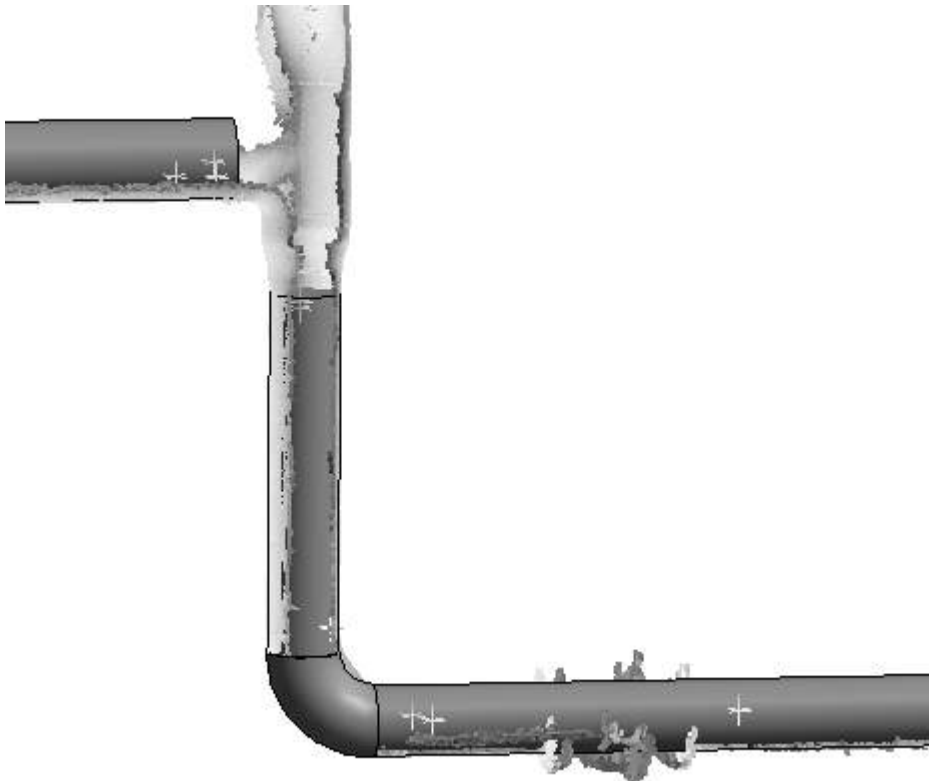
In a few cases it can happen that the pipe system of the component cannot be automatically determined. In this event you are prompted to manually select them:
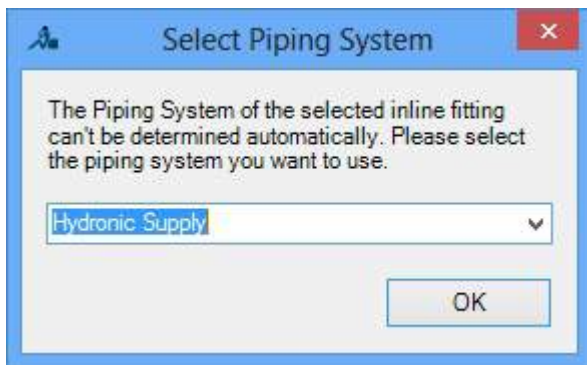
After first selecting of the pipe system will the component be inserted into the project.

[From points ...] define rotation angle

If you have chosen the button **[From points ...]** to select the rotation angle, then you will be prompted to pick a point in the point cloud, to which the component branch should point.

Select a point in the point cloud to set the rotation

Select a point in the point cloud to define the direction to which the branch of the component should be oriented.
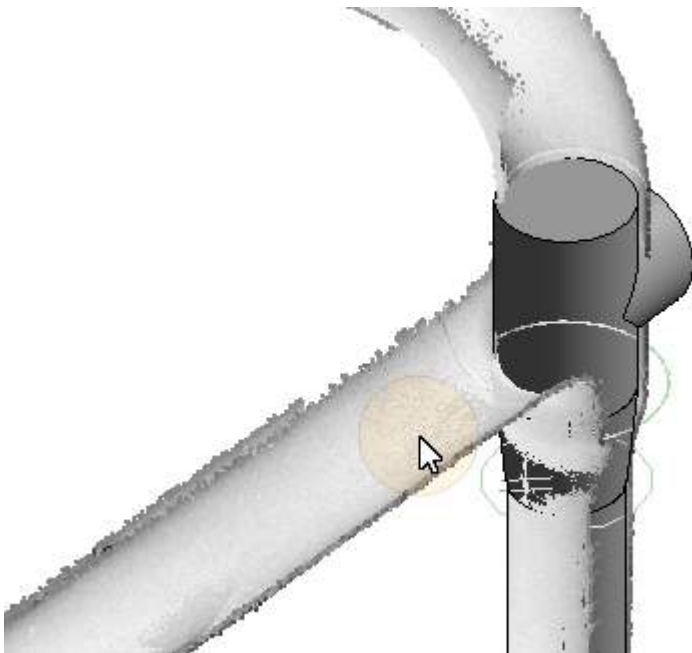
As soon as you have picked a point, the component branch points in this direction:
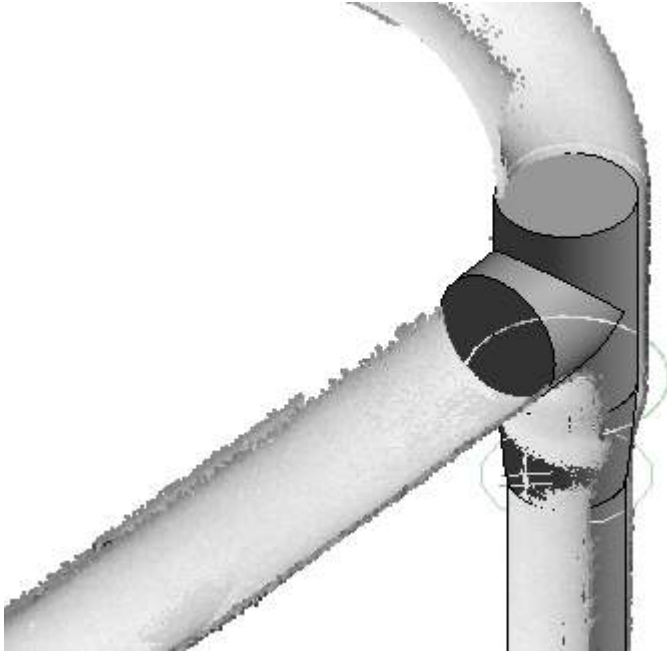
[From points ...] define insertion offset

If you have chosen the button [From points ...] to select the insertion offset, then you will be prompted to pick a point in the point cloud, to which the component should be offset in relation to the first selected pipe joint:

```
Select a point in the
point cloud to set the
offset
```

Select a point in the point cloud to define the insertion offset in relation to the first selected pipe joint.

Thereafter the pipe fitting will be offset at this position from the first selected pipe joint.
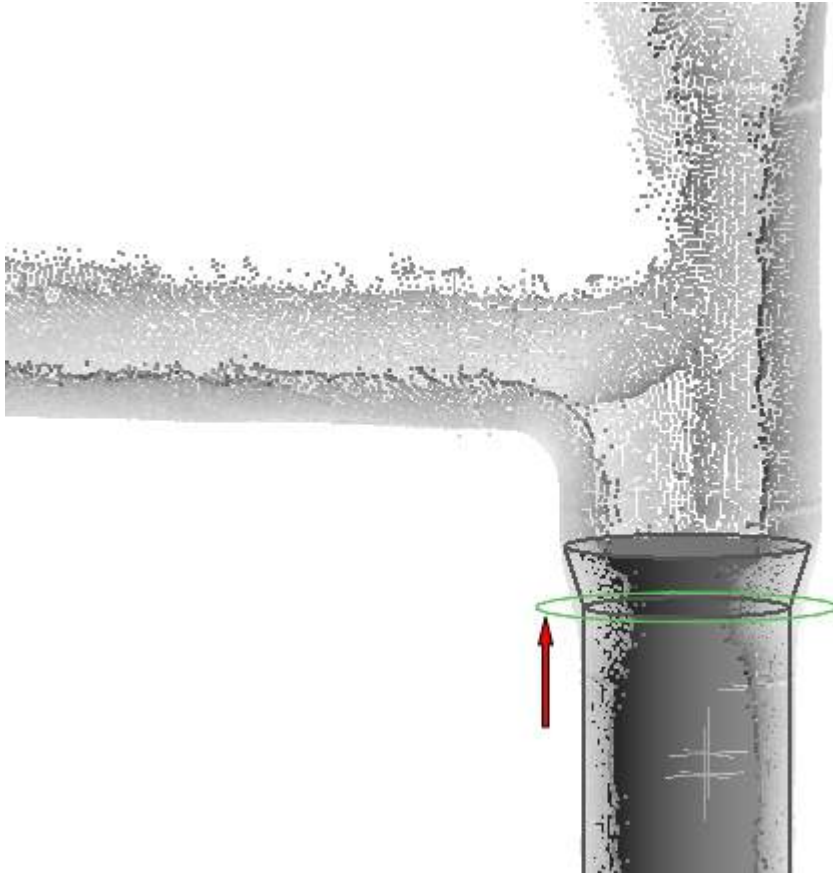
Object Tools - Fit Structural Elements

Used in:

Project

The command fits a structural element (column or beam) into the point cloud.

After calling the command the following dialogue box is displayed:
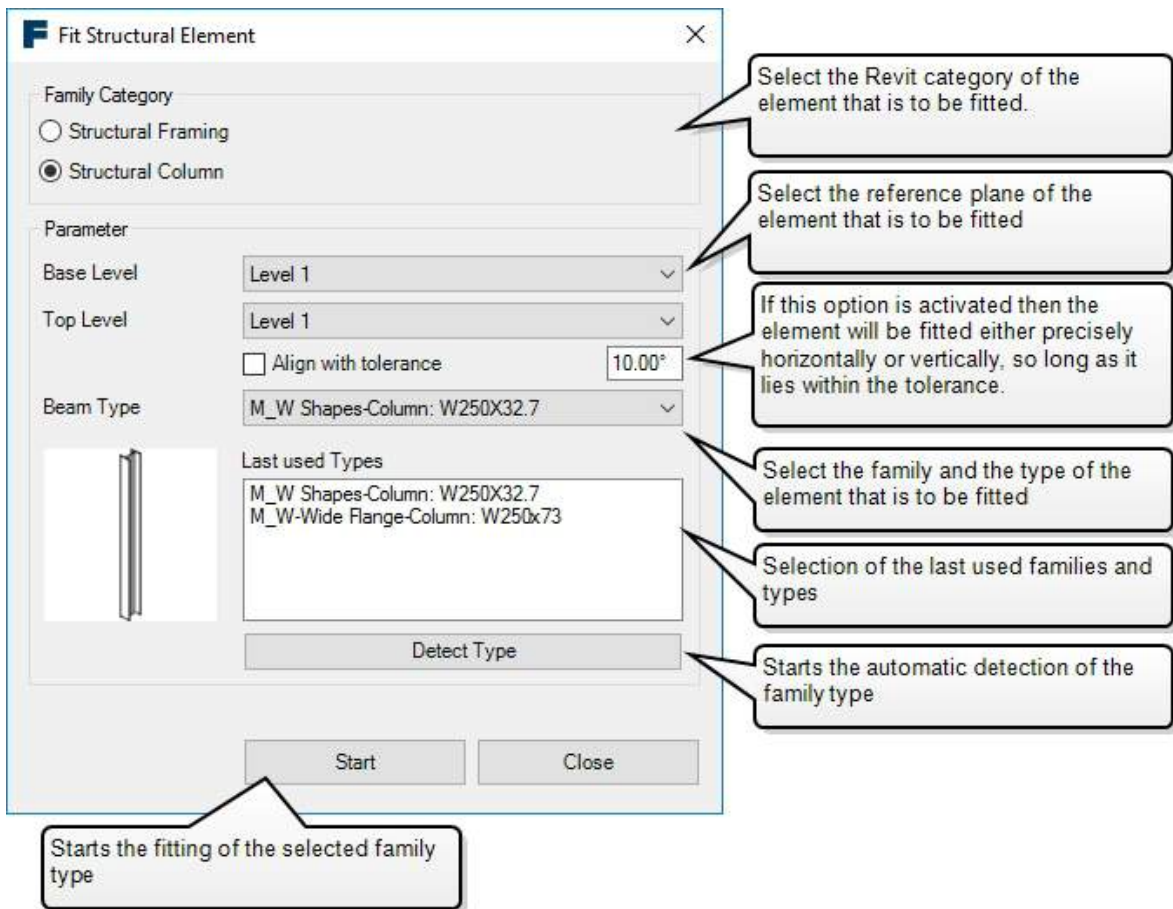
Select the family type and the insertion options and activate it with **[Start]**. You are prompted to pick two points on the beam in the point cloud.

| Pick a point in the point cloud | *Pick the first point on the scanned beam.* |
| Pick a point in the point cloud | *Pick the second point on the scanned beam.* |

After selecting the second point, an instance of the selected type will be fitted in the point cloud:
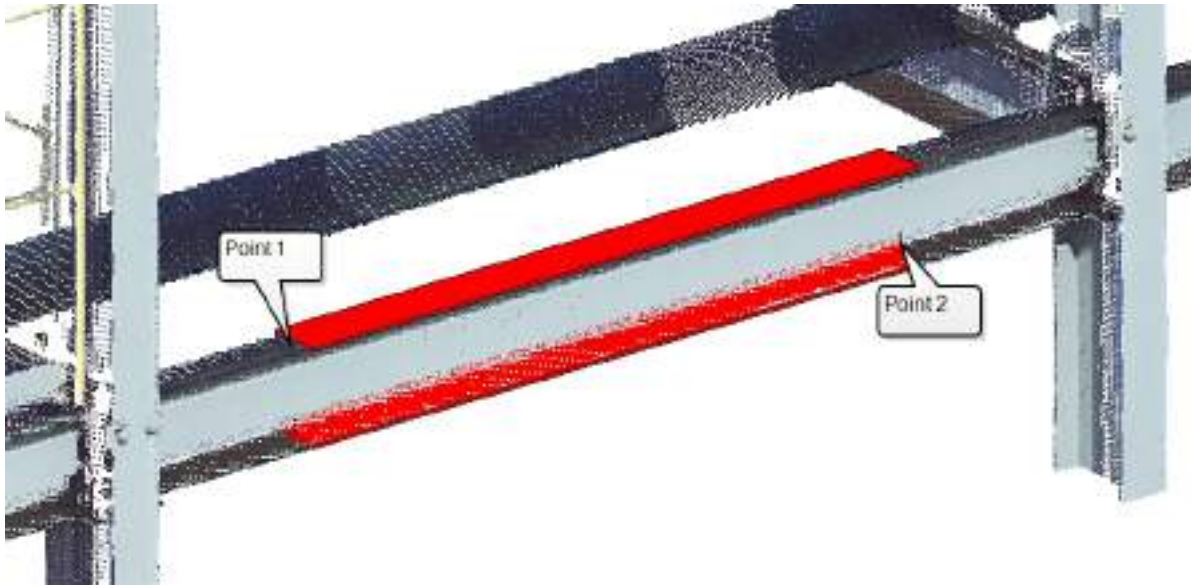
After inserting the element you can continue directly with the fitting of further elements of the same type.

When the type of the element is not known you can automatically determine it by activating the option **[Identify type]**. You are prompted to pick two points on the beam in the point cloud.

`Select two points on a flat surface of the structural element`	*Consecutively select two points on a flat surface of the scanned beam.*

Select two points that both lay on a face that is clearly identifiable in the point cloud. There should only be the face of the beam between the two points (no branches or other objects).

After the first start of type identification, the profiles of all of the families and types of the "Structural Framing" and "Structural Columns" categories will be analysed. This operation may take a few minutes. With repeated invocation of type identification this step is not required.

The following dialogue box appears:

Dialogue Element	Explanation
Preview pane	In the preview pane the currently selected type is visualised in a point cloud slice. The face initially selected by picking the two points will be displayed slightly thicker and bright blue. The point cloud noise used in the project is displayed white around the object. The noise corridor should be just wide enough so that all significant points lay within it. By keeping the left mouse button pressed you can delete in the preview pane any point cloud points that interfere with the pattern recognition. In this situation the background colour changes from green to red.
[Best fit] list	Here are displayed all of the family types of the previously selected categories from your project. Entries at the top of the list have a better fit to the point cloud than those further down in the list.
[Recently used types] list	In this list all of the types are shown that you have most recently inserted.

Dialogue Element	Explanation
Slice	The pattern recognition is carried out on a slice between both of the two picked points. The position of the slice can be moved along the beam axis by using the slider. This can be necessary when the slice to be analysed has too many interference points or too few points because of shadowing. When moving the slice the background colour of the preview pane changes from green to red.
[Recalculate] button	If something has been erased or the slice has been moved in the preview pane, the pattern recognition can be restarted from here.
Point Cloud Parameter	The **Noise** parameter is displayed as information. By clicking the button **[Modify]** the settings dialogue box is displayed to change the values (→ **As-Built - Settings**). The command → **Point Cloud- Auto Measure Parameter** is started by clicking **[Auto. Measure]**, to automatically determine the point cloud parameter.
[Pan Zoom] Button	By clicking this button you can temporarily examine the selected component in the active view. The mode is ended by pressing [Esc] and the dialogue box again displayed.
[Cancel] Button	Ends the type identification.
[Cancel and Repeat] Button	Closes the dialogue box and prompts for the reselection of two points.
[Insert Selected Type ...] Button	Inserts an instance of the selected type into the project.

After completing the type identification by the insertion of an instance of the selected type, you can continue directly with the fitting of further instances of the same type. To end the command press the [ESC] key.

Object Tools - Deform Shape

Used in:

Project

With this command you can deform a Revit floor or flat roof to model uneven surface characteristics. The command inserts high- points and low- points in the floor and roof respectively. Unlike the standard Revit form editing commands the command "Deform Shape" uses existing model lines or construction points for the changing the shape. To achieve this, proceed as follows:

- Using the standard Revit commands create a flat floor or flat roof.

- Create as many model lines or construction points as you like in the point cloud with the help of the commands → **VirtuSurv Tools - Model Line** and → **VirtuSurv Tools - CPoint**

- Now run the command "Deform Shape". In the Revit status bar appears the message:

```
Please select a roof or floor
```
Select the appropriate object in your project

```
Select multiple construction points
or model lines
```
Select them as shown in the following figure.

- Confirm the selection in the Revit Options Bar with **[Finish]**.

- High-points or low-points will be inserted into the floor or roof on the construction points or ends of the model lines.

The following example demonstrates the procedure:

1. Select a flat floor (blue)

2. Select the model lines and construction points (red)
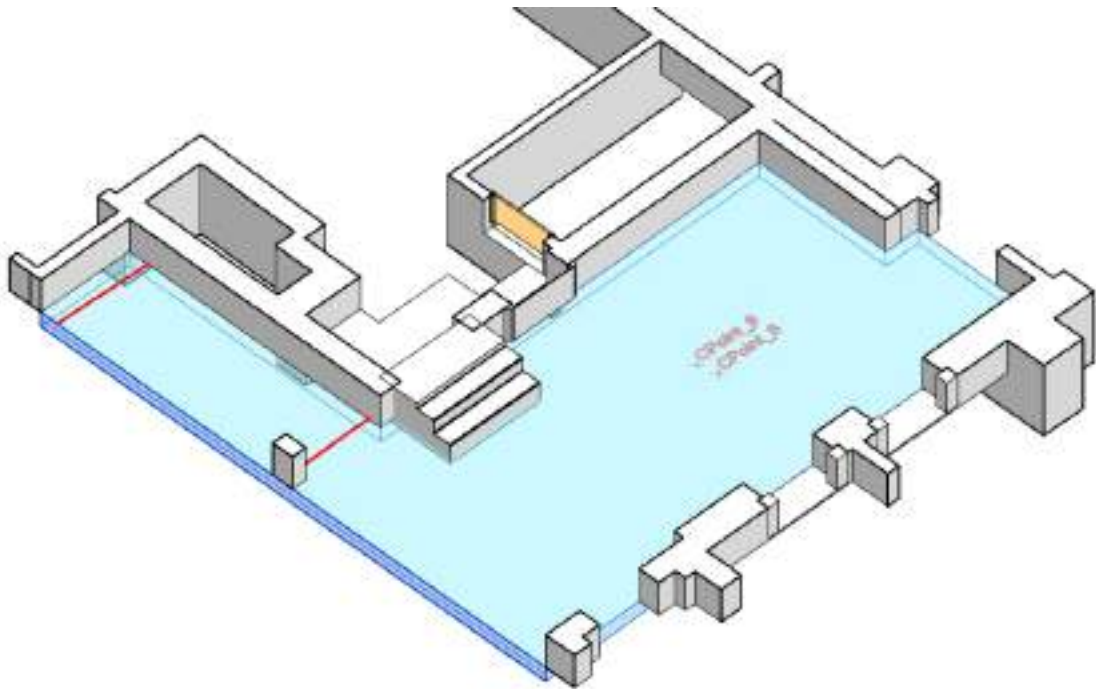
3. The result is a deformed floor with high and low-points:
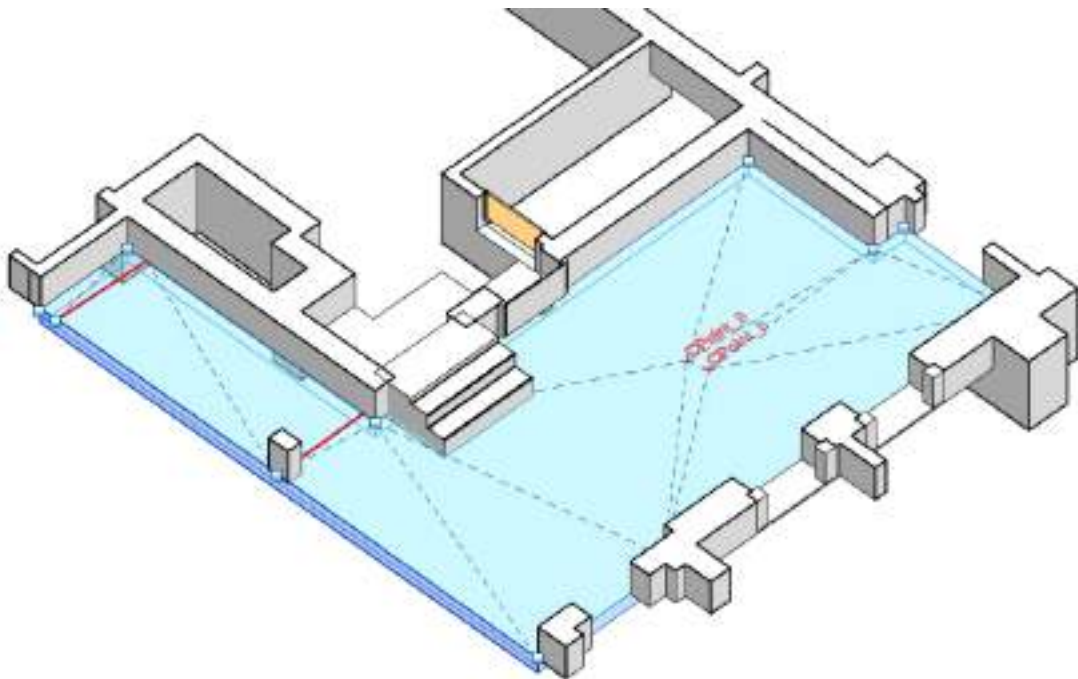
To create complex deformations you can also use the results of the surface analysis. To achieve this, proceed as follows:

■   Compute the deviation of the floor or the roof to the point cloud using the command → **Surface Analysis Tools - Calculate**. Here select the surface that is facing the point cloud.

- Create model lines based on the computed deviation by using the command → **Surface Analysis Tools - Export Results**. Here it is sufficient if the profile lines are only created in one direction (U or V). It is recommended to use the option "Average value" as the export value.

- Use the command "Deform Shape" on the newly created profile lines. If necessary you can beforehand carry out the manual creation of additional construction points or model lines, or corrections to the run of the profile lines.

Object Tools - Load TGA Families

Used in:

Project

With this tool you can simultaneously load several families, composed of pipe accessories and pipe fittings, into your project.
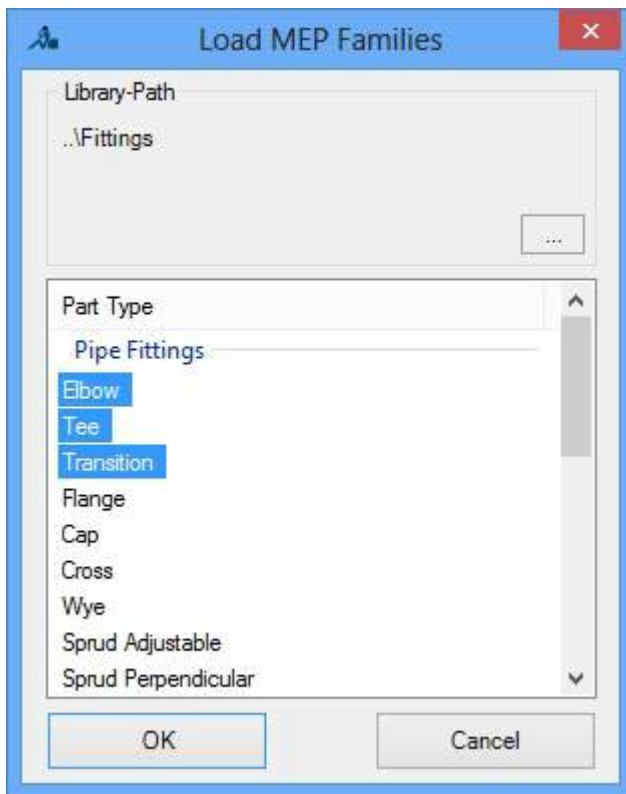
After starting the command a dialogue box opens up:

The currently selected library search path is displayed under [Library search path]. By selecting [...] you can change it.

In the fitting types list you can select the fitting types that you want to load into your project. By pressing the [Ctrl] key it is possible to perform multiple selections from the list.

As soon as you have confirmed your choice with [OK], all of the families in the selected fitting types, from the selected folder, will be loaded into your project.

Object Tools - Load Structural Families

 Used in:

Project

With this tool you can simultaneously load several families of the "Structural Framing" and "Structural Columns" categories, into your project.
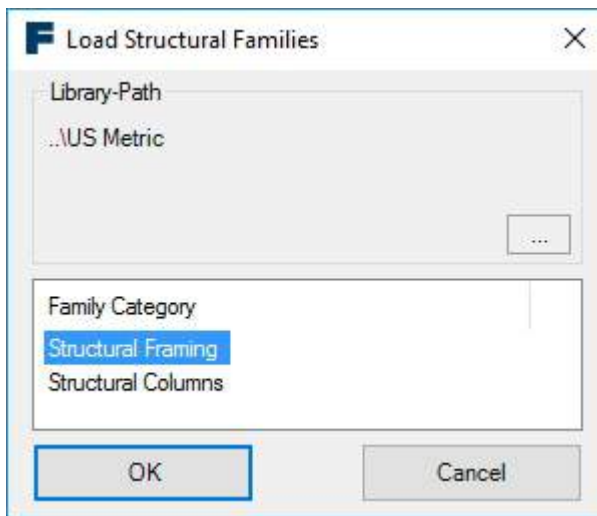
After starting the command a dialogue box opens up:

The currently selected library search path is displayed under [Library search path]. By selecting [...] you can change it.

In the categories list you can select the fitting types that you want to load into your project. By pressing the [Ctrl] key it is possible to perform multiple selections from the list.

As soon as you have confirmed your choice with [OK], all of the families in the selected categories, from the selected folder, will be loaded into your project.

Alignment - Align Walls

Used in:

Project

The command "Align walls" is used to ensure, for the selected walls, perpendicularity, parallelism, wall intersection and continuous axial alignment within a defined margin of error.

The command is primarily used to correct the results of the → **Object Tools - Fit Wall**, when the walls are created with the best possible fit to the point cloud. As a result of using "Fit Wall" the walls are still not joined to one another, wall faces are not exactly in line (especially external wall faces) and the angle between almost perpendicular walls is not precisely 90°.

The user always has full control over how far the walls, that were fitted onto the point cloud, can be corrected.

If the command "Align Walls" is started you will be first asked for the walls that are to be edited:

`Select multiple walls to align and apply orthogonality`	*Now select all of the walls that should be aligned with each other.*

In this step you can only select straight line-based walls. After you have successfully completed the selection, the dialogue box "Align Walls" opens with a first preview of the results.
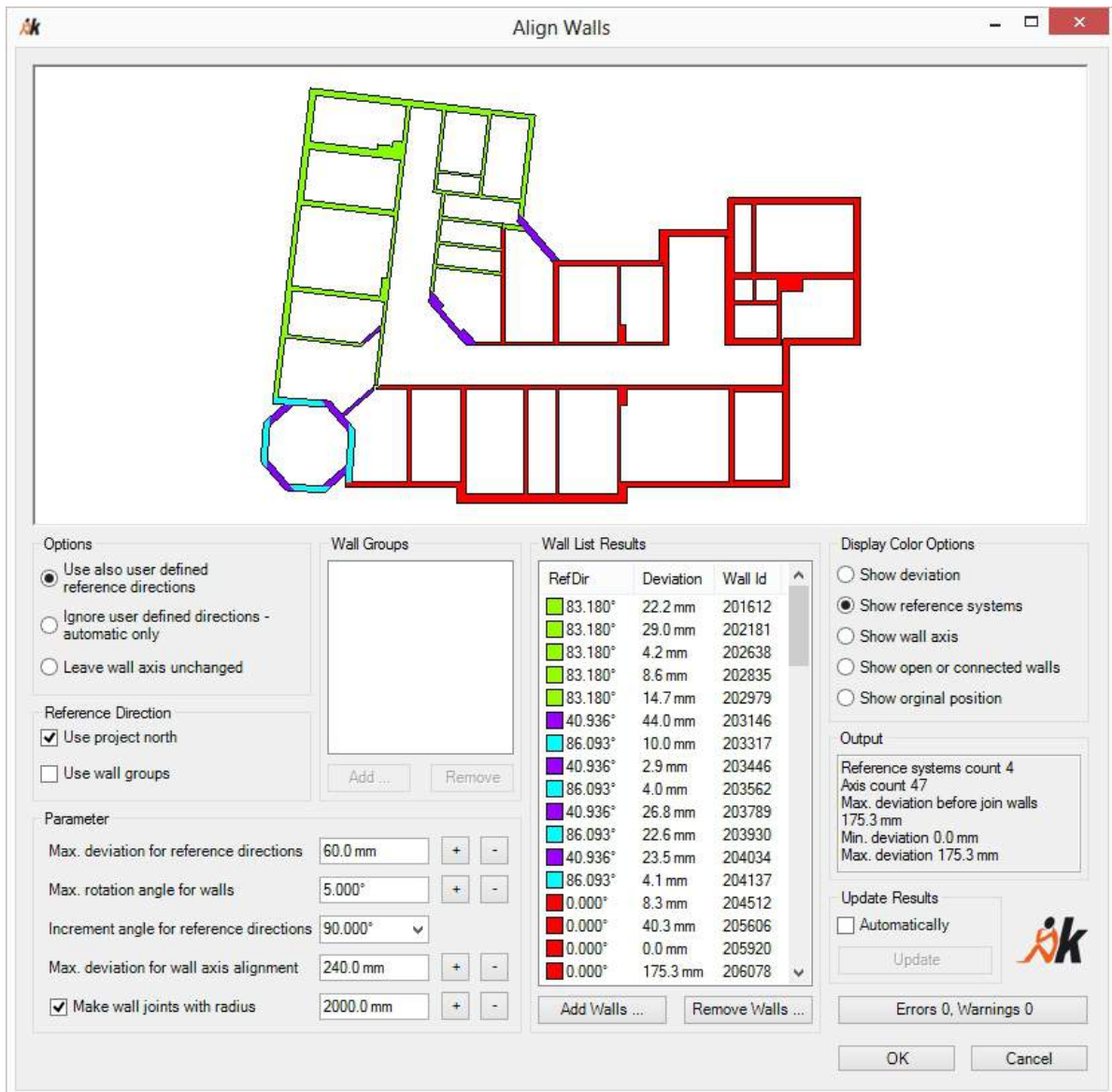
At this point some concepts need to be explained, in order to simplify the next descriptions.

■ Reference systems

All of the walls in a "Align walls" command will be assigned to exactly one reference system. This consists of a reference direction in the floor plan plane and an increment angle. The default increment angle is 90°. There are two categories of reference systems: the user-defined reference systems and automatic reference systems. User-defined reference systems have a higher priority. It is simply not possible to assign a wall, within the predefined margin of error, to a user-defined reference system, when the automatic reference system is being used or determined.

The Revit project north and wall groups will be used as a user-defined reference system.

- Wall groups

  A wall group is a user-defined selection of walls, the mean orientation of which is used as the reference direction of a user-defined reference system. Each of the individual directions of all of the selected walls within a wall group will be averaged. You should therefore select only walls as a wall group, when they run in approximately in the same direction and not for example at right angles to each other.

The user has three options to choose from for the main functionality in the "Options" group:

- **"Use also user defined reference directions"**

  This is the default functionality. Both the Revit project north and also the wall groups will be used as the reference system.

- **"Ignore user defined directions - automatic only"**

  None of the user-defined reference systems will be treated preferentially. It will be automatically attempted to find a solution with the least possible wall deviation.

- **"Leave wall axis unchanged"**

  Only the wall ends within their search radius will be intersected with one another.

In the next command group, "Reference Direction", the user-defined reference systems be selected. The option **"Use project north"** defines the project north with the reference direction 0° as the reference system. The second option **"Use wall groups"** switches the user-defined wall groups as reference systems either on or off. If this option is selected, in the command group to the right "Wall Groups", wall groups can be created or edited.
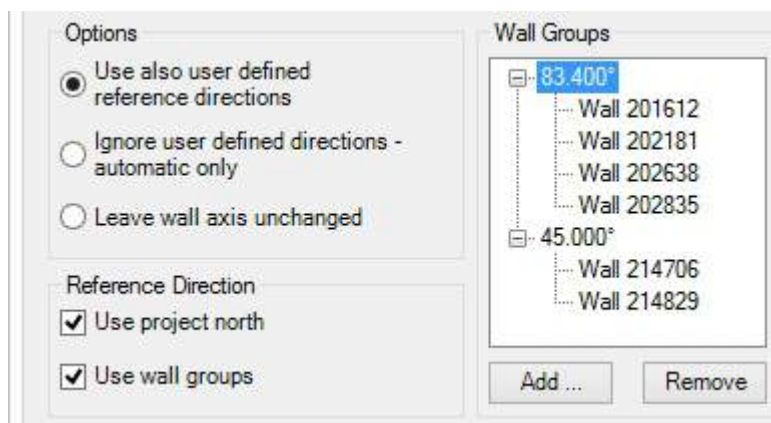
Here, underneath "Wall Groups", you should click on the button **[Add...]**. The dialogue box will disappear and you can select one or more walls for a wall group. You can here also select walls that should not be modified with the "Align Walls" command. Please note that the walls in the wall groups should not be perpendicular to one another, but more or less have the same wall alignment, so that a mean direction angle can be calculated from all of the walls.

If you afterwards want to disregard whole wall groups or individual walls, select the appropriate group or single wall in the tree view and click on the button **[Remove]**. The wall groups are

then appropriately modified to reflect the new selection of walls. By double-clicking on a wall, the element is zoomed onto in the preview window.
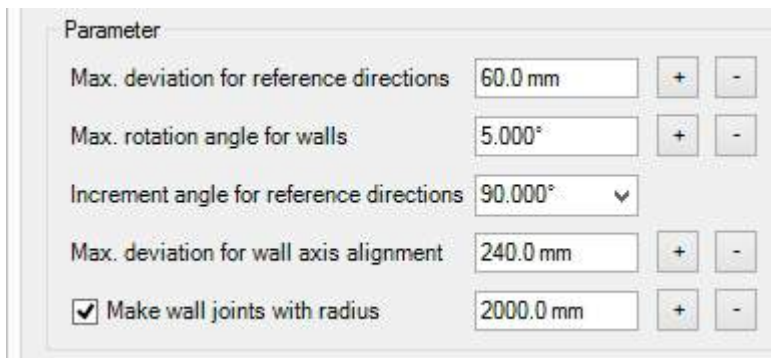
In the next command group, "Parameter", the precise tolerances and other parameters for the wall alignment can be adjusted. To be able to better understand the individual values, it is first necessary to describe the functionality of the algorithm.

The command "Align Walls" has three steps:

1. *Rotate all of the walls* to the most appropriate reference system, should they be within the allowable deviation tolerance value. This deviation tolerance value is comprised of the parameter **"Max. deviation for reference directions"** on the one hand, which represents the maximum deviation from the original wall axis, and on the other hand the parameter **"Max. rotation angle for walls"**, which defines the maximum allowable possible rotation. The second parameter is important, so that short wall segments do not lead to artefacts in the floor plan. But in most cases this can be ignored.

2. *Axially align all of the walls*, so that the displacement of the wall lies within the set deviation tolerance value for axial alignments. This deviation tolerance value for axial alignment is the parameter **"Max. deviation for wall axis alignment"**. It is usually set to a higher value than the first parameter "Max. deviation for reference directions", thereby the rotated walls can also be axially aligned.

3. *Intersect walls with each other*, walls that are within a preset search radius are trimmed or extended, to create a closed floor plan. The search radius is defined by the parameter **"Make wall joints with radius"** and can also be completely disabled by deselecting the tick-box control.

The parameter **"Increment angle for reference directions"** defines the global increment angle for all reference systems. By default it is set to 90° (perpendicular). Behind the text boxes for the above described parameters you will see **[+]** and **[-]** buttons. These double or halve the currently set value of the parameter.

Wall List Results

RefDir	Deviation	Wall Id
🟩 83.180°	22.2 mm	201612
🟩 83.180°	29.0 mm	202181
🟩 83.180°	4.2 mm	202638
🟩 83.180°	8.6 mm	202835
🟩 83.180°	14.7 mm	202979
🟪 40.936°	44.0 mm	203146
🟦 86.093°	10.0 mm	203317
🟪 40.936°	2.9 mm	203446
🟦 86.093°	4.0 mm	203562
🟪 40.936°	26.8 mm	203789
🟦 86.093°	22.6 mm	203930
🟪 40.936°	23.5 mm	204034
🟦 86.093°	4.1 mm	204137
🟥 0.000°	8.3 mm	204512
🟥 0.000°	40.3 mm	205606
🟥 0.000°	0.0 mm	205920
🟥 0.000°	175.3 mm	206078

[Add Walls ...] [Remove Walls ...]

In the next command group "Wall List Results" are tabulated the results of the command "Align Walls". Next to the respective reference direction for each wall is displayed the deviation from its starting position and the Revit "Wall ID". Similarly the legend colour of the wall is placed at the beginning, as a small square symbol. The table can be sorted, in an either ascending or descending order, by clicking the appropriate column header. A double-click on a wall zooms in on the respective wall segment in the preview window. By clicking the **[Add Walls ...]** button, additional walls can be inserted for editing with the command insert. The dialogue box for selecting the new walls will disappear whilst the walls are being selected and will afterwards reappear. Likewise, walls can also be left out of the editing by using the **[Remove Walls ...]** button.

The command group "Display Color Options" controls the colours of the walls in the preview window, and thereby which properties should be visualised in the model. For this purpose there are five display options available:
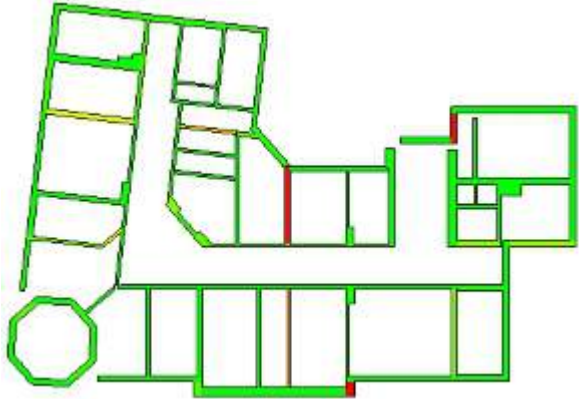
Display Color Options
- ○ Show deviation
- ● Show reference systems
- ○ Show wall axis
- ○ Show open or connected walls
- ○ Show orginal position
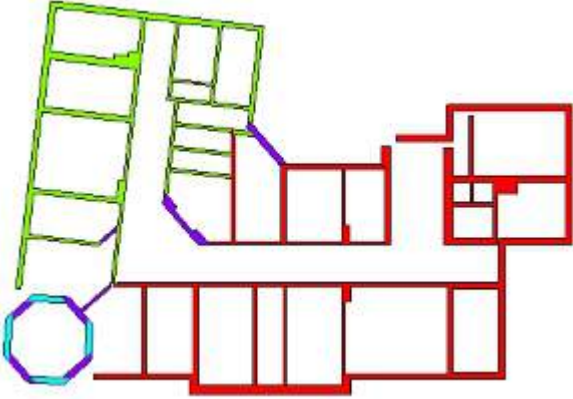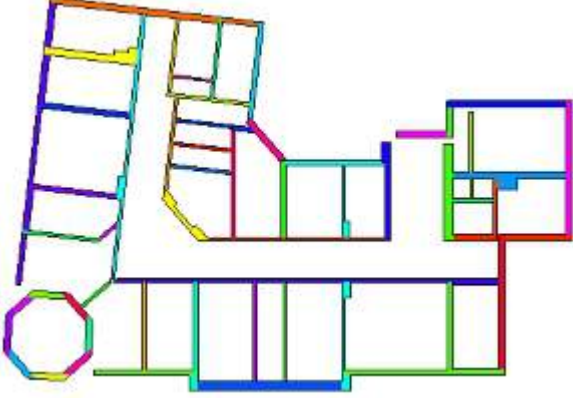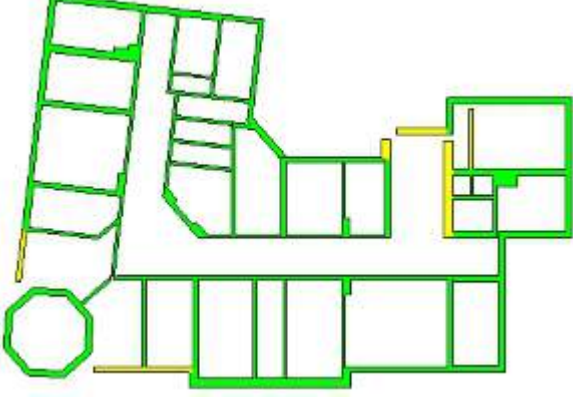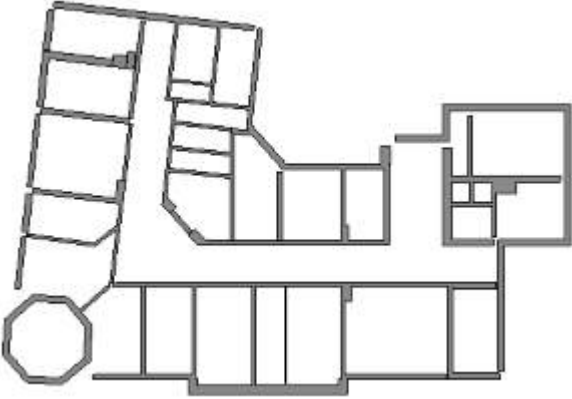
Display option	Sample floor plan
"Show deviation" Displays the wall deviation from its original position. Red corresponds to a large deviation, green corresponds to a minimal deviation.	

Display option	Sample floor plan
"Show reference systems" Colours the model in accordance the membership of the walls to a particular reference system. The same colour mean the same reference system.	
"Show wall axis" Shows wall axes that belong together in the same colour.	
"Show open or connected walls" Displays walls, that are connected at both ends to a neighbouring wall in green, walls that are connected only at one end appear yellow and walls with both ends open are coloured red.	

Display option	Sample floor plan
"Show original position" Displays all walls grey in their original position. When this dialogue box with this display option is closed, then the original position of the walls is restored as well.	

In the text box "Output" you will be shown information regarding the currently executed command "Align Walls". It is the number of the reference systems found, the number of wall axes found, as well as the minimum and the maximum deviation of the walls from their original positions that are displayed. With the maximum deviation it is further differentiated, between before and after the wall displacement.

In the command group "Update Result" the calculation can be updated. If the tick is set in **"Automatically"**, then after each parameter change the calculation will be updated. This can take a very long time in large models. So there is the possibility to disable the automatic update function and then when necessary click on the **[Update]** button to perform a new calculation. This is enabled as soon as a parameter has been changed.
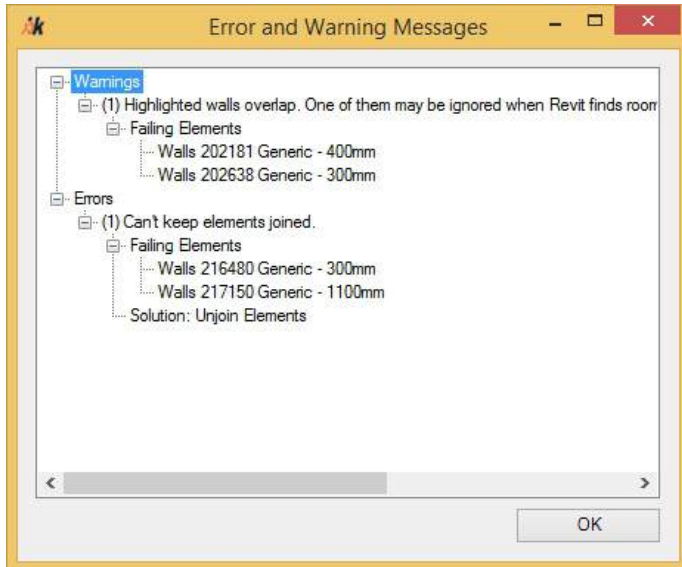
During the fitting of the Revit model using the command "Align Walls", it can lead to Revit errors and warnings being displayed. These will be caught and shown separately. On the button **[Errors xxx, Warnings yyy]** you can see whether warnings have been issued and can view them. With a click on the button, the dialogue box "Error and Warning Messages" will be opened.

By clicking the button **[OK]** in the dialogue box "Align Walls" the command dialogue box will be successfully ended and the result adopted. The **[Cancel]** button though ends the command without taking any action.
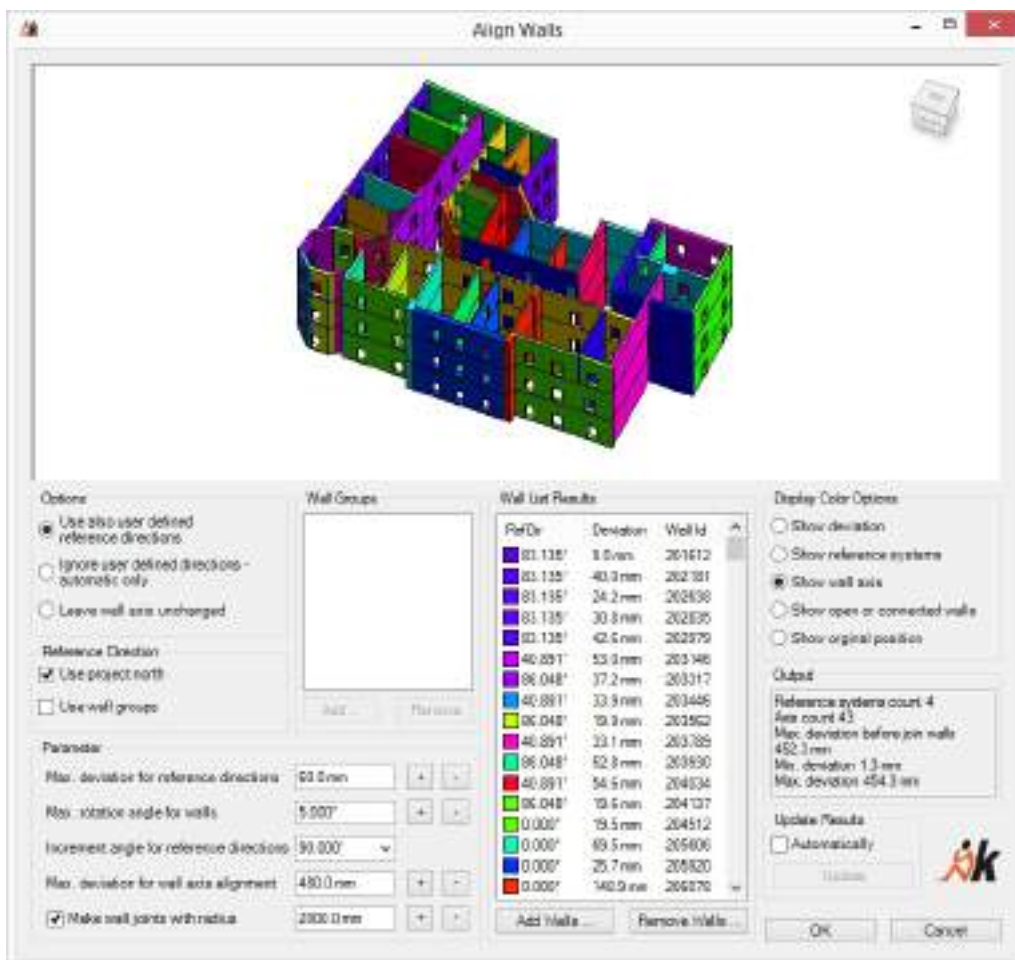
Alignment - Align Pipe Runs

Used in:

Project

To be able to create a fully connected pipe run in Revit, comprised of pipes, pipe fittings and pipe accessories, the components must meet a whole row of consistency constraints:

■ collinear or coplanar axes of the pipes and pipe fittings

■ precise matching of the tie in points to each other

The command aligns the pipe run so that these consistency constraints are met. The requirement for this is that the components already have As-Built connectors.

The purpose of modelling with the tools → **Object Tools - Pipe Fitting** and → **Object Tools - Placement of Pipe Fittings** is to model the components of the pipe run as optimally as possible ("near") to the point cloud. After these first steps in the modelling process the pipe run will not yet meet the required consistency constraints. In reality, there will always be minor deviations and distortions. Even so, it is the scan quality (point cloud noise and density) that determines the modelling precision.

You can, for example, see that the consistency constraints have not yet been met, in that the components:

• lay skewed to each other,

• protrude into each other,

• have gaps between them.
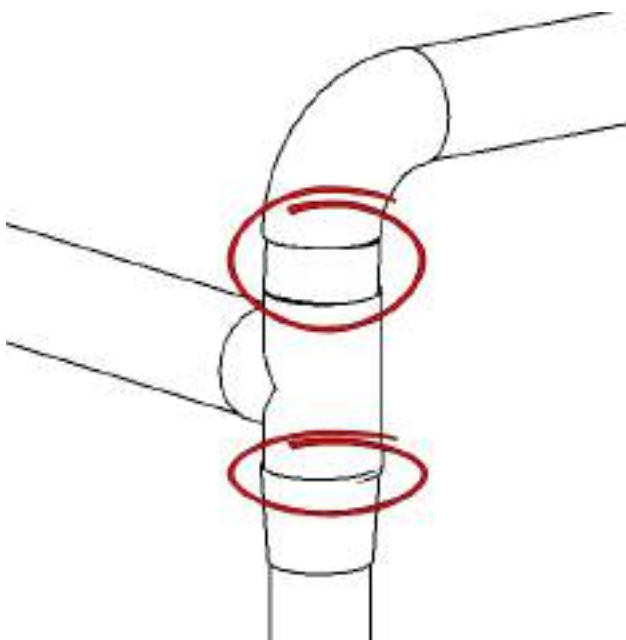
At this point in time of the modelling, the pipe run in Revit is not yet connected. You can make these disconnects visible by using the Revit command "Show Disconnects" 🔍 (Ribbon:

Analyze → Show Disconnects (Pipe):
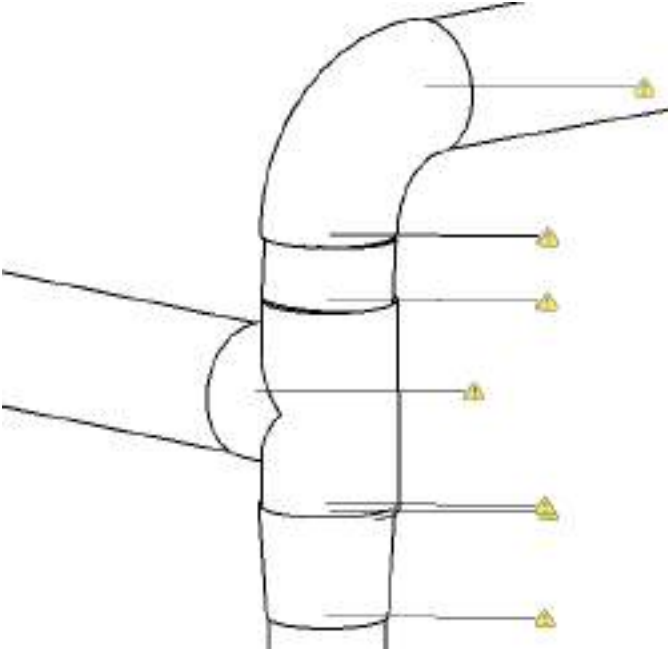
After starting the command you are prompted to select a component on the pipe run:

Please select the pipe
or pipe fitting

Select a component on the pipe run
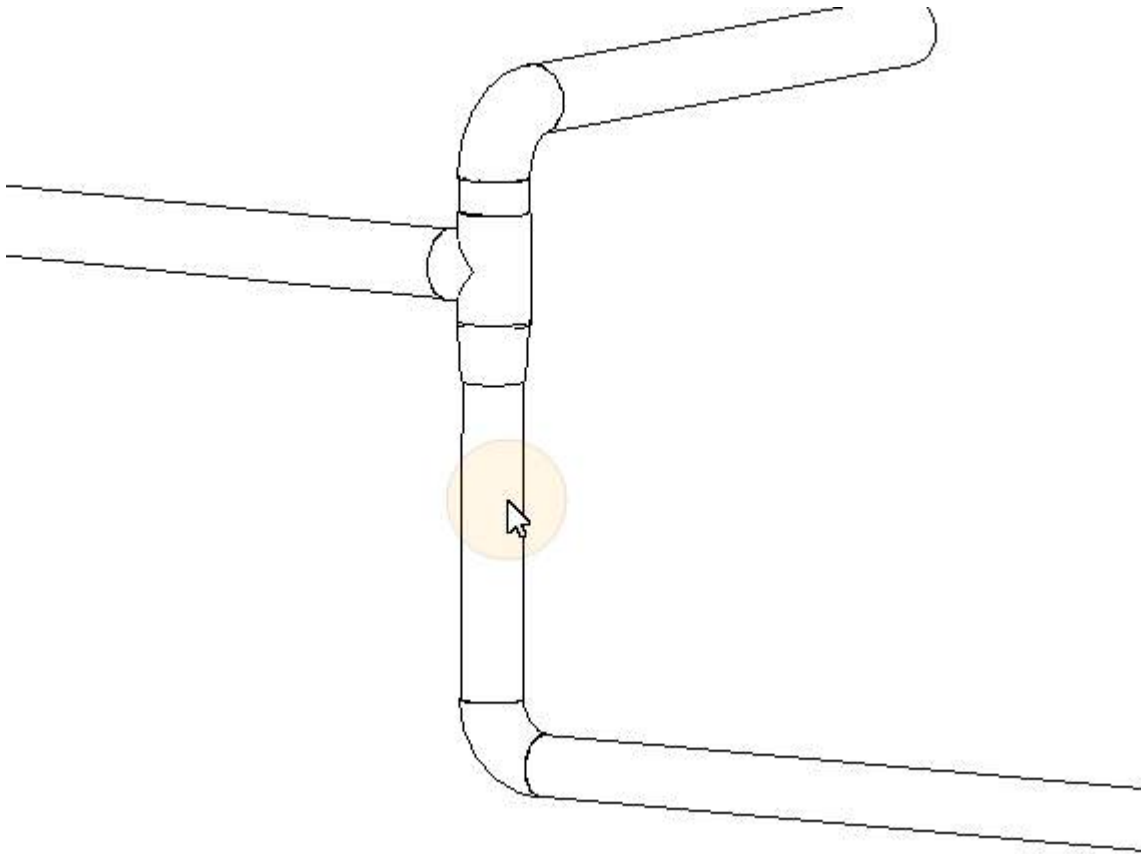
As soon as these have been selected, all of the components in your project that have been connected by As-Built will be highlighted and a dialogue box will be displayed, in which it will be shown how many components belong to the selected pipe run:

Choose Alignment Mode

Alignment Mode

This alignment mode is faster, but cannot accommodate fixed pipe axes or fixed fittings. Furthermore, cycles in the piping graph are usually disconnected.

This alignment mode is slower, but can accommodate fixed pipe axes or fixed fittings. Cycles in the piping graph are usually preserved.

○ Use fast alignment mode   ◉ Use precise alignment mode

8 elements have been selected and highlighted for pipe alignment. Proceed with this selection?

Start   Cancel

Should only a part of your pipe run be selected, please check the connectors at the pipe joints. As-Built connectors are placed automatically, if you use the command → **Object Tools - Placement of Pipe Fittings**. But you can also later insert As-Built connectors with the commands → **Alignment - Positioning Pipe Connections** and → **Alignment - Set Pipe Link (Auto)**.

For the alignment of the pipe run there are two different procedures available to you:

Quick alignment

All of the connected components will be appropriately aligned, based on a single component in the middle of the pipe run. The procedure is quicker but less precise and does not take into account any cycles or fixed elements.

Precise alignment

The algorithm minimally changes the axial position of the pipes and moves and rotates the geometry of all of the connected pipe fittings, so that a precisely fitting model is created. With this procedure, cycles in the pipe system and fixed components are also taken into account.

In the first step "Align", the pipe fittings are fitted onto the pipe axes, without modifying the axes themselves. Subsequently the mutual correction of the pipe fittings and pipes is carried out. On the positions with the largest movements the connection points are marked in Revit with red arrows, so that the user can if necessary identify the sources of the errors.

The procedure is so long carried out until one of the stop criterion has been met:

■ At least the angular tolerance set value for the connection points will be met at each position. The default value here is 0.01° and can be adjusted in → **As-Built - Settings**.

■ The computation time range of 10000 iterations has been exceeded. The model, after the iteration procedure, had not met the preset angular tolerance. In this case, the procedure for a "Quick alignment" can also be optionally applied. This option can be activated in → **As-Built - Settings**.

■ During the procedure the user can press the [Esc] key. The model remains in the currently displayed state. The "Quick alignment" will be optionally applied.

In the progress dialogue box the current maximum length of pipe of the moved connection points will be shown, as well as the maximum angular variance for all of the connection points in the selected pipe run.

Confirm the above dialogue box. The alignment of the pipe run will be started and in a progress dialogue box will be shown:

After the computation is complete your pipe run is correctly aligned and the components are connected with each other:
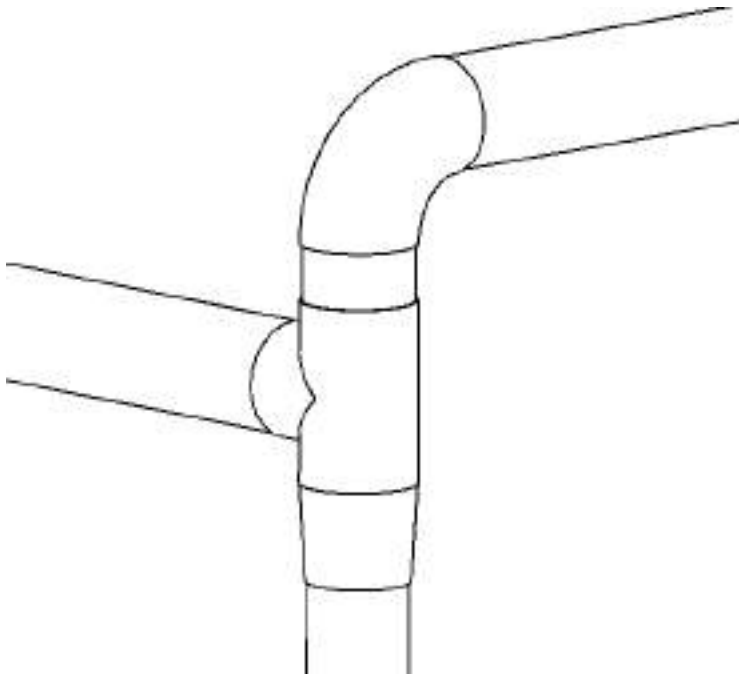
Alignment - Positioning Pipe Connections

Used in:

Project

To be able to align a pipe run with → **Alignment - Align Pipe Runs** all of the components of this pipe run that are to be aligned need to have been connected with As-Built's own pipe connectors (also see → **Alignment - Align Pipe Runs**).

If you insert pipe accessories with the command → **Object Tools - Placement of Pipe Fittings**, then the connectors are placed automatically. Usually it is only necessary to manually place the connector if you have deleted components or inserted new components with standard Revit tools.

After starting the command all of the As-Built connectors are displayed as circles around the connected pipe ends. If the connected components are not flush fitted with each other, then a line is displayed between the circles
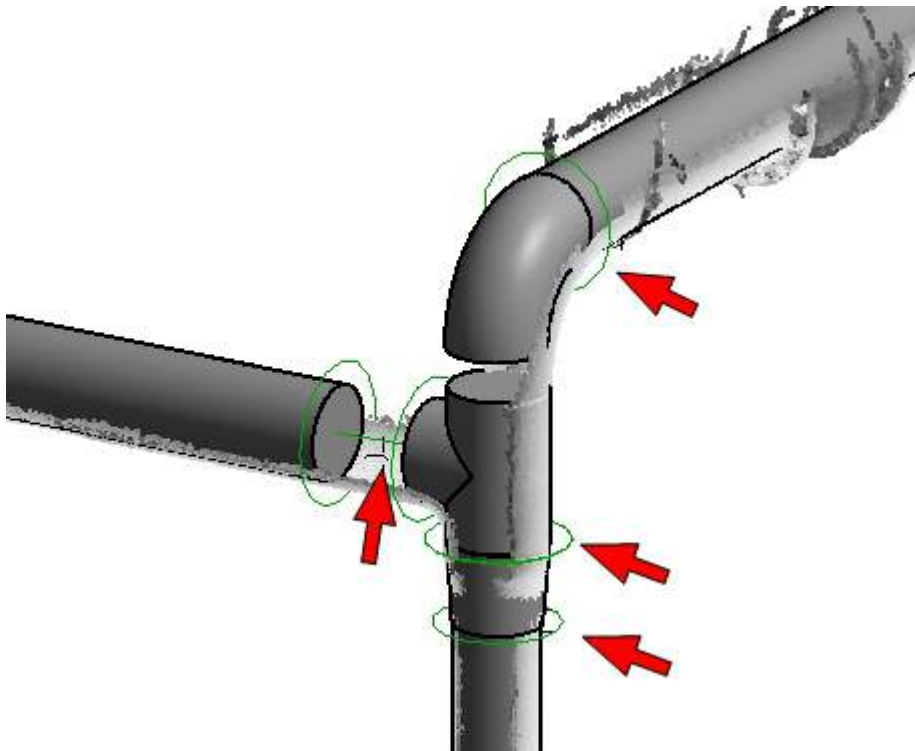
In the above example a pipe connector is missing between the T-piece and the pipe bend. The command → **Alignment - Align Pipe Runs** will not detect any completely connected pipe run.
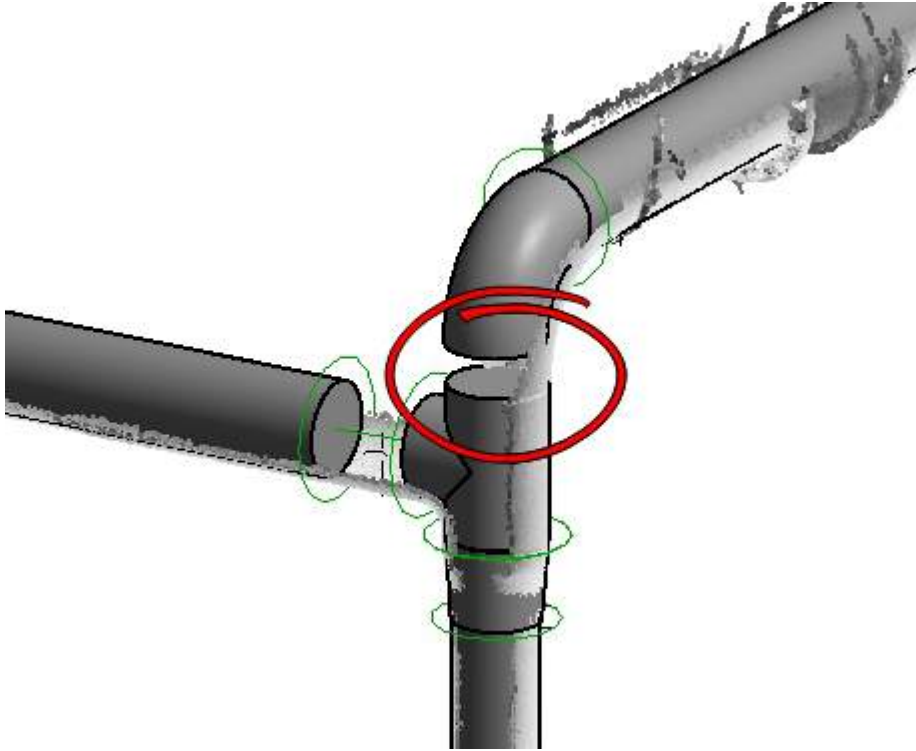
After starting the command comes a prompt to select the pipe ends to be connected:

`Pick the first pipe end`	*Select the first pipe end for the connection*
`Pick the second pipe end`	*Select the second pipe end for the connection*
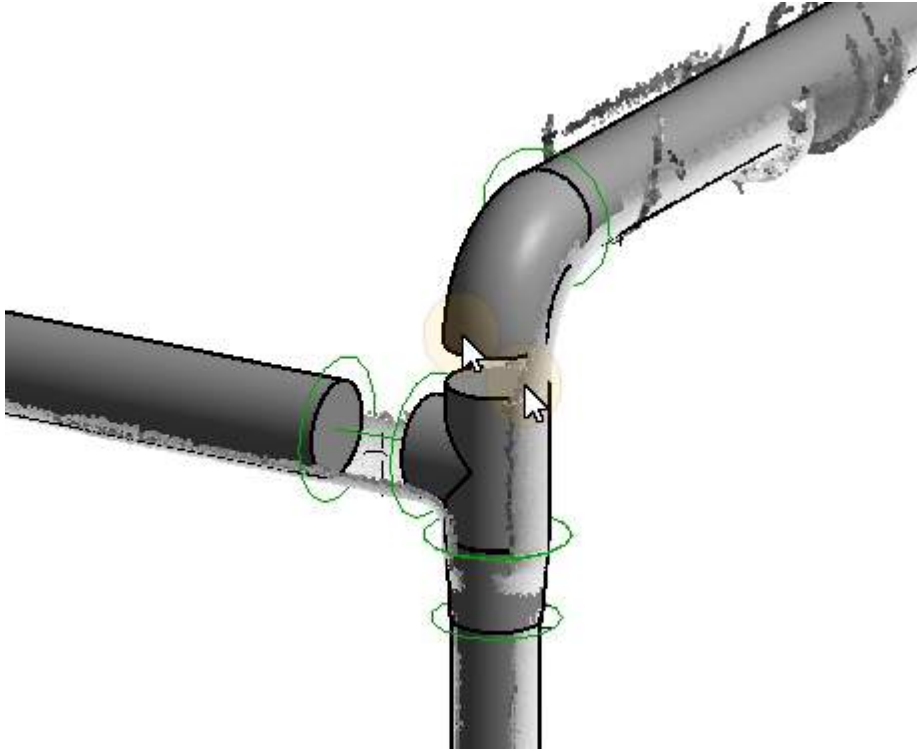
The selected pipe ends will be connected:
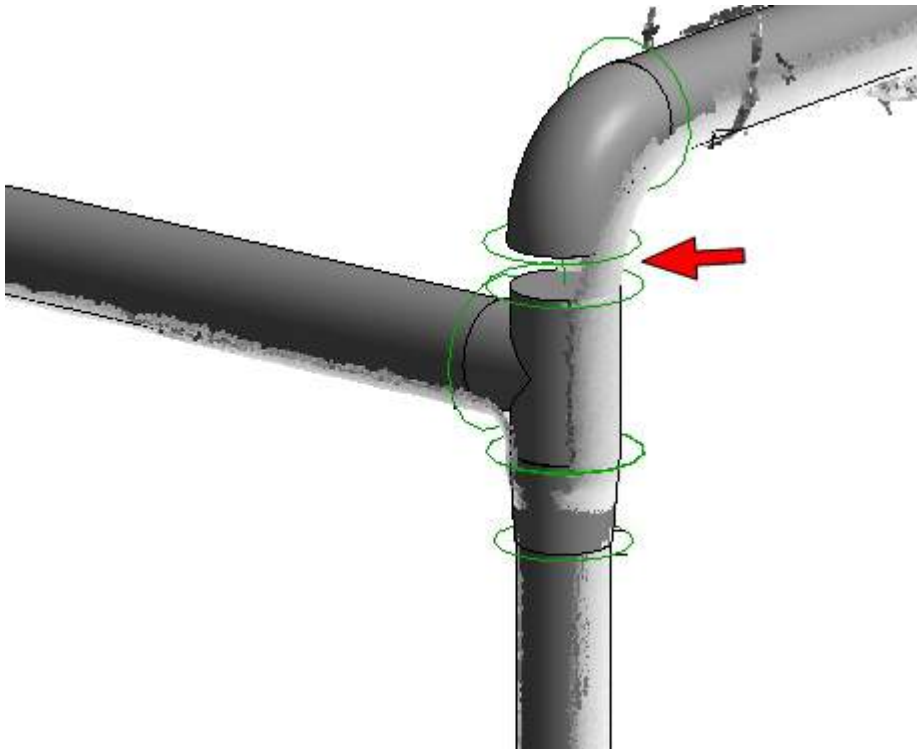
As soon as a new connector has been created, you can immediately continue with a further connector. Use [Esc] to end the command.

Alignment - Set Pipe Link (Auto)

Used in:

Project

This tool serves just like → **Alignment - Positioning Pipe Connections** for defining connected pipe runs, so that they can be aligned with the command → **Alignment - Align Pipe Runs** . The tool automatically places As-Built connectors between neighbouring components. This can be useful, when you have copied a pipe run and thereby deleted all of the connectors, or if you have modelled a pipe run without using As-Built commands and you want to align them using → **Alignment - Align Pipe Runs**.

After starting the command a dialogue box is displayed, where you are asked for the search radius to be used in the automatic placing of As-Built connectors:
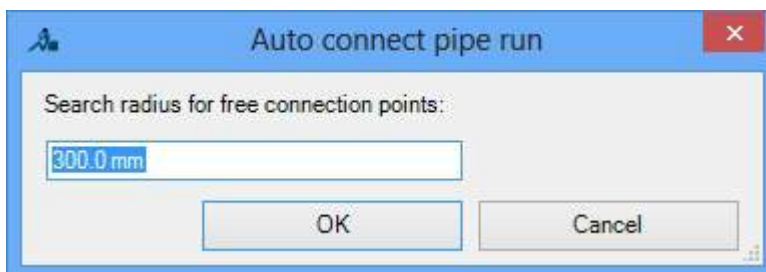
The search radius defines in which radius around a pipe end a neighbouring pipe end should be looked for. With all of the neighbouring pipe ends with the same nominal diameter then the nearest connector will be selected. After confirming the dialogue box a new dialogue box is displayed, in which the number of newly placed connectors is shown, and the newly placed connectors will be displayed in the project:
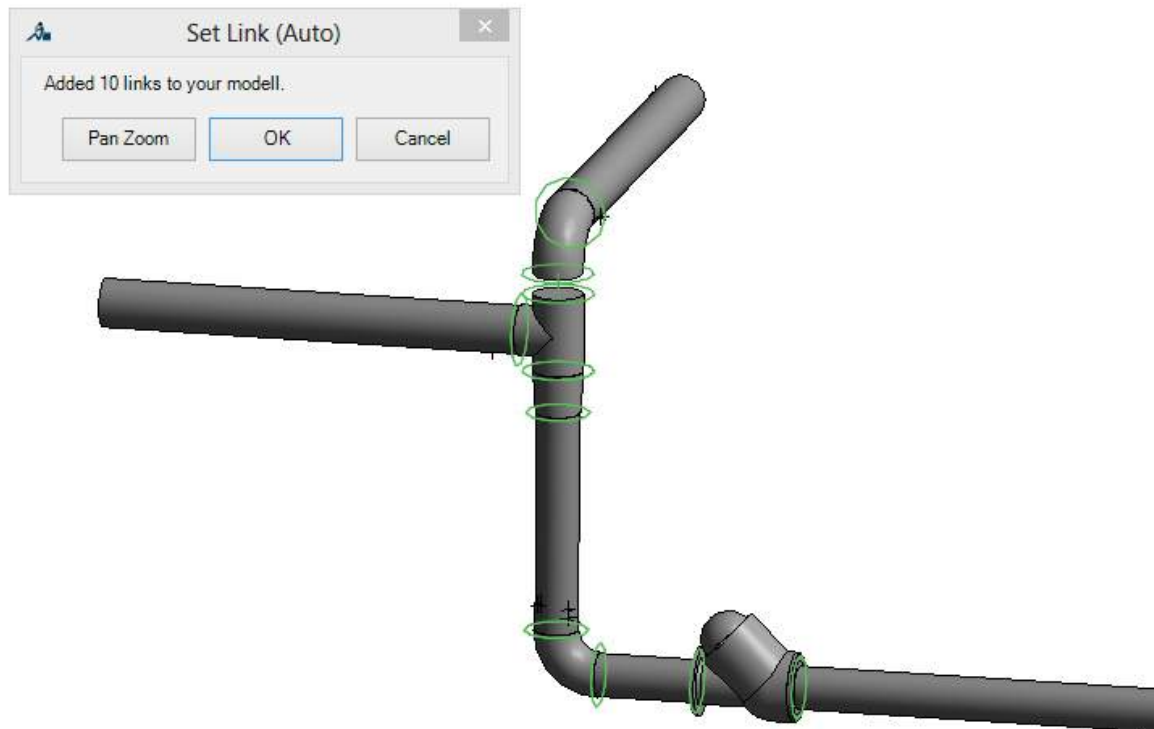
You can confirm the newly placed As-Built connectors by selecting **[OK]** or discard them by selecting **[Cancel]**. By clicking the button **[Pan Zoom]** you can first inspect and verify the suggested connectors. Using **[Esc]** you return to the dialogue box to confirm the connectors.

Alignment - Removing Pipe Connections

Used in:

Project

This tool serves like → **Alignment - Positioning Pipe Connections** for defining connected pipe runs, so that they can be used for alignment with → **Alignment - Align Pipe Runs**.

After starting the command the existing As-Built connectors are displayed as connected circles. You will be prompted on the status line to select the connectors that you want to remove.

`Pick a connecting line to delete the link`	*Select one of the circles of connected pipe ends or one of the lines, which symbolise this connector.*
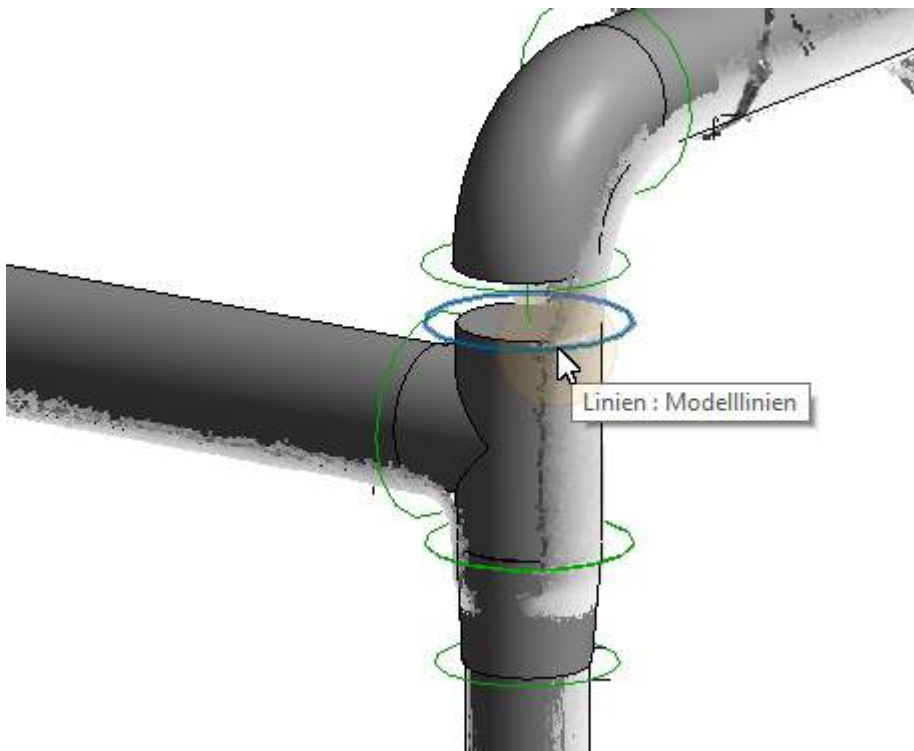
After you have selected the connector, it will be removed:
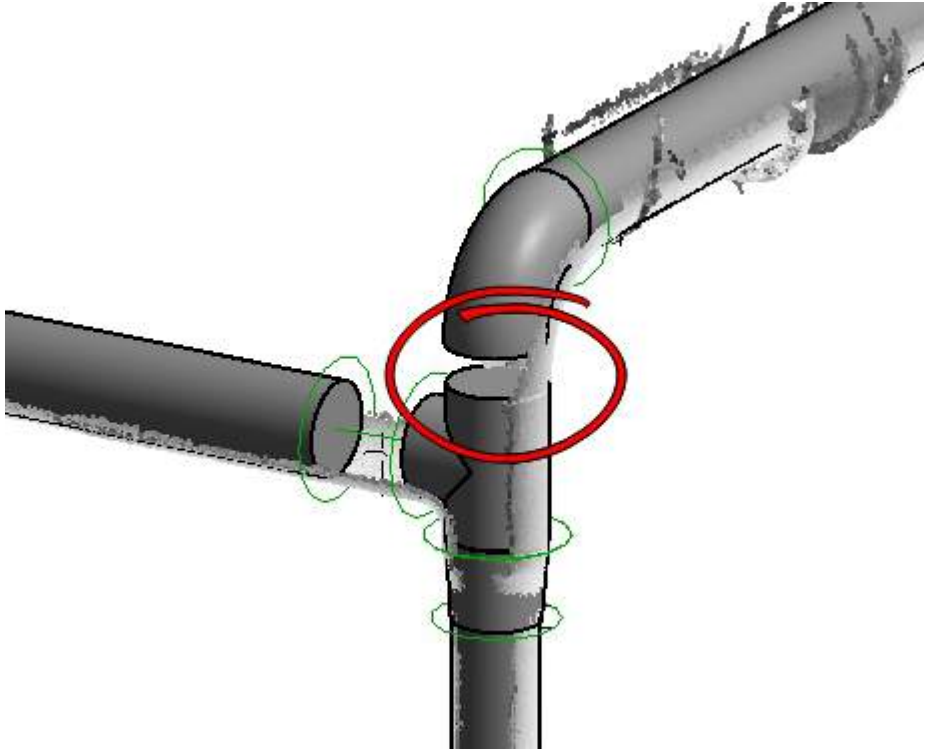
As soon as a connector has been removed, you can immediately continue with a further connector. Use [Esc] to end the command.

Alignment - Align Structural Elements

Used in:

Project

With this command you can align multiple instances in the categories "Structural Framing" and "Structural Columns" with each other. This is useful for further (structural) analyses or additional modelling of beams already fitted in the point cloud (➔ **Object Tools - Fit Structural Elements**).

The command determines an optimal coordinate system and aligns the selected beam to it. The axes of the beams are aligned, within the specified tolerances, to this coordinate system, so that they intersect on their axis end points.

After calling the command the following dialogue box is displayed:
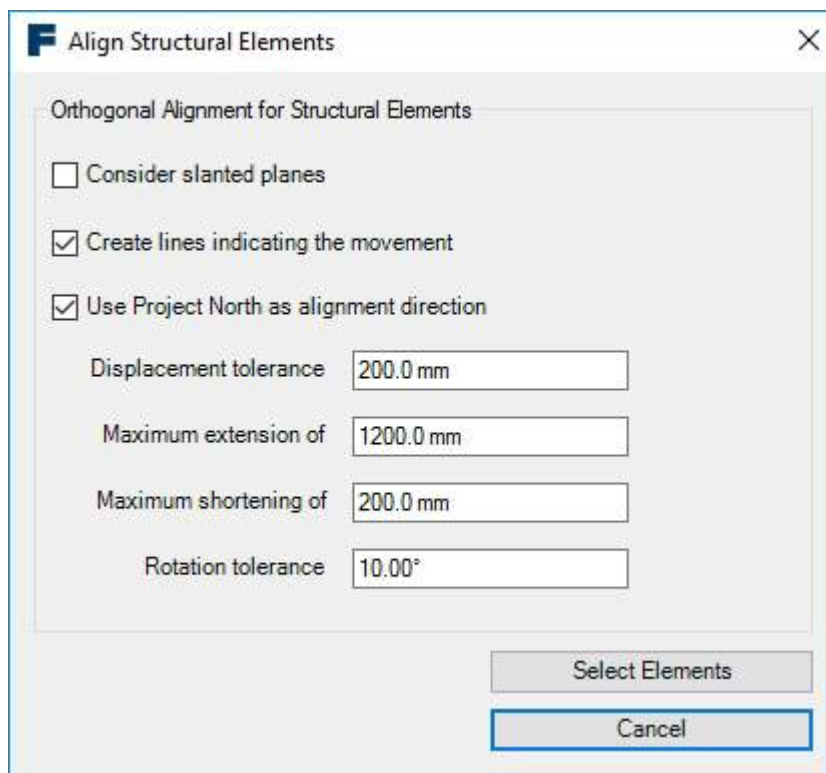
Dialogue Element	Explanation
Takes into account inclined planes	When this option is selected the beams will be aligned to the optimal coordinate system not to the orthogonal planes. In this case any diagonal beams are aligned parallel to each other.
Draw displacement lines	Short green lines show the necessary displacement of each of the beams. A line is drawn from the start point and another one from the end point of the original beam to the new axis.

Dialogue Element	Explanation
Use align to project north	The optimal coordinate system will be aligned to the project north, so long as it remains within the specified tolerances.
Displacement tolerance	Maximum permissible value for the tilting of the axis
Maximum extension of	Maximum permitted extension of the element, to create intersections.
Maximum shortening of	Maximum permitted shortening of the element, to create intersections.
Rotational tolerance	Maximum permissible value of the rotation about the axis
[Select element] Button	Selection of the elements to be aligned and start the analysis

The command ensures that a beam end (except for extending) will not be changed by more than the tolerance value. That means that the tolerance must be significantly smaller than the smallest beam, but large enough to be able to compensate for the largest deviation from the perfectly aligned composition of beams.

Work Plane Tools - Fit Plane (n Points)

Used in:

Project

The command creates an averaged plane from an arbitrary large number of picked points in a point cloud. Amongst other things, the active work plane can be created with this command.

After starting the command there comes the prompt to click on points:
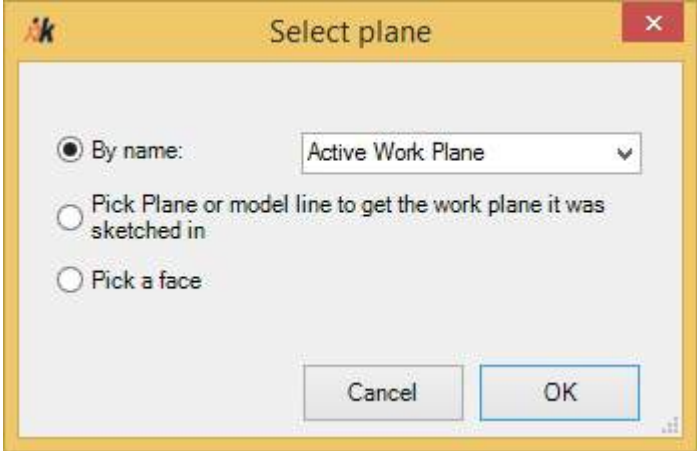
> `Select a point in the point cloud`
>
> *Pick a point on the scanned plane on which you want to model.*

As soon as at least one point has been picked the picking can be ended by pressing the [Esc] key.

A dialogue box will appear which displays the picked points, the quality of the plane as well as the output modes:
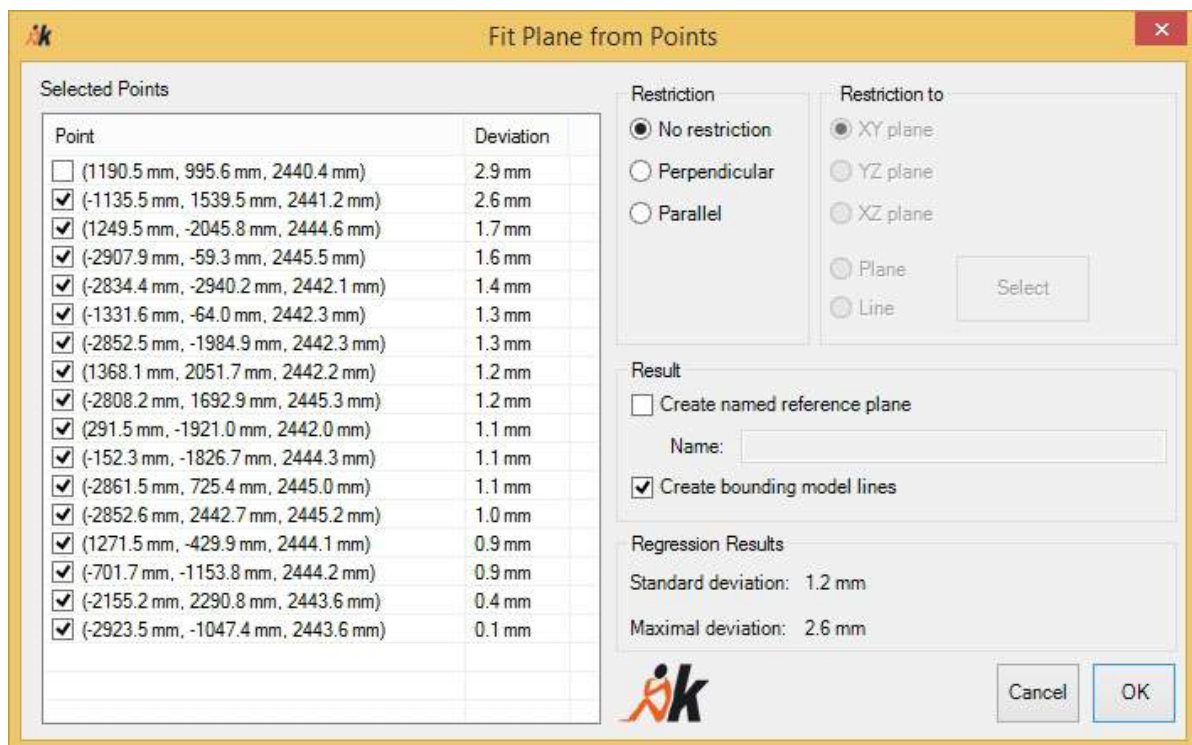
Dialogue Element	Explanation
"Selected Points" list	The selected points together with their coordinates are displayed here. The "Deviation" column displays the deviation of the respective point from the averaged plane.

Dialogue Element	Explanation
"Restriction" option	The plane averaging can be restricted with respect to another plane or line with a "Restriction to" option. With the "Orthogonal" option the plane is so averaged that it stands orthogonal to the selected plane or line. With the "Parallel" option a work plane is created that is parallel to the selected plane or line. When the "Orthogonal" option is selected, at least two clicked points are required for the averaging, in the case of "Parallel" only one point is needed.
"Restriction to" option	The "X-Y plane" is a plane that is parallel to a Revit level. The "X-Z plane" is a plane, that stands orthogonal to the Revit levels and runs in an east-west direction. The "Y-Z plane" represents the plane, that stands orthogonal to the Revit levels and runs in a north-south direction. By selecting "Level" or "Line" the button [Select] becomes active. After clicking on the button you will be prompted to select a model line or level in the Revit project. After something has been selected, there will appear next to the [Select] button a green tick mark, otherwise it will be a red question mark. When selecting a plane a temporary dialogue box is displayed, in which you can decide which type of plane is to be selected: a named Revit level or a reference plane by name, a planar model surface or the sketch plane of a model line:
"Result" options	In each case the command creates an active work plane. In addition, a reference plane can also be created with a name or a bounding polygon from model lines. The model lines contain the

Dialogue Element	Explanation
	averaged plane as a sketch plane.
"Regression Results" information	Here is displayed the quality of the plane, with its standard deviation and maximum deviation.

By confirmation of the [OK] button the averaged plane will be set as the active plane and according to the options selected under "Result" a reference plane or a bounding polygon from model lines will be created.

Work Plane Tools - Fit Plane (1 Points)

Used in:

Project

This command searches for and averages a plane with a single click on a planar point cloud surface. Thereby a work plane can be set by a single click.
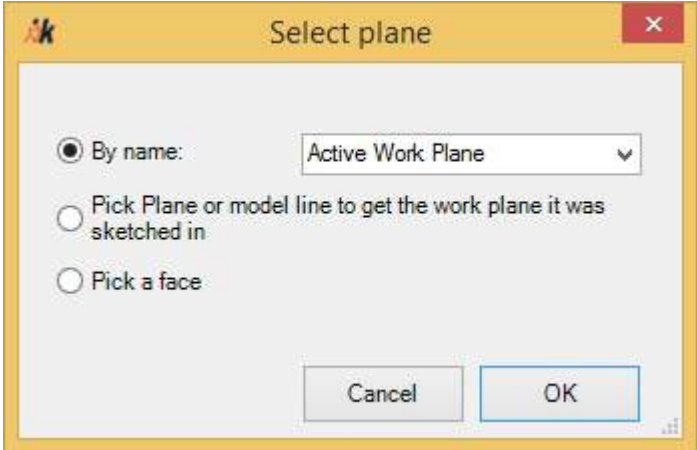
After starting the command there comes the prompt to click on points:

`Pick a Point on Point Cloud (current search radius: 1500.0 mm)`	*Click a point on a point on the planar surface that you want to be fitted as a plane. The value after "current Search Radius" shows the currently set search radius.*

As soon as you have clicked a point, a short progress bar will appear and then the following dialogue box:
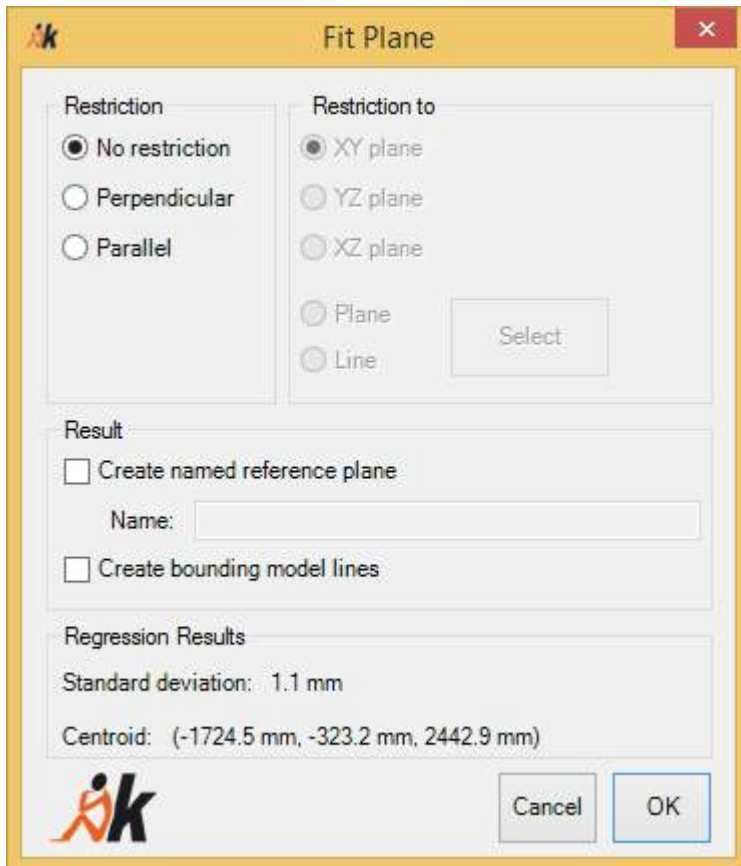
Dialogue Element	Explanation
"Restriction" option	The plane averaging can be restricted with respect to another

Dialogue Element	Explanation
	plane or line with a "Restriction to" option. With the "Orthogonal" option the plane is so averaged that it stands orthogonal to the selected plane or line. With the "Parallel" option a work plane is created that is parallel to the selected plane or line. When the "Orthogonal" option is selected, at least two clicked points are required for the averaging, in the case of "Parallel" only one point is needed.
"Restriction to" option	The "X-Y plane" is a plane that is parallel to a Revit level. The "X-Z plane" is a plane, that stands orthogonal to the Revit levels and runs in an east-west direction. The "Y-Z plane" represents the plane, that stands orthogonal to the Revit levels and runs in a north-south direction. By selecting "Level" or "Line" the button [Select] becomes active. After clicking on the button you will be prompted to select a model line or level in the Revit project. After something has been selected, there will appear next to the [Select] button a green tick mark, otherwise it will be a red question mark. When selecting a plane a temporary dialogue box is displayed, in which you can decide which type of plane is to be selected: a named Revit level or a reference plane by name, a planar model surface or the sketch plane of a model line:
"Result" options	In each case the command creates an active work plane. In addition, a reference plane can also be created with a name or a bounding polygon from model lines. The model lines contain the averaged plane as a sketch plane.

Dialogue Element	Explanation
"Regression Results" information	Here is displayed the quality of the plane together with its standard deviation and maximum deviation.

You can set the command relevant parameters with the command → **As-Built - Settings** ☼.

The relevant settings are to be found in the "Plane Detection" and "Point Cloud Parameters" group:

Parameter value	Explanation
"Plane Search Radius"	This value determines the radius from the picked point within which the points will used to find and form the plane. You will get the best results when this value is approximately the size of your structure. For example, if you want to fit a plane on a building facade that is 10 x 10 m, then use a search radius of 5 m. However, should you want to fit planes on room walls of approximately 2.5 x 4 m, then use a search radius of around 2 m. The search radius can be slightly larger than your structure size, but the plane fitting can, in certain circumstances, take a little longer.
"Noise"	This value sets how far points can lie from an ideal plane. The value depends on the characteristics of your scan data (noise of your scanner) as well as the quality of the surface and the curvature of the scanned surface as apposed to an ideal plane. The value can be somewhat larger than the actual value. The accuracy of the results can, however, only be assured within the set value. For example, with a set noise value of 5 cm, an accuracy can only be guaranteed to 5 cm. You can achieve the best results, for example if you use 3 mm for room walls, but for facades you should use 1-2 cm and 5 cm roof surfaces.

You can also automatically determine the point cloud parameter using the command → **Point Cloud- Auto Measure Parameter**.

Work Plane Tools - Plane fitting with contour

Used in:

Project

This command allows you to extract planes from the point cloud, together with their boundaries as model lines.

This thereby allows, for example, the creation of roof edges for modelling from roofs:

Starting with the point cloud...
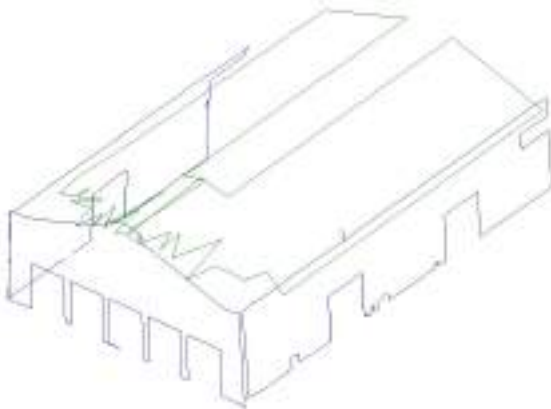
... With the assistance of the command **"Plane fitting with contour"** the plane of a roof together with its boundary is determined.
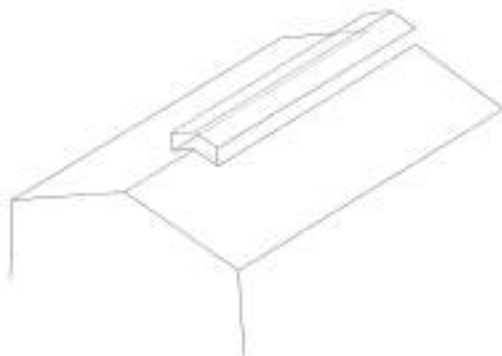
From these planes, the roof edges can be created as model lines, with the assistance of the command → **Model - Intersect 3 Planes**.
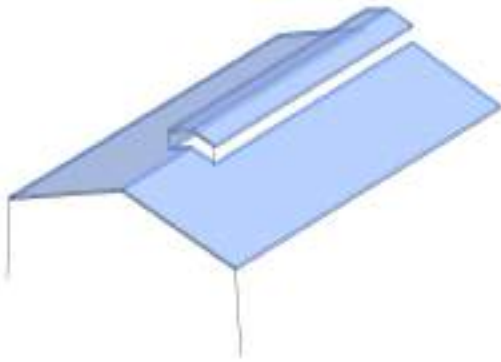
This "wire-frame model" can now be used as the basis for creating an in-place family (tab Massing & Site → panel Conceptual Mass → In-Place Mass), the roof can then be very easily and precisely created from these surfaces (tab Massing & Site →panel Model by Face → Roof by Face).

After calling the command the following dialogue box is displayed:
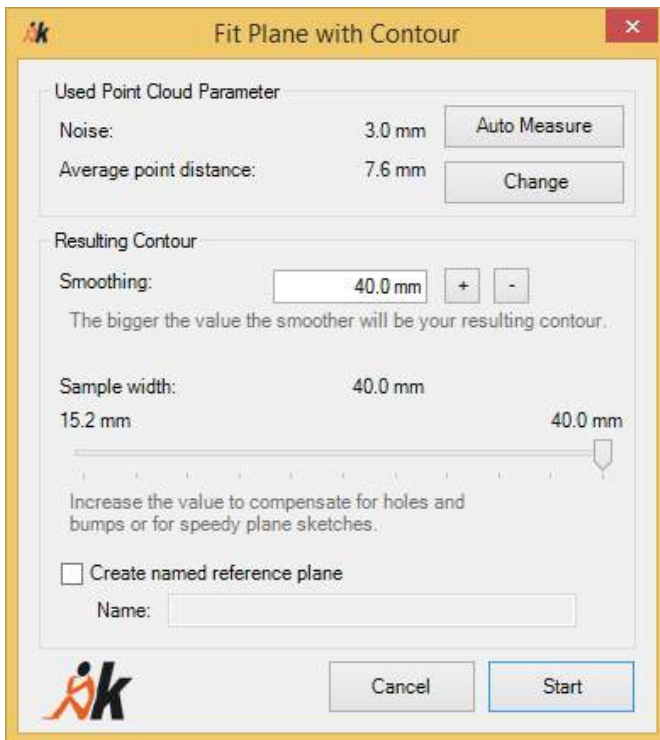
This shows the settings that will be used with the command. Confirm the dialogue box with **[Start]**, the following prompt appears in the Revit status bar:

> Pick a point from the point cloud that lies on a planar surface

Now pick a point on the point cloud surface that you want to model as a plane with a boundary

Then the command starts to define the plane and its contour. During the calculation the following status dialogue box will be displayed:
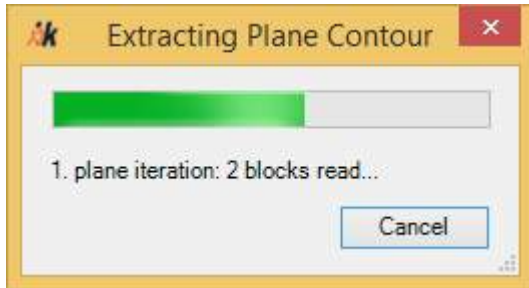
As soon as the command is finished the status dialogue box disappears and you get a work plane on the point cloud plane that has been recognised, together with its boundary as model lines.

For the following façade you can, for example, get with a single click on the façade wall, the adjacent result of the bounded façade plane:

Parameter

The dialogue box offers several settings which influence the quality of your results:
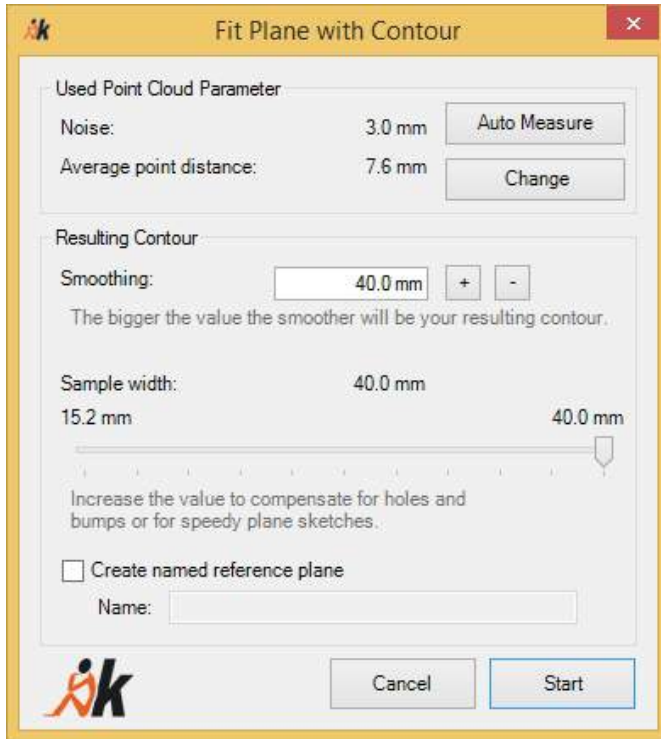
The point cloud parameters **Noise** and **Average point distance** are displayed as information. By clicking the button **[Change]** the settings dialogue box is displayed to change the values (→ **As-Built - Settings**). Over **[Auto Measure]** the command → **Point Cloud- Auto Measure Parameter** started, to automatically determine the point cloud parameter.

The **Noise** influences the accuracy of the plane. The parameter defines by how much the plane may deviate from the ideal plane. The value must take into consideration as well as the noise of the scan also the surface properties and the curvature properties. For example, the value for a rough render should be larger than 5 mm, because the surface of the rough render alone deviates by around 5 mm. Likewise an uneven floor behaves in the same way. When it curves in the middle by approximately 1 cm, then the noise here must also be at least 1 cm. The larger the value is the less precise is the resulting plane. When the value is set to 1 cm it means that the plane can deviate by up to 1 cm at the edges. If the value is too small it means that determination of the plane terminates prematurely and not all of the points in the scanned plane will be used in determining the plane.

The parameters **Smoothing** and **Sample width** influence the quality of the resulting contour.

With **Smoothing** you enter a value which is the tolerance within which points on the contour belong to a line segment. The larger this value is the less "frayed" the contour will appear. The following example illustrates this:

A small contour smoothing of 5 mm. Many corner points are generated. This can be recognised by the light blue circles on the façade edges. What is really wanted is a continuous façade edge without any intermediate points.

With a smoothing value of 50 mm there are no intermediate points on the façade edge. The façade edge is consistently "smooth".

By using the buttons **[+]** and **[-]** you can quickly double or halve the value.

With **Sample width** you define by how much it should extend over holes and swells. The following example of a hall roof demonstrates this.
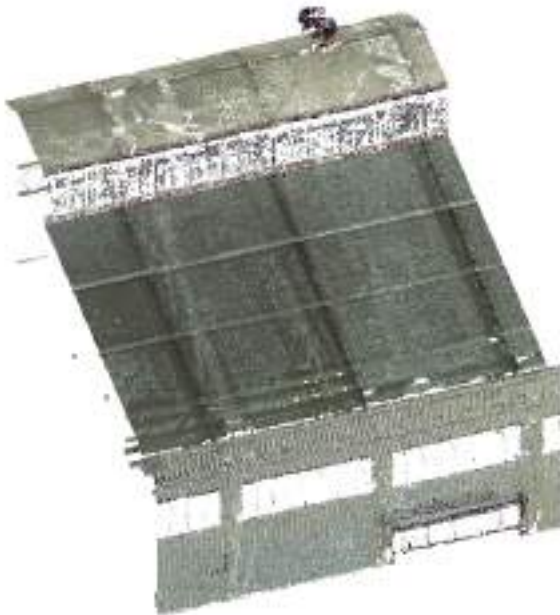
Look at the roof from the front and you will see that these swells, of approximately 15 cm, break up the roof cladding.
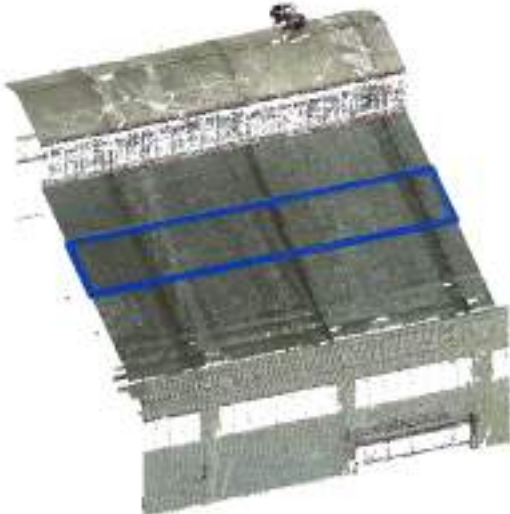
With a **Sample width** of 5 cm you only get a segment of the roof area. It would, however, have been much better to be able to model the whole roof area.
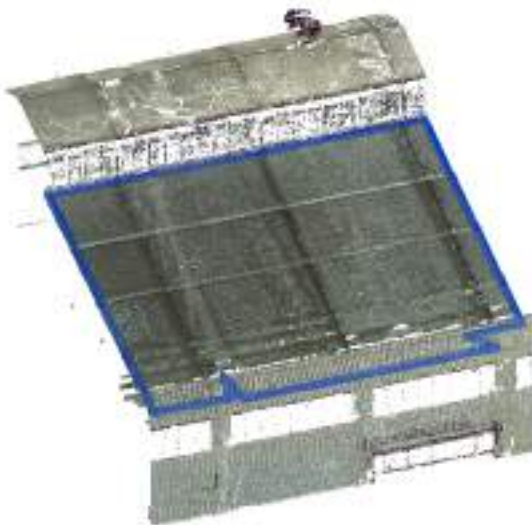
Should you use a **Sample width** of 20 cm, then it would have been extended over the swells and you have the whole roof area.

As well as the property of extending over holes and swells, the **Sample width** parameter also influences the speed of finding the contour. If you use a large value then you will quickly get a plane sketch. The minimum value of the **Sample width** setting is determined by the **Average point distance** setting, the maximum value is determined by the current **Smoothness** setting. To get a higher value, the value of the **Smoothness** setting must first be increased.

Model - Fit Polygon

Used in:

Project

With this command it is possible to fit a polygon or line chain composed of model lines onto a point cloud:
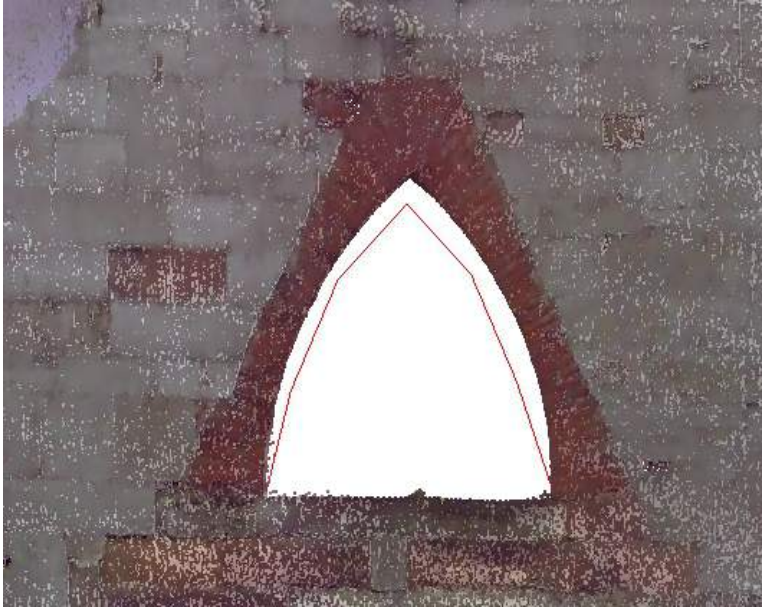
before

(Initial polygon)
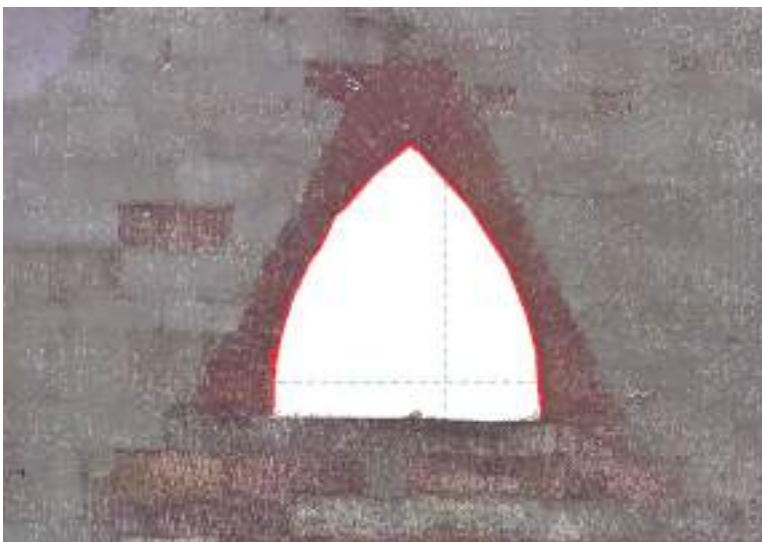
after

(Fitted polygon)
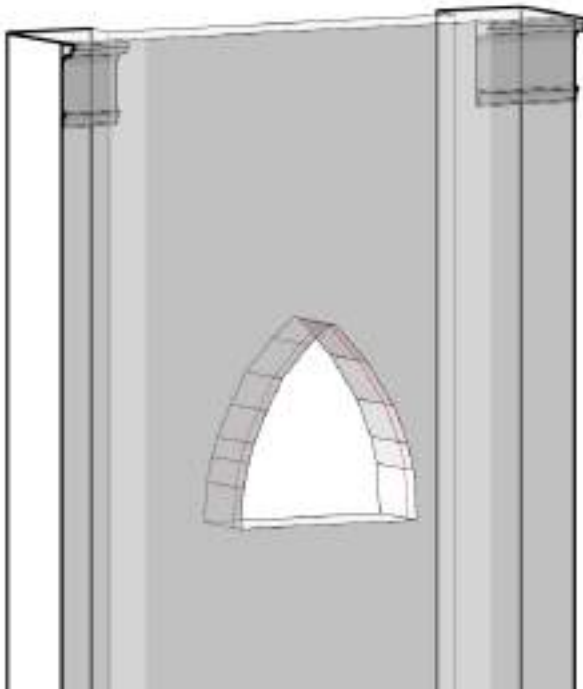
The fitted polygon can be used, for example, to modify the profile of a wall, or for the modelling of an opening.

To do this select the wall then select "Modify profile" ▨ (tab Modify | Walls → panel Mode). In the sketch mode for the wall profile you can now select the fitted polygon using the command "Select lines" ▨.

So that the line segments can be recognised as connected line chains / polygon chains, they must lie on the same sketch plane. The sketch plane corresponds to the work plane, which was active when the lines were created. Such a work plane can be created, for example, with the help of the command → **Work Plane Tools - Fit Plane (n Points)**.

The point cloud region in front of and behind the sketch plane of the line chains / polygon chains must be linear (and not planar). This can be checked, by setting the Revit section box just a few centimetres around the plane. For the above example the reveals of the opening must cross the sketch plane of the polygon chain, that is the reveal, and must not lie on the façade plane. A 3D clipped region that lies just a few centimetres away from the work plane will result in the following image:
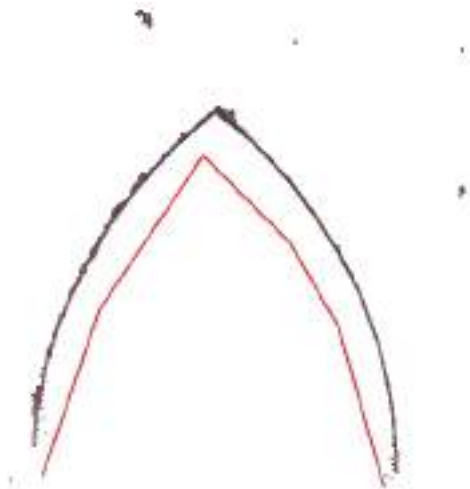
If a line chain be fitted instead of a closed polygon, the start and end positions will be retained. In this situation you should best pick these end points directly in the point cloud.

Please avoid using line segments in regions where there are no, or only very few, point cloud points. This can lead to unexpected results during the fitting. In the example shown above it was the sill edge of the opening that was the case. Because of the scan shadow there are no point cloud points to be found there.

After starting the command "Fit Polygon" you will see the following prompt:

Select a segment of the connected lines   *Select an appropriate segment of a con-*
                                                   nected line chain, that you want to fit onto
                                                   the point cloud.

As-Built for Revit Automatically finds all of the connected line segments on the same sketch plane and highlights the line chain. The following dialogue box opens:
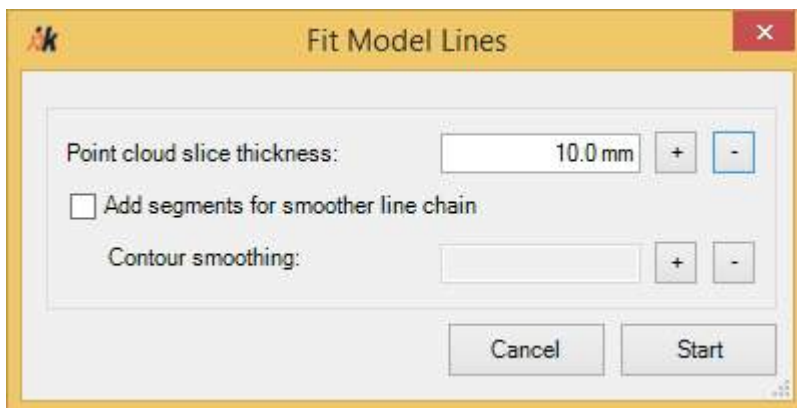
If the option **"Add segments for smoother line chain"** has not been activated, then after clicking the **[Start]** button the line chain will be fitted using the same number of line segments (in the example it was 6 line segments).
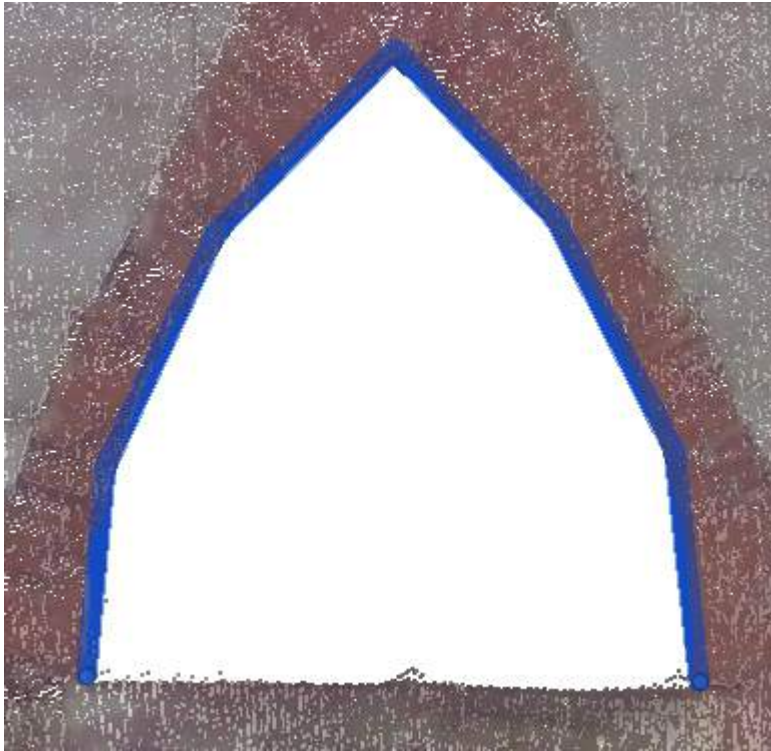
If the option **"Add segments for smoother line chain"** has been activated, then line segments will be inserted to give a better fit to the point cloud:
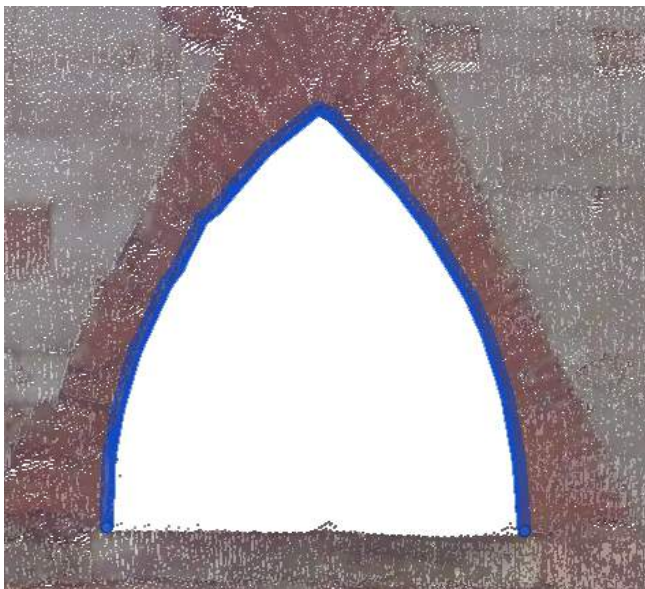
With the help of the parameter **"Contour smoothing"** you can control for this case how many line segments will be inserted. If the value is set small, then the polygon chain will be very detailed. If the value is set large, then the line segments will be very straight-lined against the point cloud.

The following example illustrates this:
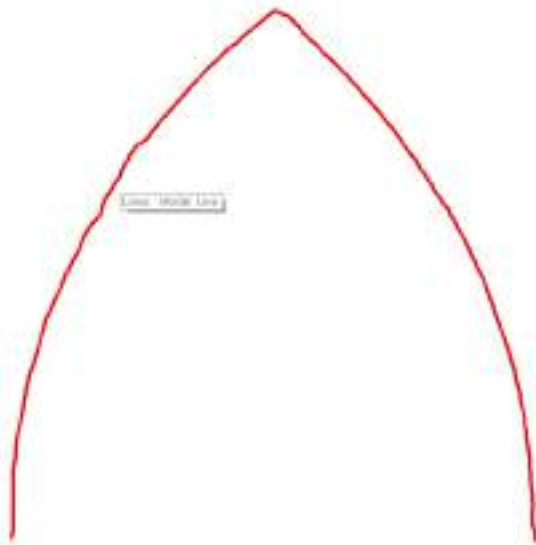
small **Contour smoothing** with 1 mm. Very short line segments are created.
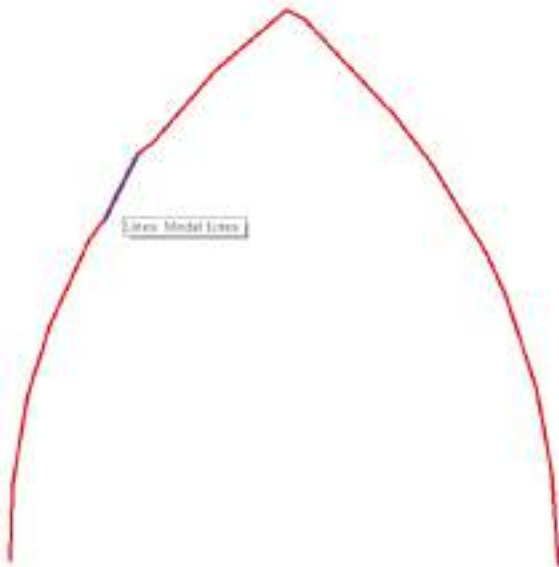
Contour smoothing of 10 mm. The line segments will be longer and more straight-lined.

With the parameter **"Point cloud slice thickness"** you control how many points will be used that lie in front of and behind the sketch plane of the line chains / polygon chains. The slice should be thick enough so that there are sufficient points available for the fitting. The slice should also not be so thick that they do not protrude into other profiles. For example, should the slice in the above window reveal not intersect with the façade surface:
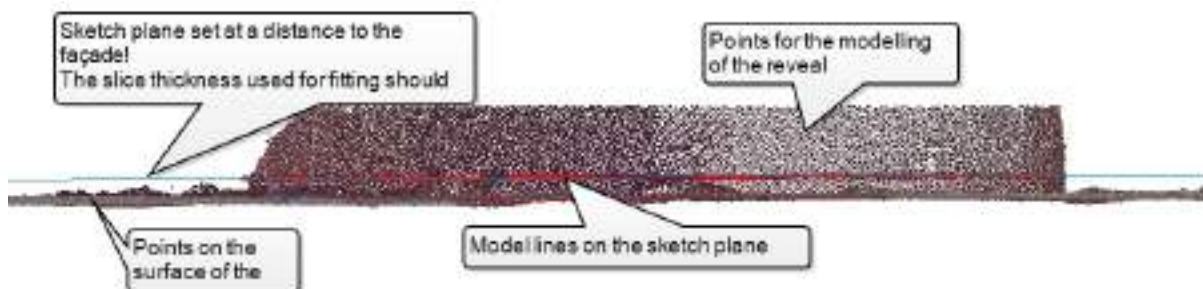
So that a fitted polygon created in such a way can be afterwards also be used as a construction aid for loadable families. With the help of the command ➔ **Model - Lines to Family**you can save the resulting model lines as 2D or 3D families.

Model - Lines to Family

Used in:

Project

With this command you can save as many model lines as you like as 2D or 3D families. In the families editor you can then use the model lines as construction aids for further modelling.

The following example demonstrates the use of the command:

A line chain is created in the point cloud and then fitted to the scan data with the command Polygonfit.
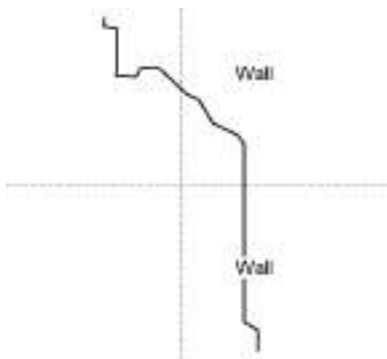
The line chain is saved as profile family with the command "Lines to Family". This basis can now be extended in the families editor into a closed profile.
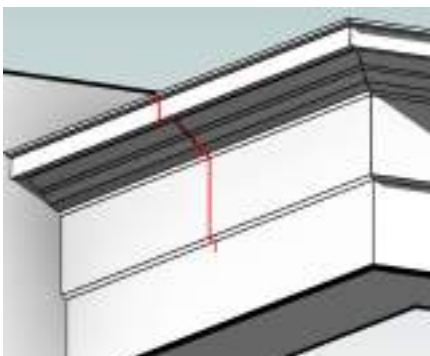
With the Revit command "Wall Sweep" the so created family can be used for modelling, for example for cornices along walls.

After calling the command the following dialogue box is displayed:
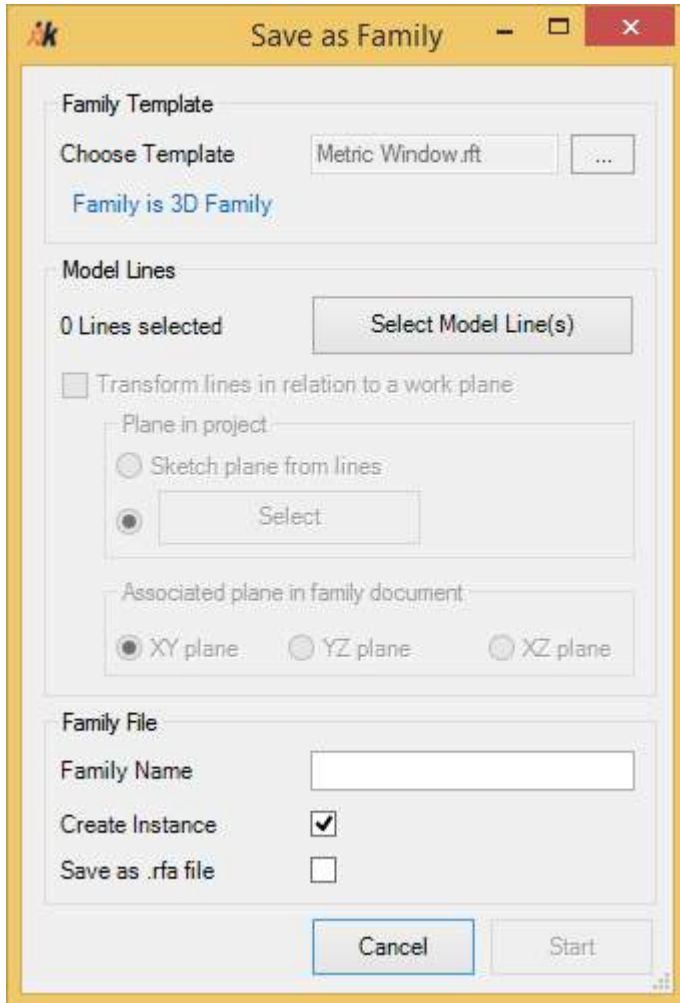
Now select a family template by clicking the button **[…]**. If a valid template has been selected it will be shown in the dialogue box whether it is a 2D or 3D family.

Click the button [Select Model Line(s)] to select the model lines. The dialogue box is temporarily hidden and the following prompt appears in the Revit status bar:

Select one or more model lines	*When it is a 3D family, you can select multiple model lines with any orientation that you like. Confirm the selection with the button [Finish].*
Select a segment from the connected model lines	*When it is a 2D family, you only need to select a single line segment. All of the lines connected to the segment, that are on the same sketch plane, will be also automatically selected.*

After completing the selection the number of model lines selected will be displayed in the dialogue box. Enter a name for the family in the "Family Name" text box and confirm the dialogue box by clicking **[Start]**. The family will now be created and then immediately loaded

into the project. The family is then to be found in the appropriate category in the Project Browser.

If the option **"Create Instance"** has been selected, As-Built for Revit tries to insert an appropriate instance of the newly created family on the same position as the model lines. This options is only selectable, if it is a 3D family. It is not always possible, it depends on the family template selected. In this situation an appropriate warning message is displayed.

If the option **"Save as .rfa file"** has been selected, the new family is also saved as a file. The appropriate "Save" dialogue box is then displayed.

More options

Should a 3D family have been created, you can with the option **"Transform lines with reference to the work plane"** use a rotation to a reference plane on the selected lines. To do this first in the frame "Planes in Project" select the reference plane in your current project. If all of the selected the lines lie on a sketch plane, then the option "Sketch plane of the lines" is selectable. Otherwise by clicking the button **"Select"** any model plane can be selected. The following dialogue box appears:
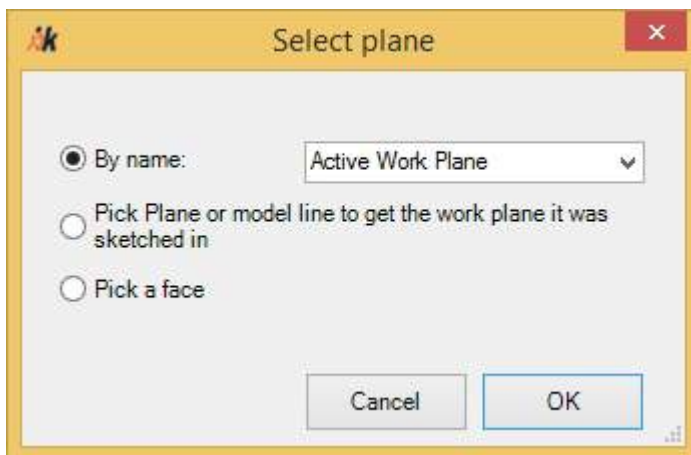
After selecting the model plane define in the area "Assigned planes in the families document" with the appropriate alignment, in which the model lines should be brought.

An example:

The following figure shows the point cloud of a window in floor plan (left) and 3D view (right). This window should be modelled as a family. For this purpose model lines have already been drawn as construction aids in the point cloud. The red line has been created with the command → → **Model - Fit Polygon**, the blue lines with the Revit line command on a previously fitted work plane:
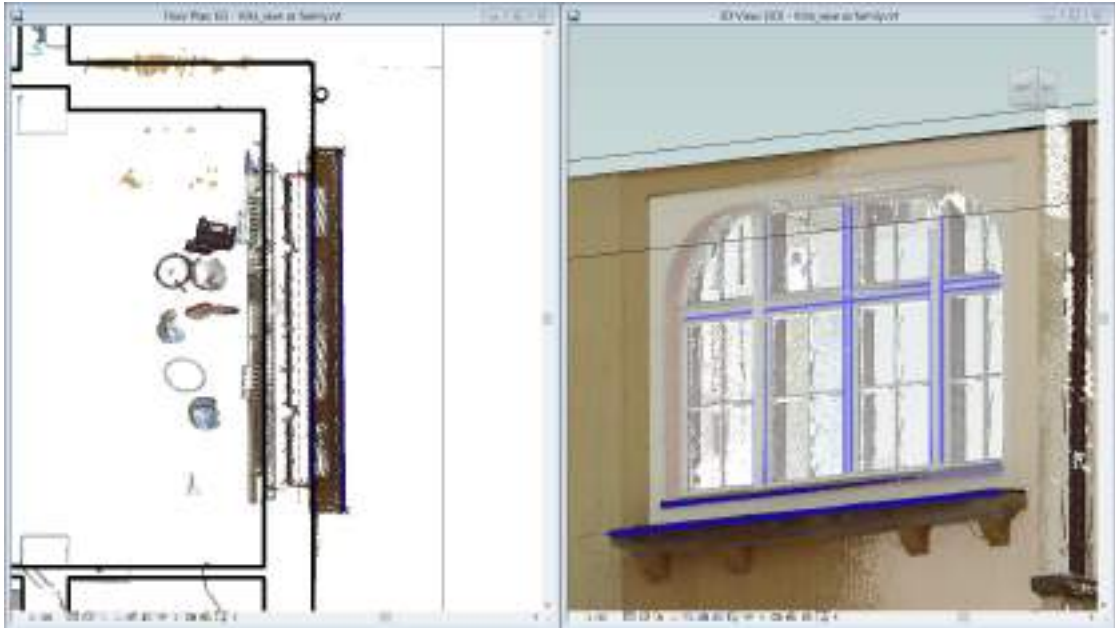
The exterior wall has been already modelled and runs in an approximately north-south direction (Y-axis in the Revit project coordinate system). If the command "Lines to Family" has been used without addition transformation of the family template "M_ Fenster.rte", the results in the families editor will look like this:
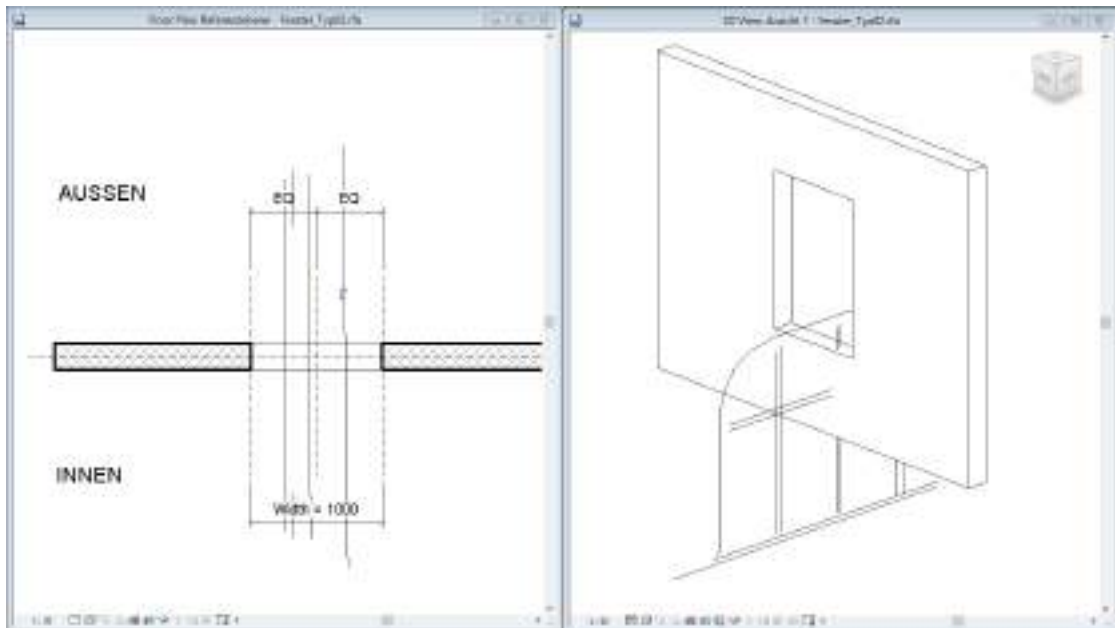
You can see here that the lines have been inserted into the family with their associated alignment (that is along the Y-axis). But the desired result was that the lines should have been aligned along the wall in the families document. This can be achieved with the option **"Transform lines with reference to the work plane"**.

The external wall face can be selected as the reference plane and as associated plane in the families document the X-Z plane:
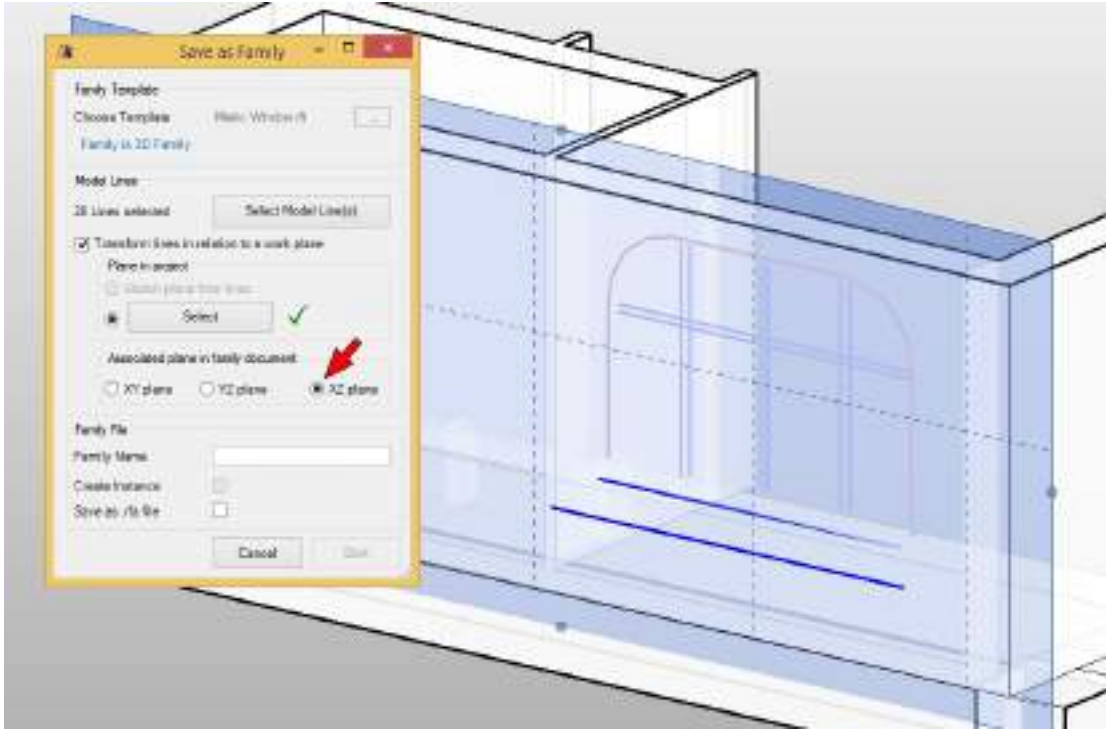
The resulting family now look like this:
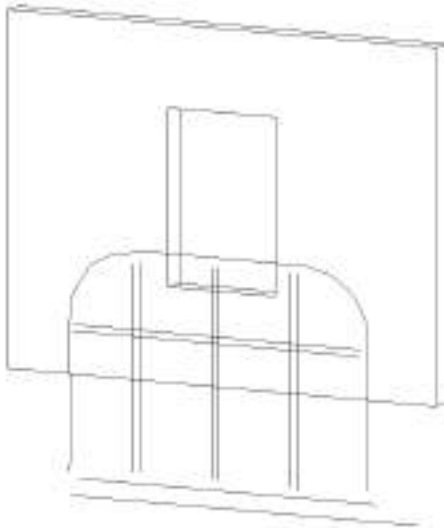
The model lines can now be moved to the correct position, so that they can be used as construction aids. Under certain conditions it can be necessary to first form the lines into a group (select all lines at one time → Tab: Modify→ Create Group → Create Group

), to be able to move them all together.

The following figure shows the completed family after modelling:
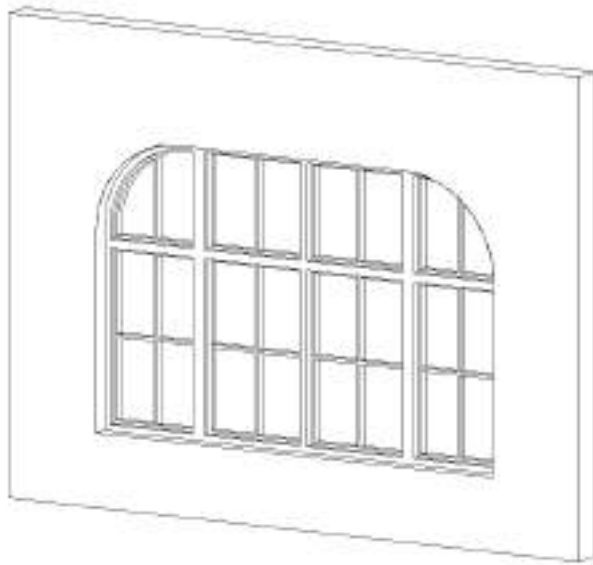
If a transformation has been applied to the selected lines, the option **"Create Instance"** is not available.

Model - Intersect 2 Planes

Used in:

Project   Family

With this command you can intersect two planes with each other, the result is a model line. As a plane you can select the active plane, Revit planes, reference planes, model lines with their work plane or planar surfaces.

The following figure shows an example, of how you can create the eaves edge (green model lines) from modelled roof surfaces (blue model lines), created with the command ➔ **Work Plane Tools - Fit Plane (n Points)**) by intersecting them with the Revit plane of the roof:
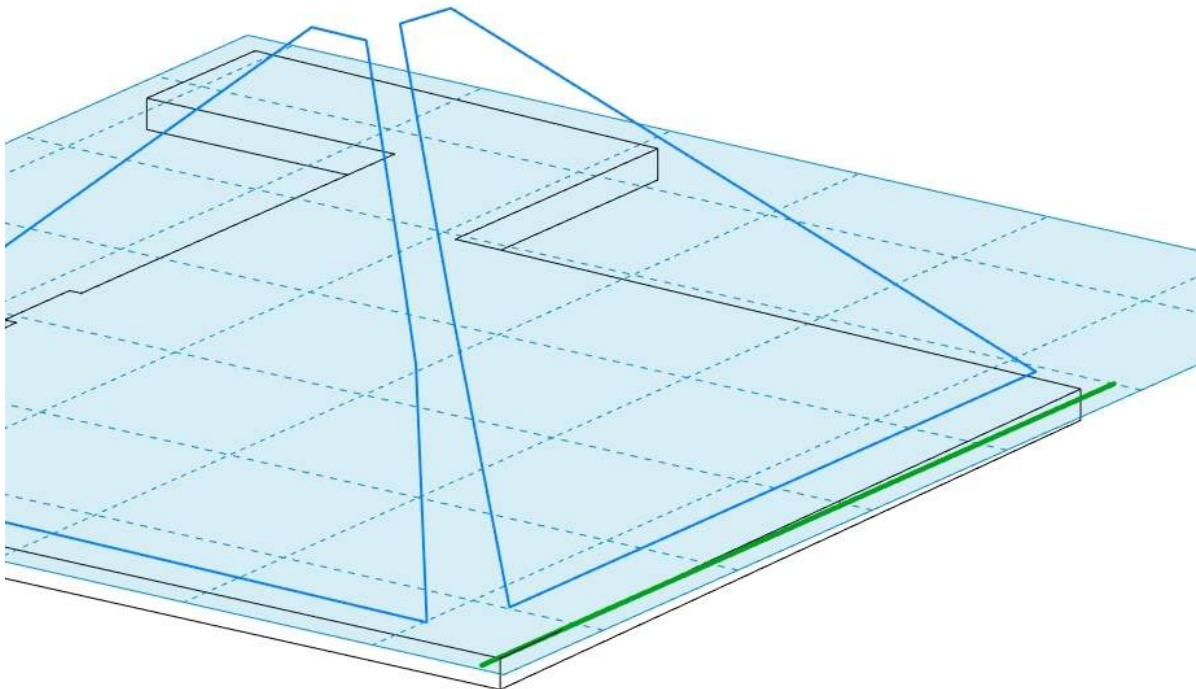
As soon as you start the command a dialogue box appears, in which you are prompted to select the first plane:
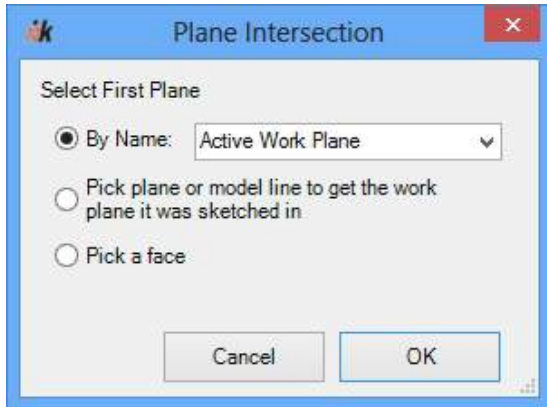
Selection option	Explanation
"Name"	Here you can select the active work plane, Revit planes or named reference planes.
"Pick plane or model line to get the work plane it was sketched in"	Use this option, if you want to select a reference plane, a Revit plane or a model line with its work plane in the currently open view.
"Pick a face"	Use this option, if you want to select as reference plane, a planar surface or the active work plane in the currently open view.

Confirm your selection by clicking **[OK]**. In the case of "Pick plane or model line to get the work plane it was sketched in" and "Pick a face" you will be prompted in your open view to click on the appropriate choice.

As soon as you have done this, the selected plane will be set as the active plane and then another dialogue box opens, that now asks for the selection method for your second plane:
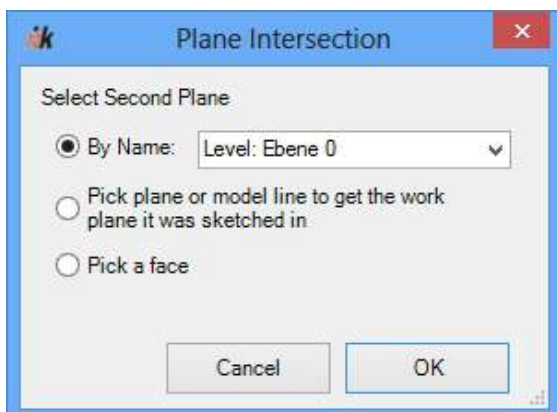
As soon as you have selected the second plane a model line is created that is the result of the plane intersection. The intersection line is at least 1 m long. The length of the intersection line is determined by the geometry of the selected planes.

Model - Intersect 3 Planes

Used in:

Project   Family

Analogue to the command → **Model - Intersect 2 Planes** but here three planes can be intersected with each other.

In contrast to the "Intersect 2 Planes" command, the result is a three to six leg figure of model lines or a construction point, respectively.

For example you can create with this command the verge and ridge lines from the modelled planes of a roof skin (green model lines) and a facade (black model lines):
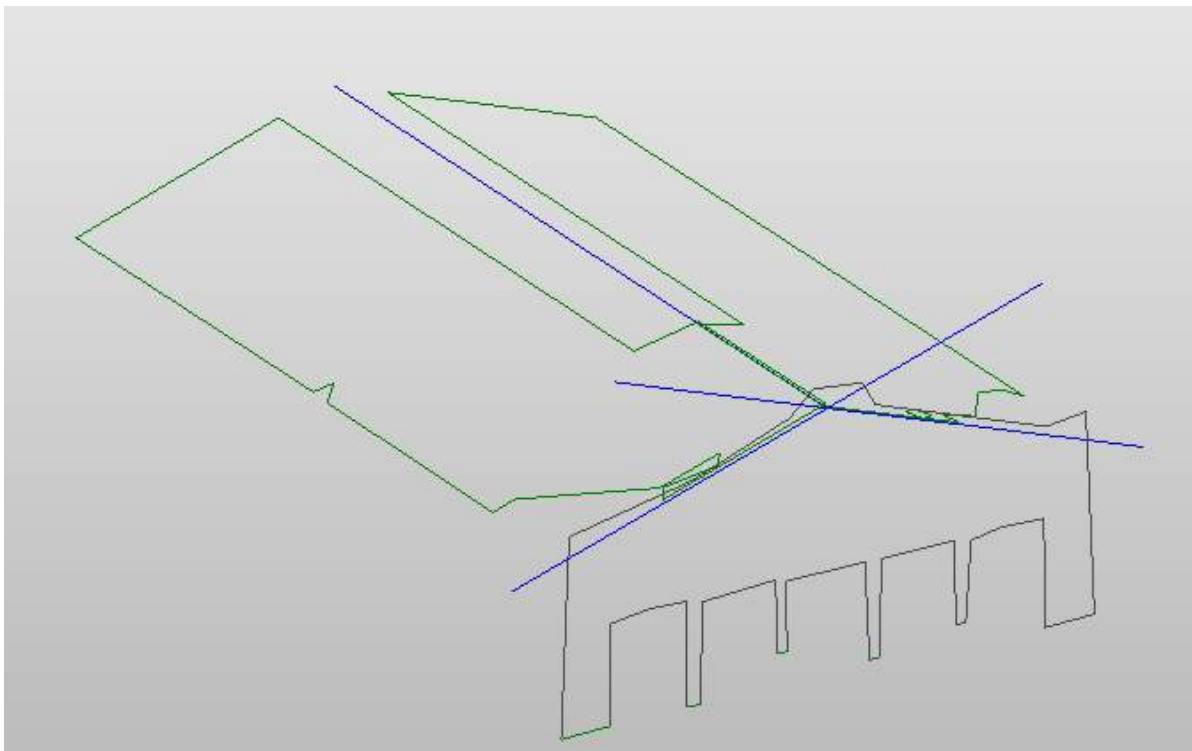
The command requests, analogous to the command → **Model - Intersect 2 Planes**, the selection of two planes. In contrast to the "Intersect 2 Planes" command, the command is not ended after the selection of the second plane, rather there appears another dialogue box prompting for the selection of a third plane and the type of the resulting geometry (Result):
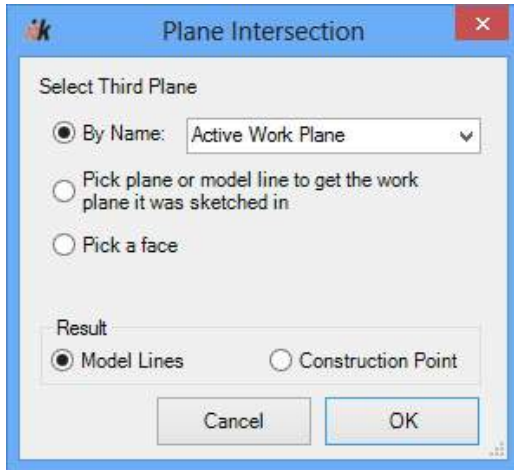
"Result" options	Explanation
"Model Lines"	Creates a three to six leg figure, analogous to the verge and ridge lines example. The length of the model lines is dependent on the selected geometry.
"Construction Point"	Creates a single construction point at the intersection point of the three planes

Regions - Manager

Used in:

Project

As-Built point cloud regions are clipped regions in the point cloud, which are independent of the views defined in the project, and are capable of being shown or hidden. The clipped regions will be saved as named regions in the Regions Manager and saved together with the project. It is at anytime possible and in every 2D or 3D view to access any one of the defined regions.

The command shows or hides the Point Cloud Regions Manager in the Revit user interface. The Regions Manager is used to control the user defined point cloud regions.

The tree structure in the upper part shows the hierarchically organised point cloud sections, whereas in the lower part it is the currently active clipped planes of the active point cloud section that are listed.
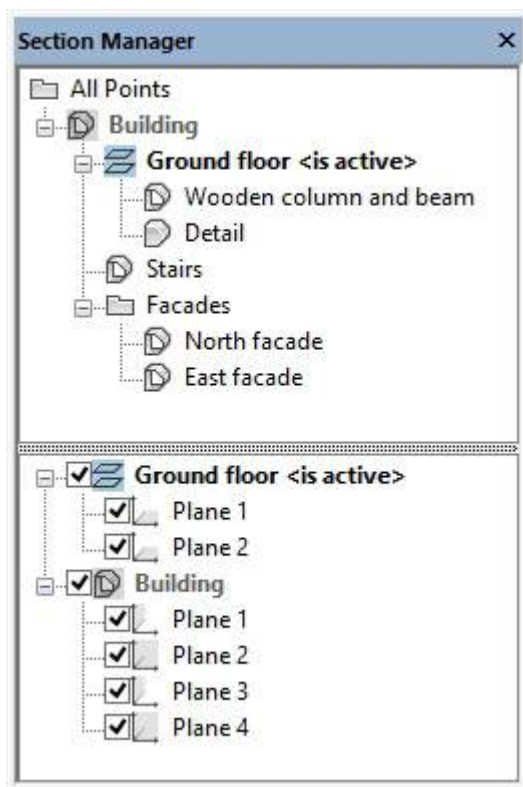
Point cloud regions can be individually displayed in Revit by using a range of oriented levels. Only the point cloud points that lie on the positive side of all of the clipping planes will be displayed.

Each node and sub-node in the tree structure above displays a named number of such clipping planes. They can be combined and grouped together with each other. That means that every

node effectively inherits the active clipping planes of the higher-ranking branches in the tree structure. There can only ever be one node that is active. The region that is displayed is the one defined by the planes of the active node and those of its parent nodes.

For special use cases the clipped planes can be switched active or inactive, either singly or in groups, by using the option switches in the lower area. In this way you can create a structural order and a working structure for point clouds. A double click on a node in the above tree structure will activate this named point cloud region, so one can, for example, isolate the previously defined and so named ground floor. Elements can be moved around in the upper tree using drag and drop The moved elements will be copied by the simultaneous pressing of the [Ctrl] key. If one hovers for a while with the mouse pointer over a tree node, an optionally defined user tool-tip text for the point cloud region will be displayed.
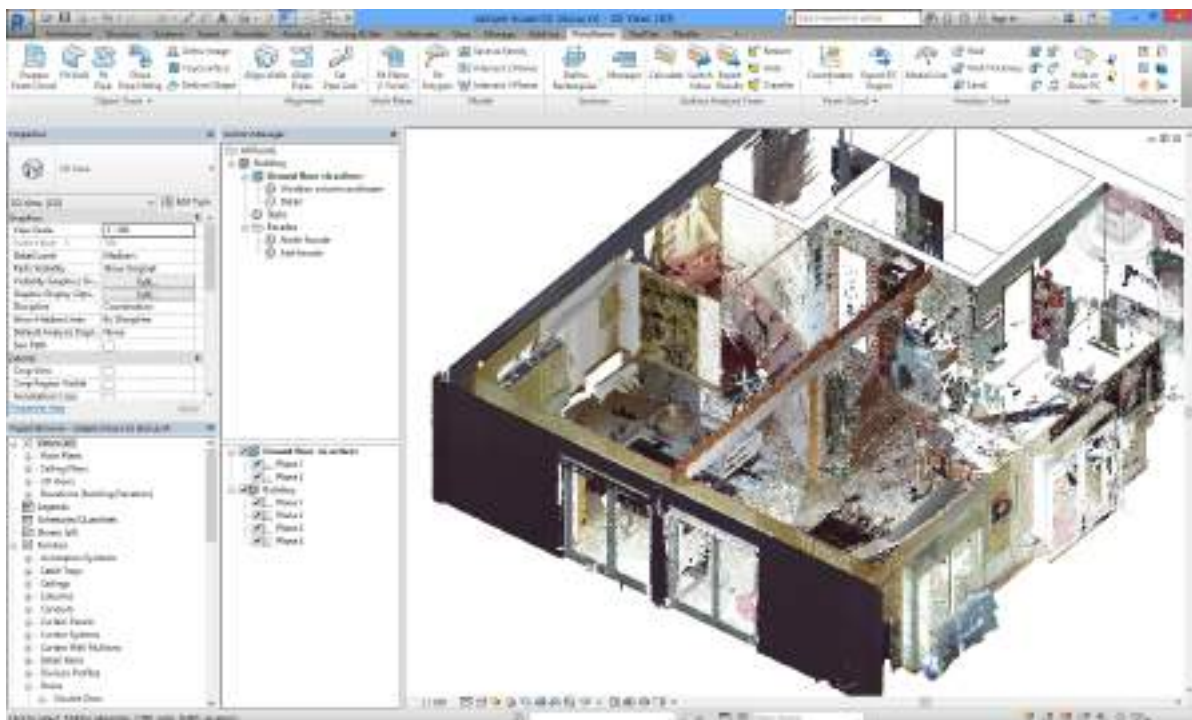
The icons before the text in the tree structure above correspond to their geometrical region property:

Icon	Description
	Named tree nodes without any clipping planes. These are used exclusively for the organisation of the child nodes.
	Slice defined by two parallel clipping planes
	Cuboid cropped region
	Prismatic cropped region

Icon	Description
	Any complex cropped region

The active clipping planes of the active region are listed in the lower area. These can be activated or deactivated, as required. The icons in this tree show the geometrical position of the planes. If one hovers for a while with the mouse pointer over one of the planes, a tool-tip is displayed with its coordinates in the Hessian plane normal form.

Icon	Description
	Skewed planes
	Any vertically placed plane
	Horizontal plane (XY)
	Axis parallel plane (YZ)
	Axis parallel plane (XZ)

If one of the tree elements in the upper tree is right clicked, the following context menu is displayed:
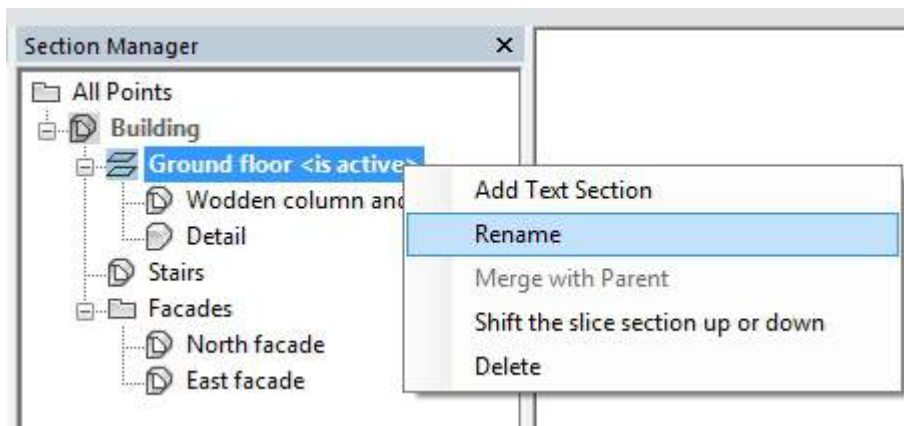
Here you have the following options at your disposal:

■ Add Text Section

Creates a new node in the tree without any clipping planes. A dialogue box is displayed for

the definition of the name and the tool-tip.

- **Rename**

  A dialogue box is displayed for the definition of the name and the tool-tip for this node.

- **Merge with Parent**

  The clipping planes of the node will be merged with the parent node. This option is only enabled when node doesn't have any sub-nodes.

- **Shift the slice section up or down**

  A dialogue box is opened where the slice can be moved along its alignment normal by an adjustable amount.
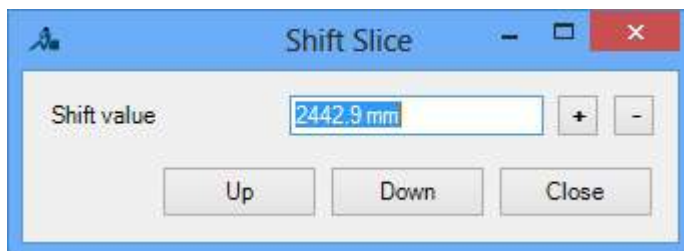
- **Delete**

  Deletes this node from the tree.

Sections - Rectangular

Used in:

Project

This command starts the creation of a new rectangular point cloud section. The following prompt is displayed:

```
Please draw the rectangular limits of
your desired section in the point cloud
```

With the left mouse button drag a rectangular section over the point cloud.

Finally the following dialogue box is displayed, in which you can give the new section a name. Optionally, a tool-tip description can be added, which appears in the section manager when the mouse pointer hovers for a while over this entry. In the situation where there are no other subordinate point cloud regions next to this new region in the tree structure, the option "Skip name and merge with parent" is enabled. Hereby the newly defined clipping planes are merged directly with the higher level section.

Regions - 3D Clipped Region

Used in:

Project

This command starts the creation of a new point cloud region from the current 3D clipped region of the active 3D view.

The command requires a 3D view with an active clipped region, otherwise you will get warning message. Start the command after the 3D clipped region has been set up as required. The limits will be taken over as the new clipped region.

Finally the following dialogue box is displayed, in which you can give the new region a name. Optionally, a tool-tip description can be added, which appears in the section manager when the mouse pointer hovers for a while over this entry. In the situation where there are no other subordinate point cloud regions next to this new region in the tree structure, the option "Skip name and merge with parent" is enabled. Hereby the newly defined clipping planes are merged directly with the higher level section.

Sections- Polygon

Used in:

Project

This command starts the creation of a new convex polygonal point cloud section. The following prompt is displayed:

```
Please select as many contour
points as you want
```

Pick any number of contour point in the point cloud which can be allocated to a convex section. After selecting the first point , a visualisation of the current section is displayed in the form of lines. During the input please avoid camera pans in the perspective window. After you have input all of your points, you can end further input by pressing [Esc].

Finally the following dialogue box is displayed, in which you can give the new section a name. Optionally, a tool-tip description can be added, which appears in the section manager when the mouse pointer hovers for a while over this entry. In the situation where there are no other subordinate point cloud regions next to this new region in the tree structure, the option "Skip name and merge with parent" is enabled. Hereby the newly defined clipping planes are merged directly with the higher level section.

Regions - Slice

Used in:

Project

This command starts the creation of a new slice point cloud region. The following dialogue box appears:
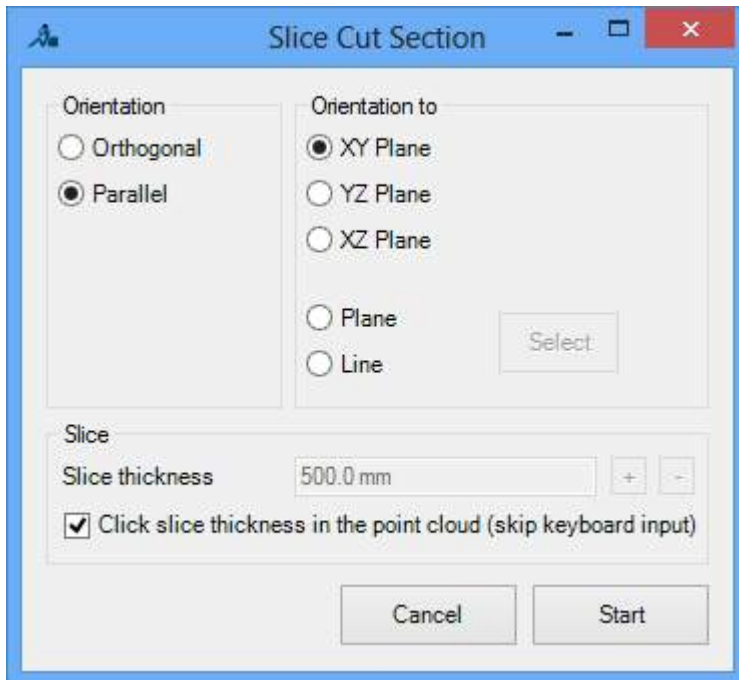
The orientation to the newly created slice is defined in this dialogue box. Likewise, the precise thickness of the slice can also be defined. If you have selected the option **"Click slice thickness in the point cloud"**, you will be prompted after clicking **[Start]** to pick two points in the point cloud. Otherwise it is sufficient to pick just one point in the point cloud, for the definition of the first clipping plane.

Finally the following dialogue box is displayed, in which you can give the new region a name. Optionally, a tool-tip description can be added, which appears in the section manager when the mouse pointer hovers for a while over this entry. In the situation where there are no other subordinate point cloud regions next to this new region in the tree structure, the option "Skip name and merge with parent" is enabled. Hereby the newly defined clipping planes are merged directly with the higher level section.

Surface Analysis Tools - Calculate

Used in:

Project

The surface analysis computes the deviation from the Revit model surface to the loaded point cloud and then visualises the results as a false colour image on each of the respective surfaces. The analysis results can be subsequently exported.
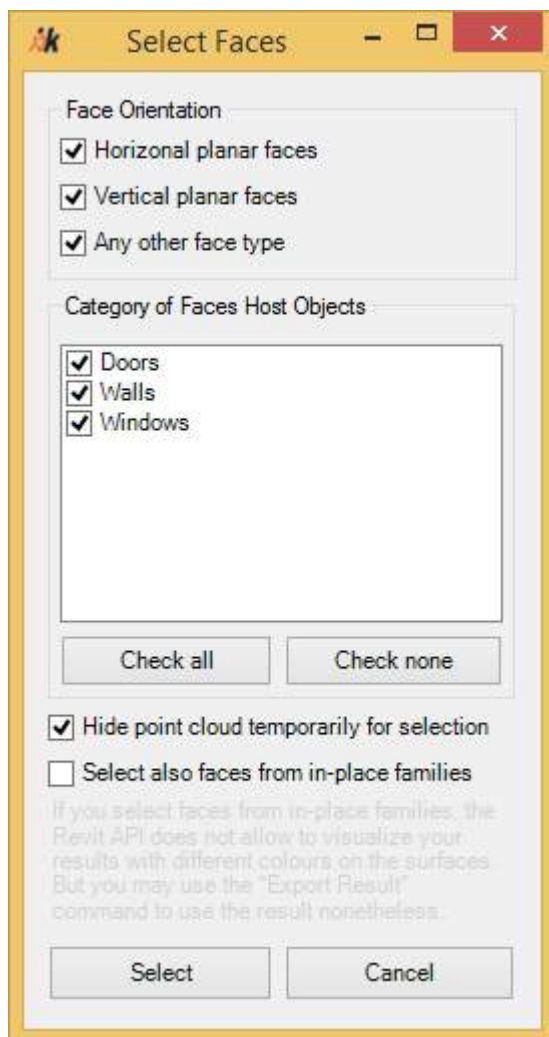
When the button "Surface Analysis" is clicked then first the dialogue box shown to the left is displayed. Hereby it can be controlled which type of faces in Revit can be selected for analysis.

The first selection group allows the selection of the faces with respect to their spatial orientation, that is, horizontal, vertical or all others.

In the next selection group the face affiliation to specific host components can be filtered. Here, all of the component categories are displayed that are in the project. The buttons **"Select all"** and **"Deselect all"** allow the selection or deselection of all categories.

With the next option "Hide point cloud temporarily for selection" the point cloud can be temporarily hidden for selection.

The last option **"Select faces from in-place families"** should be used with care. These faces can be analysed, but the results cannot be visualised, because of restrictions in the Revit API. Having said that, the analysis results can be exported and used again. Click the button **[Select]** to select the desired faces. The status

line changes to:

```
Please select multiple surfaces.
```

Please select one or more surfaces.
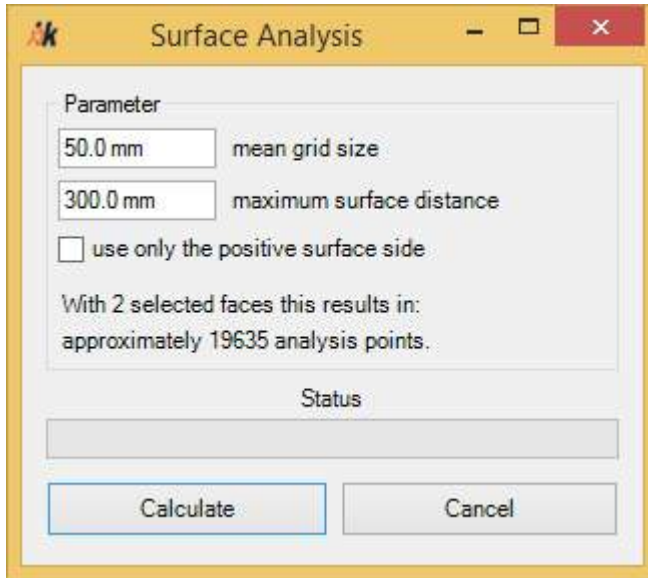
After the selection of the surfaces, the dialogue box on the left is displayed with the analysis parameters. The analysis is carried out on the basis of a defined grid on the individual surfaces that have been selected. The **"mean grid size"** defines the width and height of each cell of the grid. On curved surfaces these values are used in the centre of the surface, but because of the curvature they will vary at the edges of the surface.

After changes have been made to the grid size, the information to how many analysis points there approximately are on the number of selected surfaces is also changed.

The parameter **"maximum surface distance"** defines the corridor around the surface, in which points from the point cloud should be considered for the deviation analysis. The option **"use only the positive surface side"** permits only point cloud deviations on the outer face of the surface to be considered.

Now click the button **[Calculate]** to start the surface analysis, Your result can then look something like this:
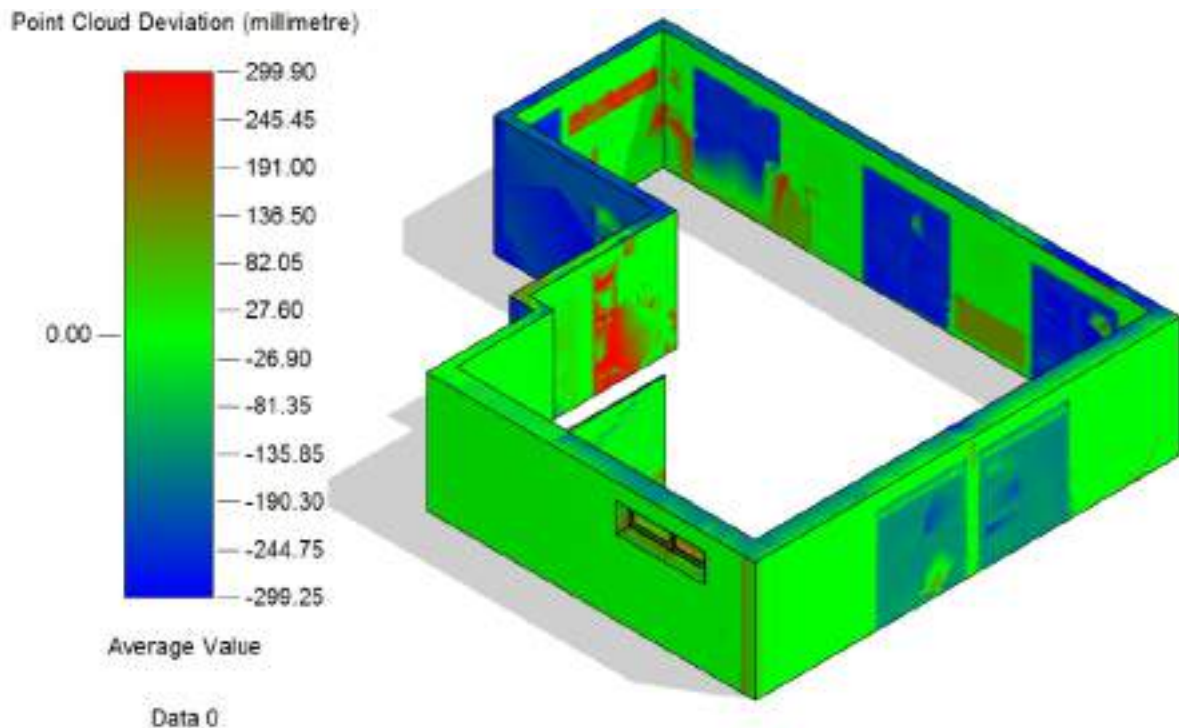
The analysis value "Average value" is the default value, and this represents the averaged point cloud deviation. There are a further 4 analysis values that can be visualised, which can be displayed with the function → **Surface Analysis Tools - Switch Results**

The colour gradation of the analysis as well as the appearance of the legend can be controlled by the Revit Analysis Display Styles (tab Manage → panel Settings → drop down list Additional settings → Analysis Display Styles 🎨):
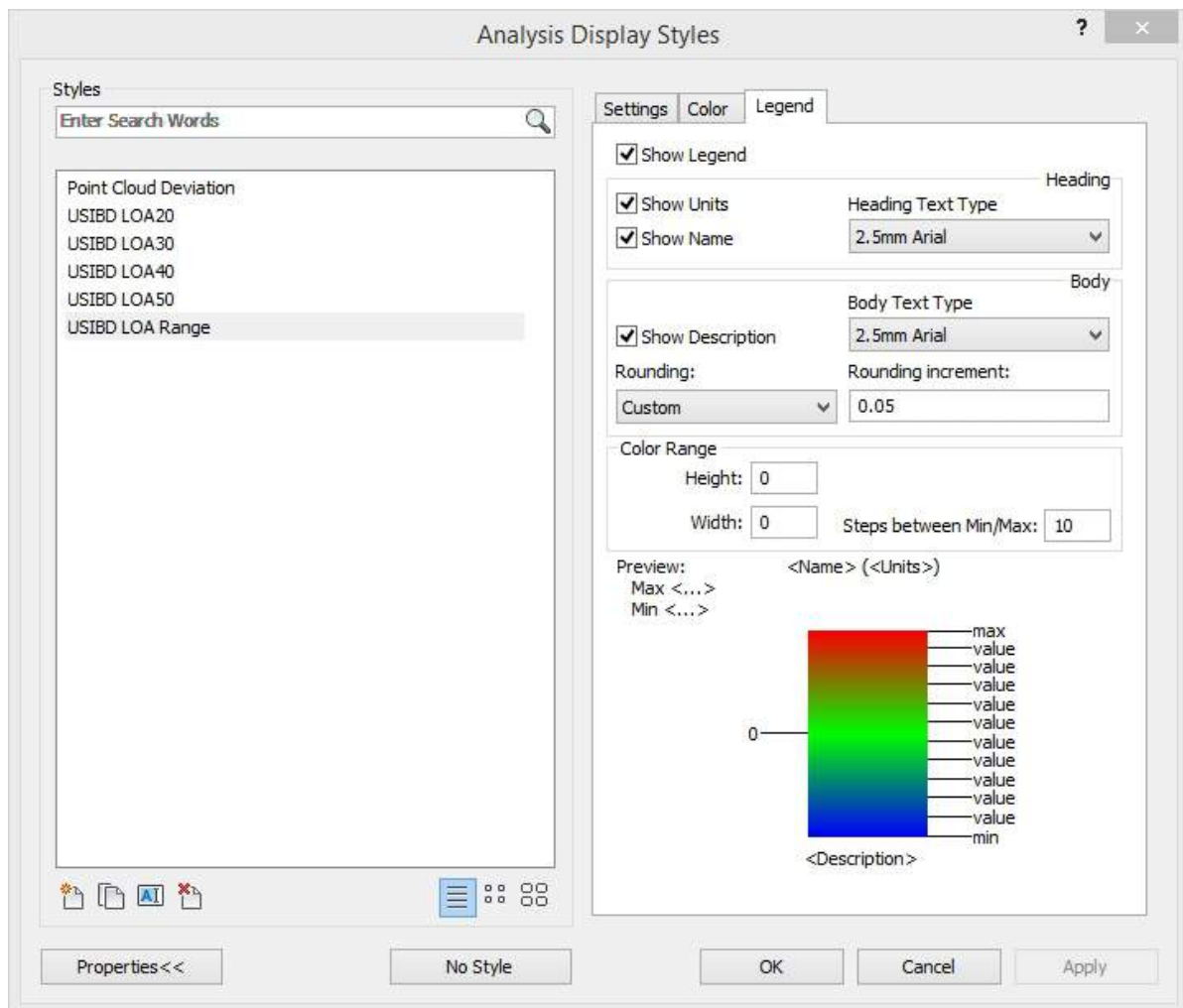
Neben dem Standardstil "Punktwolkenabweichung", erstellt As-Built for Revit weitere Stile zur Darstellung der Abweichung entsprechend der "Level of Accuracy (LOA)", wie sie durch die USIBD definiert werden (http://usibd.com). Die Stile "USIBD LOA 20- 50" färben die gewählten Flächen in den Farben Rot und Grün ein. Grüne Bereiche liegen innerhalb der durch die LOA definierten Toleranz und rote Bereiche außerhalb. Der Stil "USIBD LOA Bereich" zeigt alle "Level of Accuracy" in unterschiedlicher Farbabstufung. Hierfür kann mit dem Befehl → **Surface Analysis Tools - Place USIBD Legend** auch eine Legende in einer Planansicht eingefügt werden.

The visualisation of the analysis results are not saved in the project file by Revit! However at least As-Built for Revit saves the computation results, so that you can, when reopening the project, reload the visualisation with the command → **Surface Analysis Tools - Restore**.

To transfer the visualisation of the computation results to another view, use the command → **Surface Analysis Tools - Transfer**.

Surface Analysis Tools - Switch Results

Used in:

Project
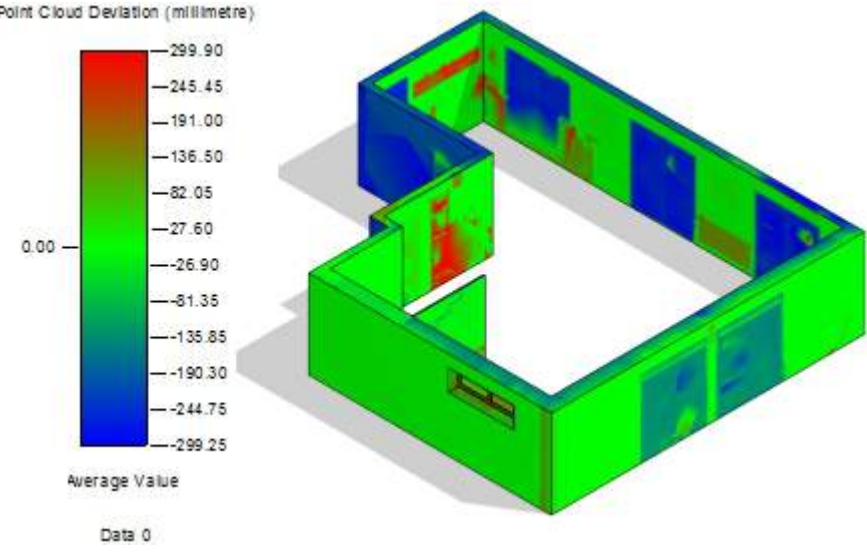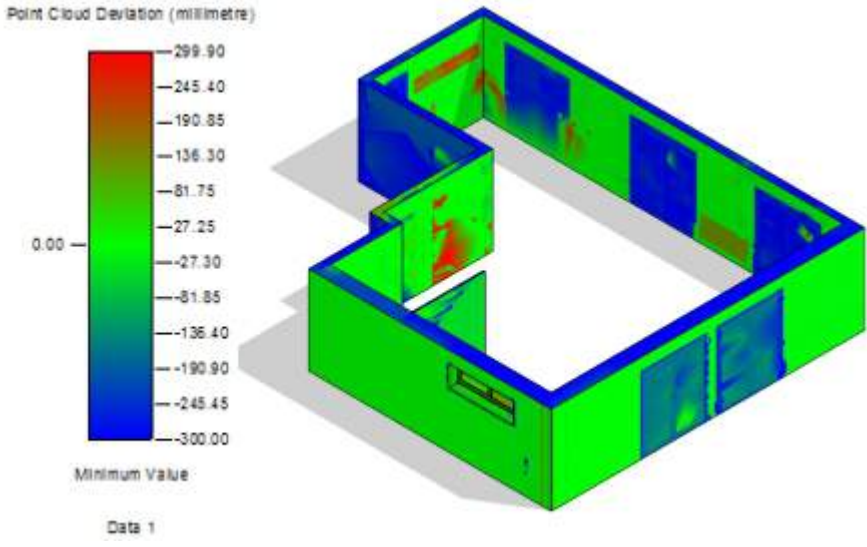
This function serves to switch the visualisation of surface analysis results. To compute a surface analysis, first run the command → **Surface Analysis Tools - Calculate**.
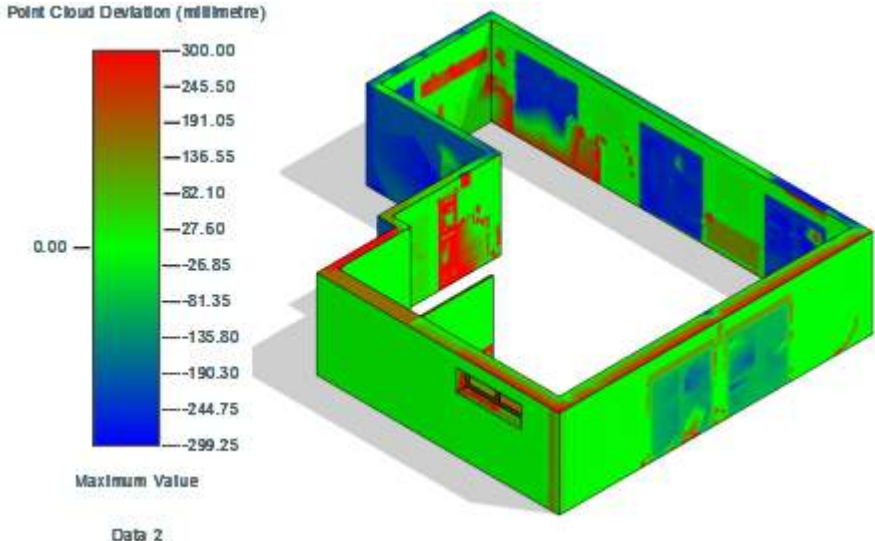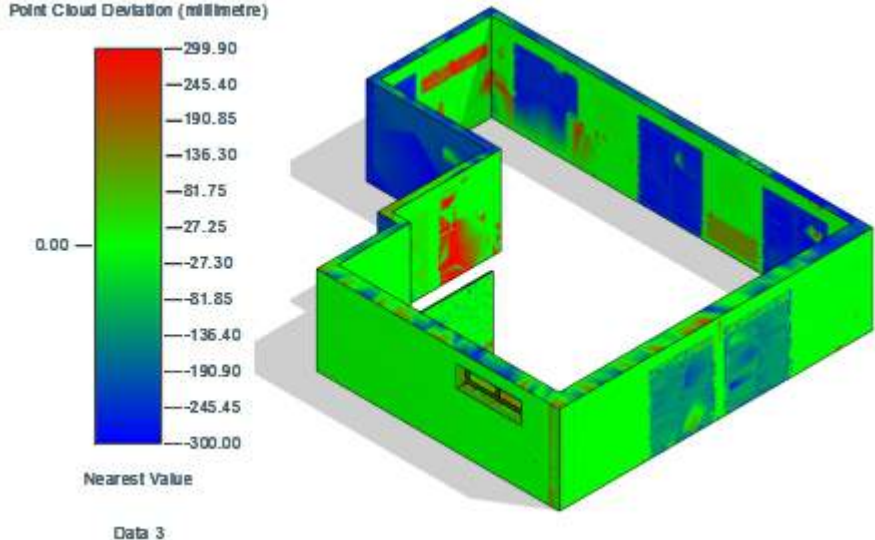
After clicking the button it switches to the nearest analysis values and the legend is updated. In a surface analysis there are always 5 different values calculated per grid cell:
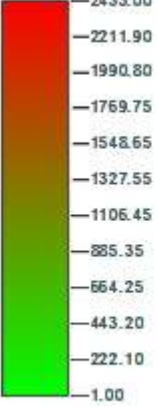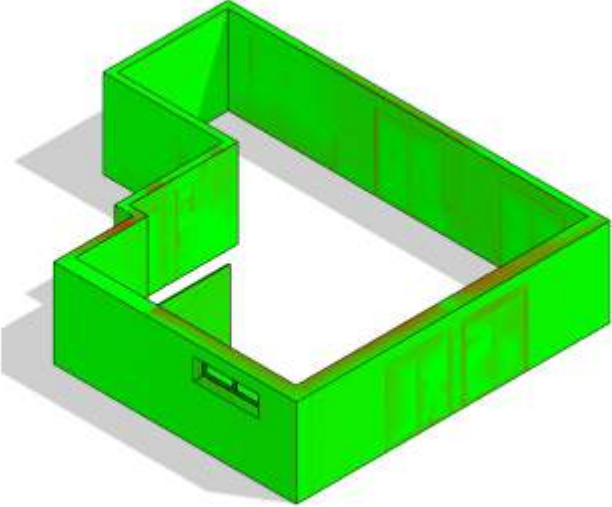
- **Average Value** - The average distance between all point cloud points in this cell.
- **Minimum Value** - The smallest absolute deviation within the grid cell. Positive values lie outside of the component, negative values inside of the component.
- **Maximum Value** - The largest absolute deviation within the grid cell. Positive values lie outside of the component, negative values inside of the component.
- **Nearest Value** - The distance used is that of the point that when projected onto the surface is closest to the centre of the grid cell.
- **Number of Points** - The number of all of the points found in this cell.
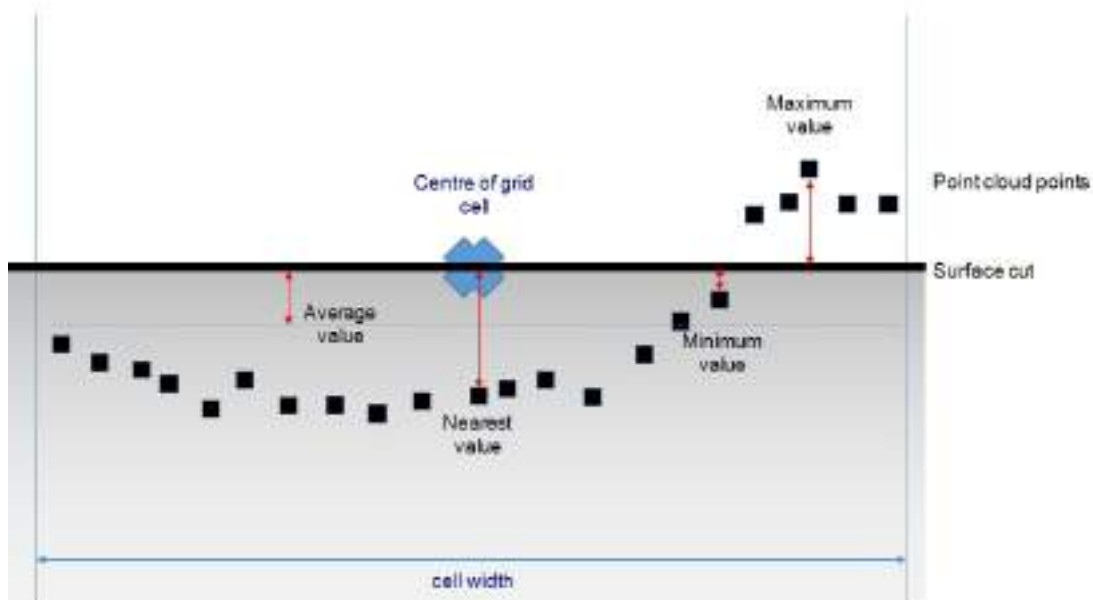
In the following tables all of the possible visualisations are shown, with one example for each:

Analysis Values	Example
Average value	
Minimum value	

Analysis Values	Example
Maximum value	
Nearest value	

Analysis Values	Example
Number of points	

Point Cloud Deviation (points)

- 2433.00
- 2211.90
- 1990.80
- 1769.75
- 1548.65
- 1327.55
- 1106.45
- 885.35
- 664.25
- 443.20
- 222.10
- 1.00

Point Number

Data 4

Surface Analysis Tools - Export Results

Used in:

Project

The function "Export Results" is used to export the analysis results, which had been previously computed with the command → **Surface Analysis Tools - Calculate**. As the first step select just the surfaces in the model, whose computed analysis results are to exported. The status line changes after starting the command to the following prompt:

```
Please select multiple surfaces.
```
Select one or more surfaces with surface analysis results for export.

Confirm the selection with the button [Finish]. The following dialogue box appears:
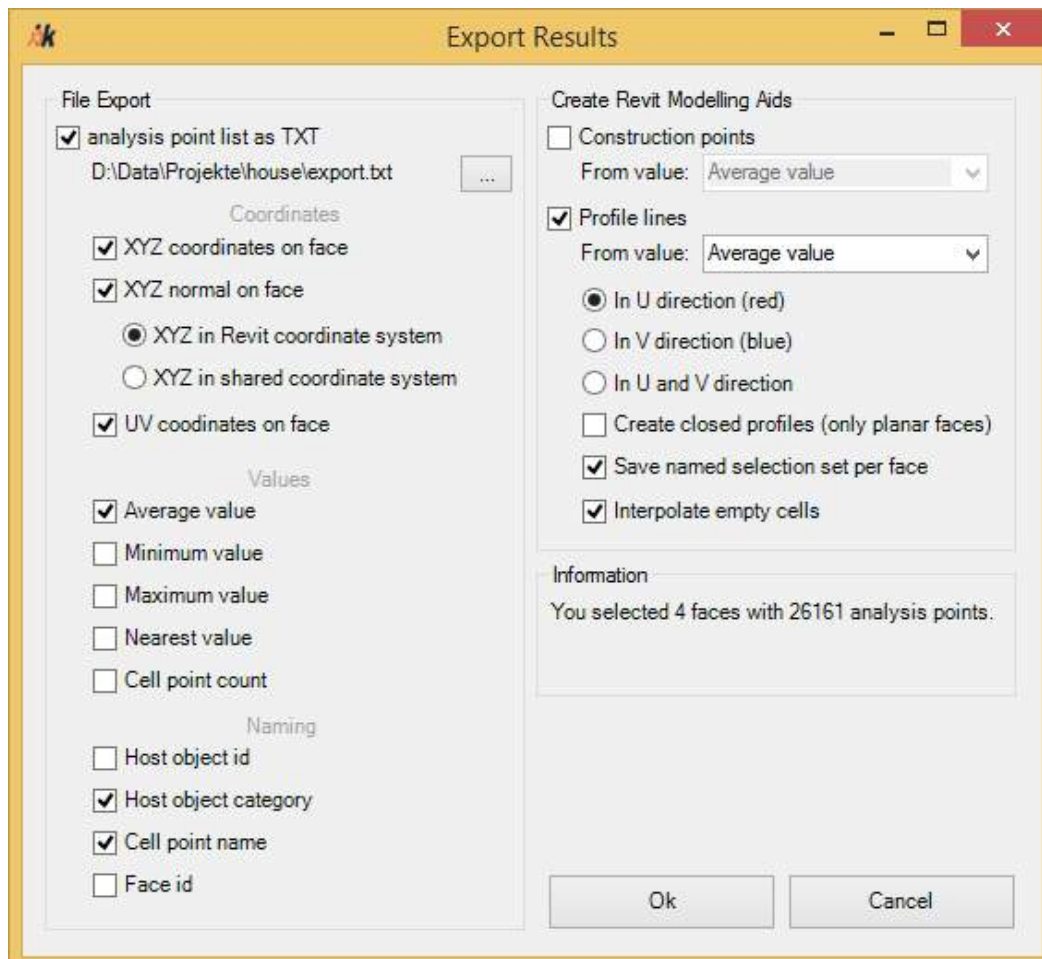
In the "File Export" group you can save the analysis results of the selected surfaces as a TXT file. The values of each cell point are separated by a semicolon and each cell point

appears on a separate line. The first line contains the description of the value, so that the file can, for example, be imported into Excel:

Now select a file name and location by clicking the button **[...]**. Then you can select the values that should be written to the file.

Value	Description
XYZ coordinates on face	The coordinates of the cell point of the surface
XYZ normal on face	The vector of the surface normal on the cell point
UV coordinates on face	The UV coordinates of the cell point of the surface in accordance with the Revit internal surface description
XYZ in the Revit coordinate system XYZ in the shared coordinate system	This option defines which coordinate system should be used for the export.
Average value	The averaged distance of all point cloud points in a cell
Minimum value	The distance of the lowest point value to the surface, this can also be negative
Maximum value	The distance of the highest point value to the surface
Nearest value	The distance used is that of the point that when projected onto the surface is closest to the centre of the grid cell.
Number of cell points	The number of all of the points found in this cell.
Host object ID	The ID of the host to which the surface cell belongs
Host component category	The category of the host component to which the surface

Value	Description
	cell belongs
Cell point name	The name of the cell point with the format: <Host ID>:<sequential surface number>:<sequential cell point number> E.g.: "305873:5:1"
Surface ID	The Revit internal ID for the surface to which the cell point belongs

In the "Create Revit Modelling Aids" group the analysis results can be also output as construction points or model lines. For both options you can define for each the value (averaged, minimum, maximum or nearest), which should be used for the export.

When you output the analysis results as construction points, the name of the cell point will be used each time for the name of the construction point.

Attention: Construction points in As-Built for Revit are instances of a loadable family. When you create a large number of points, this can lead to performance losses in Revit when regenerating the model.

There are several options available to you when creating profile lines:

■ **In U (red) or V (blue) direction**

You can create profile lines in differing directions (U and V) along the surface. As long as you are in the "Export Results" dialogue box, these directions will be displayed in the model as red and blue lines on the individual surfaces:
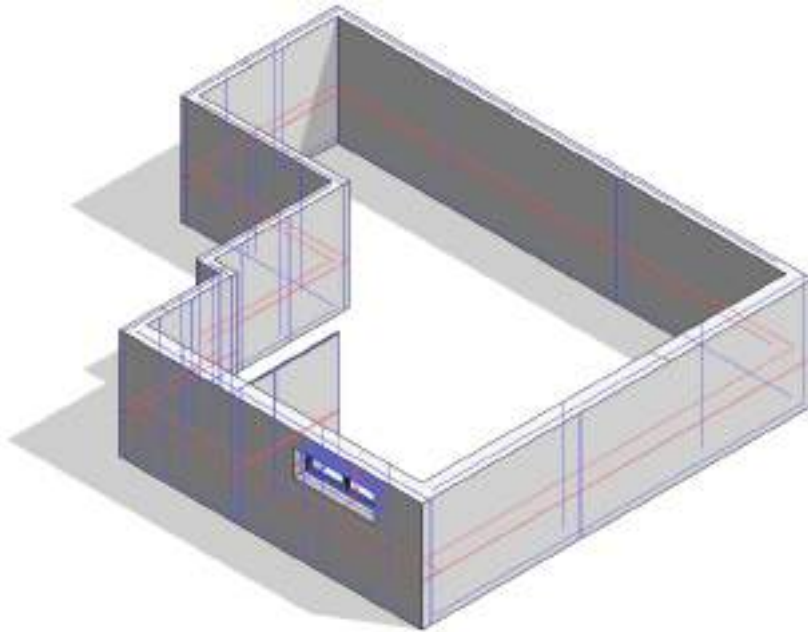
- **Create closed profiles (only planar faces)**

  If this option has been selected, the profile lines will be created as closed line chains. The start and end points are then closed along the surface:
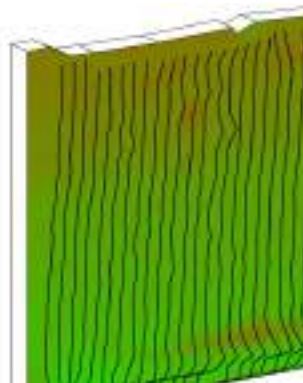
| Surface analysis results | simple profile lines | closed profile lines |
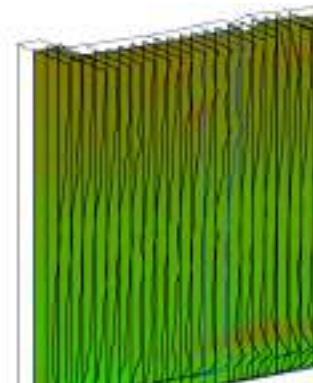
  When dealing with planar surfaces, the profile lines can also all be selected together. To do this hold down the [Tab] key during the selection.

- **Save named selection set per face**

  When this option has been selected, all of the profile lines of a surface will be saved as a named selection set. These can afterwards be called from the Revit tab Manage → panel Selection → Load [icon] to be able to select all of the profile lines at once.

- **Interpolate empty cells**

When computing the surface analysis and there are cells that contain no point cloud points, then with this option a value can still be generated for the profile lines. The missing value is then interpolated from the neighbouring values.

By clicking the [OK] button you will start the export using all of the selected options. A status message is displayed, that shows the progress of the output:
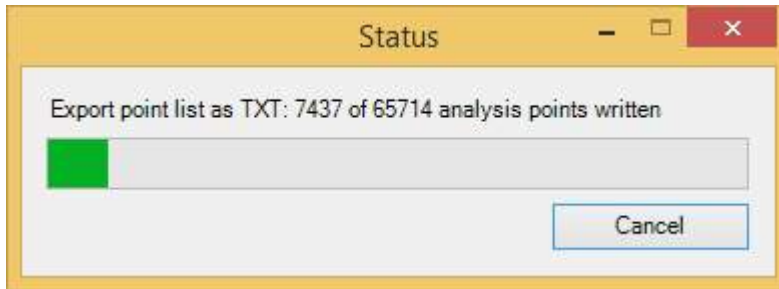
Surface Analysis Tools - Restore

Used in:

Project

The results of the command → **Surface Analysis Tools - Calculate** have been placed by As-Built for Revit in the Revit file. The visualisation of the analysis results by Revit itself can, however, not be saved in the project.

With this command you can with As-Built for Revit redisplay the saved results in your view, without the need of having to once again carry out the computation

Surface Analysis Tools - Hide

Used in:

Project

If you want to deactivate the analysis results of the command → **Surface Analysis Tools - Calculate** for specific surfaces, without removing the calculated values from the project, then use the command **"Hide"**. The following prompt appears in the status bar:

```
Please select multiple surfaces.
```
Select one or more surfaces with surface analysis results.

After clicking the **[Finish]** button the results will be hidden in the current view. Should the results need to be displayed again, use the command → **Surface Analysis Tools - Transfer**.

Surface Analysis Tools - Transfer

Used in:

Project

The results of the command → **Surface Analysis Tools - Calculate** are always displayed only in the view in which the computation was carried out. With the command **"Transfer"** you can transfer the visualisation of analysis results that have already been computed into another view. To achieve this, open the desired view and run the command. This prompt appears in the status line:

```
Please select multiple surfaces.
```
Select one or more surfaces with surface analysis results.

Only surfaces for which analysis results have already been computed can be selected. After clicking the **[Finish]** button the results will be displayed in the current view.

Surface Analysis Tools - Place USIBD Legend

Used in:

Project

This function inserts a legend for the display style "USIBD LOA Range" in the current plan view. The gradation that is displayed represents the Standard for the "Level of Accuracy (LOA)" in accordance with the definition of the USIBD (http://usibd.com).
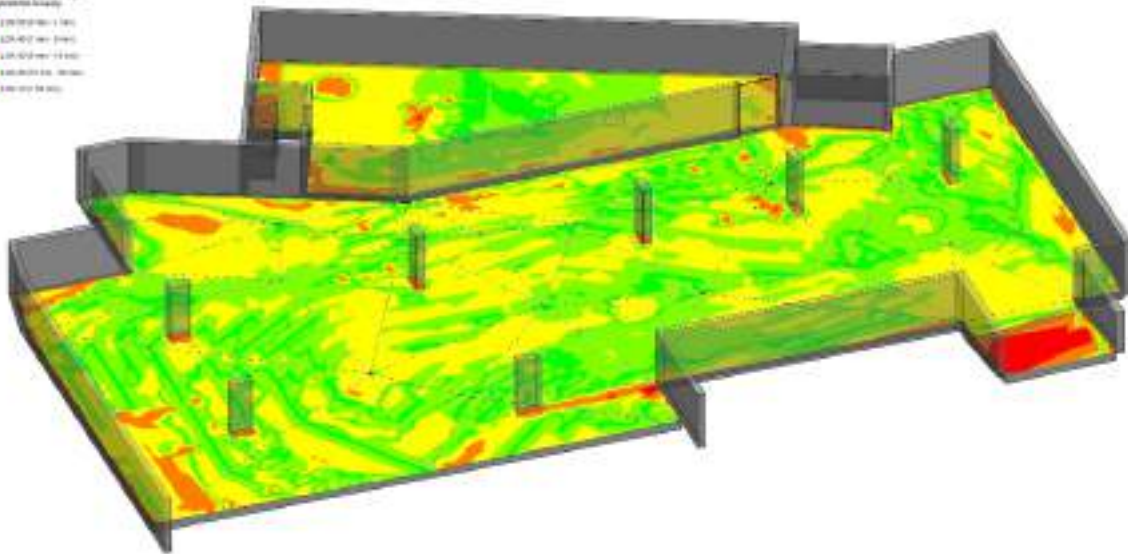
Point Cloud - Coordinates - Linking PC Coordinates

Used in:

Project

The command "Linking PC Coordinates" allows you to link a point cloud that is loaded in Revit with a VirtuSurv project. This is meaningful, if you want to work in parallel with VirtuSurv and the point cloud in Revit. Quite often when inserting point clouds in Revit, they are not in the middle of the project or they are not orthogonally aligned. With the command "Linking PC Coordinates" you can rotate and move the point cloud in Revit as you want, and at the same time continue working in the same coordinate system in the VirtuSurv project. To do so it is necessary that the point cloud is loaded in Revit and the VirtuSurv project has been referenced in the same superior coordinate system.

Following are the steps for linking a point cloud in Revit with a VirtuSurv project:

■ Insert a point cloud into your project using the option "Auto - Center to Center" (Insert tab)
→ Link panel → Point Cloud button)

■ Start the command "Linking PC Coordinates". The following prompt appears in the Revit status line:

| Select a Point Cloud | *Click on the point cloud* |

Now the point cloud is linked with the VirtuSurv project. You can now freely rotate and move them. The transformation of the linked point cloud is considered for every command started in VirtuSurv.

To remove a link, once again call the command "Linking PC Coordinates" and confirm the following dialogue box:
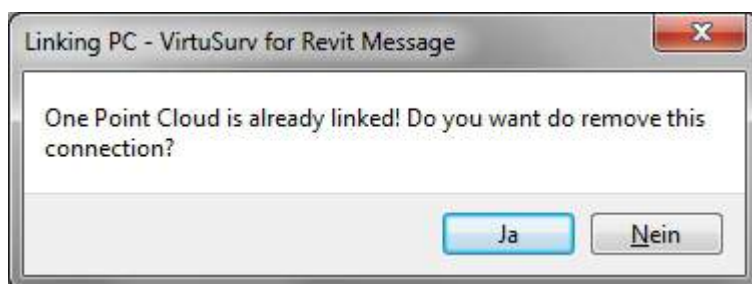
Now you may link to another point cloud.

Note: If you have previously set the project coordinates in VirtuSurv (command "Project coordinates") with the option "use shared coordinates", then you can insert the point cloud in Revit with the option "Auto - from shared coordinates". The position of the inserted point

cloud is exactly coincident with the project coordinates set in VirtuSurv. The link with the point cloud is not absolutely necessary.

Point Cloud - Coordinates - Acquire PC Coordinates

Used in:

Project

With this command you can retrieve the internal point cloud coordinate system, and align the shared coordinate system on to it.

Proceed as follows:

1. Insert a point cloud into your project using the option "Auto - Center to Center" (Insert tab) → Link panel → Point Cloud button 🖼️)

2. Align the point cloud in the project by moving and rotating it, until it meets your requirements

3. Start the command "Acquire PC Coordinates". The following prompt appears in the Revit status line:

Select a Point Cloud   *Click on the point cloud*

As-Built for Revit has now moved the survey point to the origin of the internal point cloud coordinate system and the rotated the geographical north relative to the Y-axis of the point cloud coordinate system. The shared coordinate system therefore corresponds to the internal point cloud coordinate system.

If you now want to insert other external data (e.g. a site plan as CAD drawing), that have the same coordinate system as the point cloud, then select the Positioning option "Auto - from shared coordinates". The external file is then aligned to the point cloud and inserted into the project.

Point Cloud - Coordinates - Import Scan Positions

Used in:

Project

With the command "Import Scan Positions" you can insert as symbols into the Revit project, the individual scan positions of a VirtuSurv project. For this it is first necessary to export the VirtuSurv scanner locations using the context menu of the VirtuSurv project, in the Project Explorer in to a TXT-file.

The Revit command will insert a pre-defined family symbol at the location of each single scan. Via the symbol parameters you may access the coordinates of the scan positions as well the information of the VirtuSurv project and scan file:
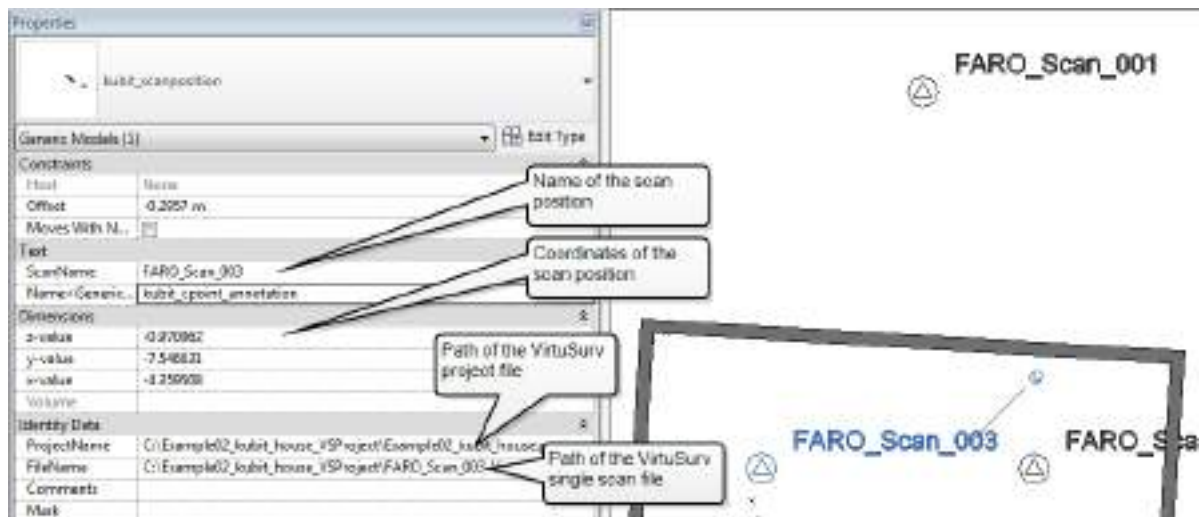
From the file path you can start VirtuSurv directly from Revit. The appropriate VirtuSurv project will be automatically loaded and the selected scan view opened. Just simply click on the file path and then on the "Open Hyperlink" symbol that then appears.

Point Cloud - Coordinates - Import scan positions from RCP

Used in:

Project

With this command you can insert, as symbols into the Revit project, the individual scanner positions from all of loaded RCP point clouds. The prerequisite is that RCP file also contains Individual scans.

The Revit command will insert a pre-defined family symbol at the location of each single scan. You have access to the scan position coordinates from the symbol parameters.

Point Cloud - Export PC Region

Used in:

Project

The command "Export PC Region" exports a region of a Revit point cloud into a PTS file, so that it can be subsequently inserted in the family editor (→ **Connect Point Cloud - Insert PC Region**). This command only works in an orthogonal 3D view. Next activate the section box of the view and clip the point cloud to create the region that you want. After issuing the command the visible region of the point cloud will be exported.

Point Cloud- Auto Measure Parameter

Used in:

Project

The command serves to automatically determine the point cloud parameters **"Noise"** and **"average distance between points"**. These parameters can be manually modified in the → **As-Built - Settings**.

Both parameters are essential for object recognition in As-Built for Revit. By individually setting both parameters it is now possible for the first time to process point clouds of varying qualities from different models of scanner and manufacturers.

Parameter "Noise"
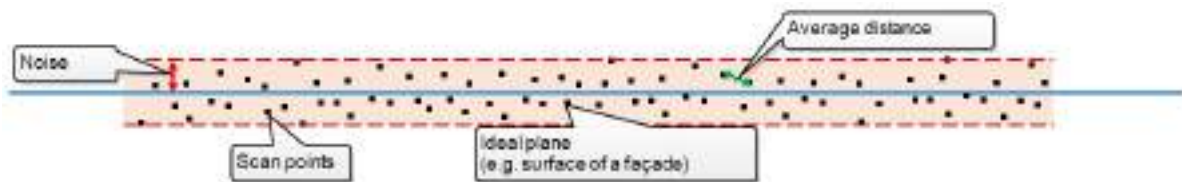
This parameter specifies by how much the observed points in the point cloud deviate from the ideal surface. With regard to a normal distribution it is approximately twice the standard deviation. The majority of scanner manufacturers give a maximum point deviation as the quality of their model of scanner.

In case you want to manually set the **"Noise"** parameter, first refer to the documentation of your scanner. Pay particular attention that the **"Noise"** parameter in As-Built for Revit includes as well as the point deviation from the scan also the surface properties. For example the "noise" of a rough rendered façade will vary from several millimetres to centimetres from the ideal plane of the façade. An additional influence is also the quality of the scan registration of the noise of the point cloud.

Parameter "average distance between points"

This parameter specifies by how far on average a point lies, from other related points (e.g. on a façade surface), to its next adjacent point. This parameter varies significantly within a point cloud. This is dependent upon: from how many locations an object has been scanned; how far is the object from the scanner location; how high is the scan density; and what is the angle of incidence that the object surfaces have been scanned.

Automatically measure

As has already been described, both parameters can vary significantly depending upon the point cloud and even the point cloud region. It is therefore important, that these parameters can

be determined within the point cloud.

In order to do so start the command. A prompt appears requesting that three points be picked in the point cloud:

```Pick the first point for the definition of a plane for the command automatic measure```	*Pick the first point within a closed surface in the point cloud*
```Pick the second point for the definition of a plane for the command automatic measure```	*Pick the second point within a closed surface in the point cloud*
```Pick the third point for the definition of a plane for the command automatic measure```	*Pick the third point within a closed surface in the point cloud*

All three points must define a plane. For example the picked points on a façade surface could look like this:

The picked triangle should, where possible, be closed, that means there should not be any openings (e.g. windows) contained within it. Furthermore there should not be any other objects that lie before or behind the picked plane (e.g. window reveals or a wall behind the plane).

After picking the three points the following dialogue box is displayed:
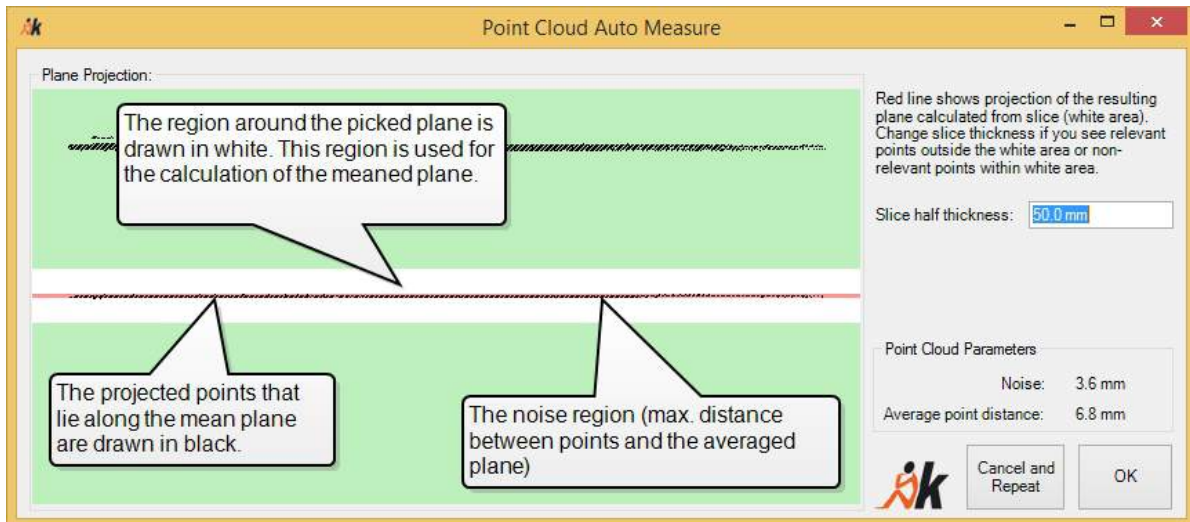
A plane is defined from the picked points, it is viewed from the side and displayed in the pane (black points) **"Plane Projection"**. The parameter **"Slice half thickness"** defines a region (shown white in the preview pane) around the points, in which a mean plane will be calculated.

Based on this mean plane and the points used, the noise (maximum point distance to the plane) and the average point distance to the surface will be calculated. The noise (maximum point distance) is shown in the preview as a red region around the meaned plane.

For the correct determination of the parameters there should always be a thin strip of point cloud points to be seen, that is surrounded by a thicker red strip, that in turn should have a white border.

If the slice thickness is set too big, it can happen, that points which lie behind the picked plane will also be used in determining the plane:
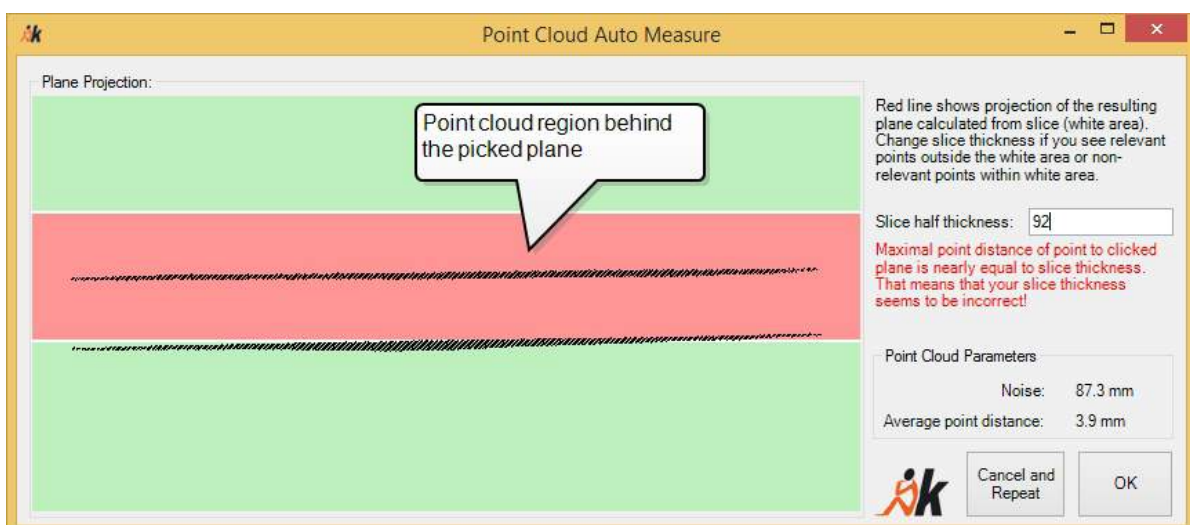
The slice thickness should be set here as small as possible, so that there is only a single plane that lies in the white region, e.g. 50 mm.
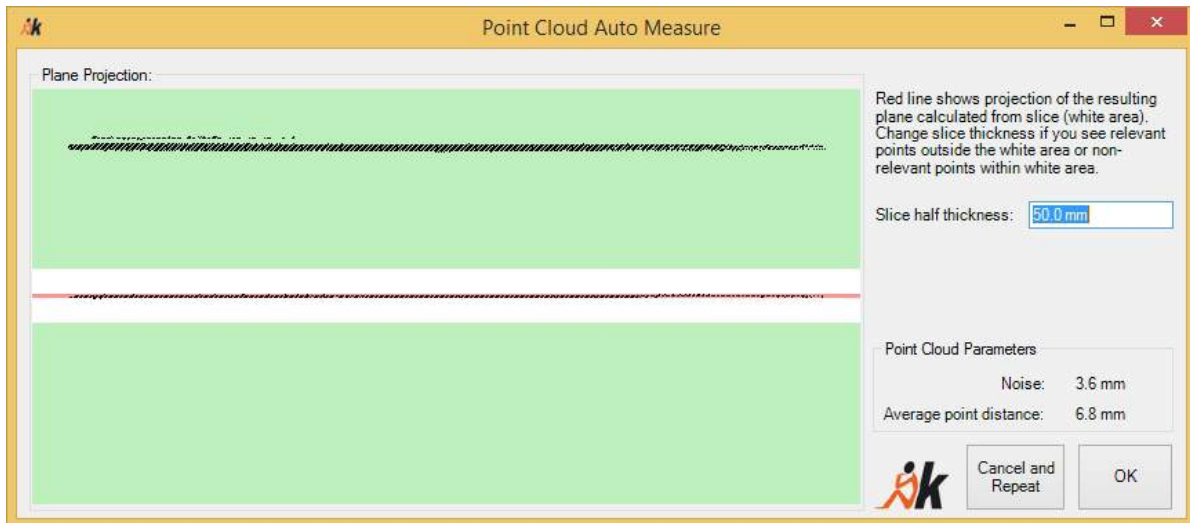
Should the slice thickness be set too small, then the noise cannot be determined correctly anymore, because the region used for determining the plane and the noise are almost the same. In this case a warning will be displayed, that a bigger (seldom also a smaller) slice thickness must be used:
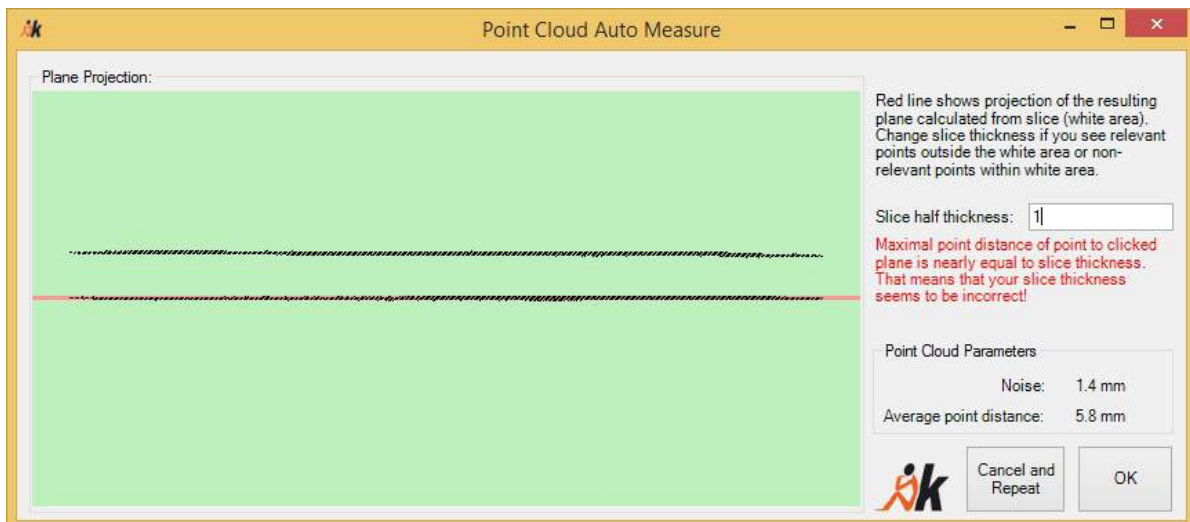
The calculated values are shown in the section "Point Cloud Parameters". By clicking [OK] you confirm the values, so that they can be set in the As-Built for Revit settings.

Connect Point Cloud - Insert Ortho Image

Used in:

Family

With this command it is possible to insert ortho images into the family editor with their correct scaling, in views, floor plans and sections.

The images must have previously been created and saved in the Revit project using the command → **Object Tools - Ortho Image**. If you click on the button for the command, an "Open File" dialogue box opens asking for the relevant *.xmlscale file. This *.xmlscale file is always created by As-Built for Revit at the same time as the ortho image, and contains as well as the precise scaling the path to the saved image file. If you ever want to archive or share ortho images, please remember to copy both files.
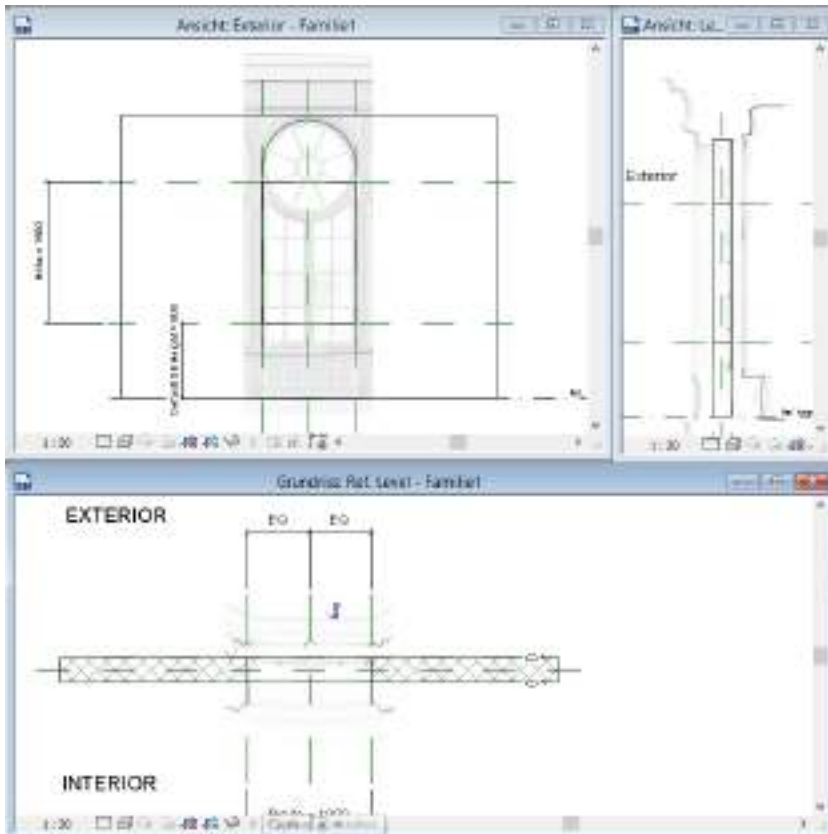
After the correct *.xmlscale file has been selected, then the referenced image with the correct scaling will be inserted in the centre of the families editor. Afterwards you can move the images to the required position.

In the figure on the left you can see three X-ray ortho images in two elevations and in the floor plan of the families editor.

Connect Point Cloud - Insert PC Region

Used in:

Family

With this command you can insert point cloud regions from PTS data into a Revit family document. The following dialogue box appears:
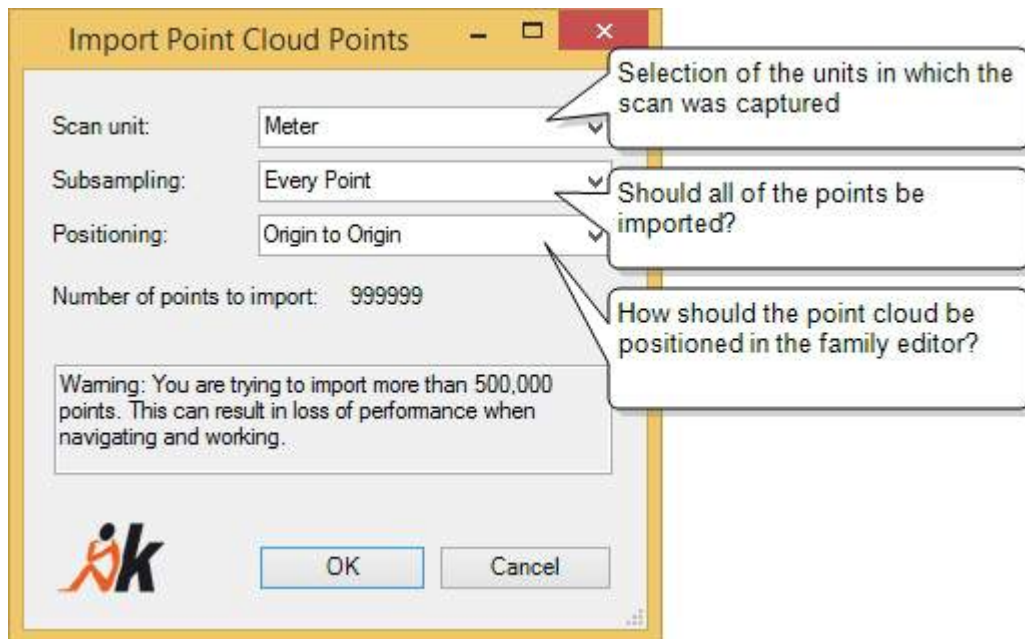
On grounds of performance it is recommended to limit point cloud regions to a maximum of 500,000 points. If necessary you can reduce the number of points to be imported in the option "Sub sampling". To position the point cloud regions you have the following options to select from:

- **Origin to Origin:** The global origin of the point cloud regions will be placed on the origin of the family file.

- **Centre to Origin:** The centre of the point cloud regions will be placed on the origin of the family file.

- **By VirtuSurv Project Coordinates:** If you have previously set the project coordinates in VirtuSurv (→ **VirtuSurv - Set Project Coordinates**), the point cloud region will be place relative to these coordinates.

VirtuSurv Tools - Model Line
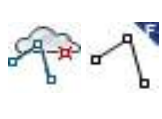
Used in:

Project   Family   VirtuSurv

With this command you can create model lines in the REVIT project.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for the model lines can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

If you want to create model lines in the families editor, only the scan view of VirtuSurv is available to you.

After starting the command the following option window opens up:
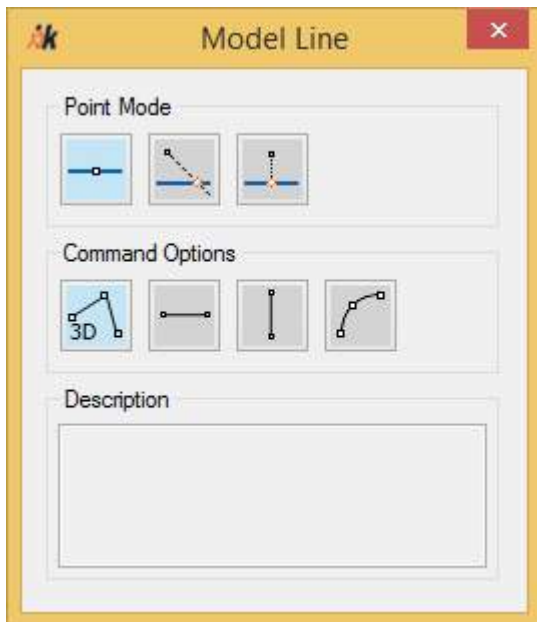
The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To create the model lines you have the following options to select from:

Icon	Command	Operation
3D	Continuous 3D model lines	Click on a number of points in the scan to create model lines in 3D space. If you use this option with point mode set to "Standard" then every line section will create a Revit work plane perpendicular to the xy plane.
⊶	Horizontal model line	Click on two points in the scan to create a horizontal model line. The line segment will be created at the level of the averaged Z-coordinate of both points on a work plane parallel to the XY plane.

Icon	Command	Operation
	Vertical model line	Click on two points in the scan to create a vertical model line. The line segment will be created on a work plane that is perpendicular to the xy plane. The x and y coordinates of both points will be averaged.
	3D arc	Click on three points in the scan to create an arc in 3D space.

VirtuSurv Tools - Wall

Used in:

Project   VirtuSurv

With this command you can place walls by clicking on only 2 points on one wall face in the Revit project.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for placing the wall can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:

In the frame "Levels" you define the lower upper constraints of the component. For more information on this, see Section **4.2.2** .

All of the currently loaded family types are available in the Type drop down selection list. The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the wall you have the following options to select from:

Icon	Command	Operation
	Straight wall	Click on two points in the scan to create a wall.
	Curved wall	Click on three points in the scan to create a curved wall.
	Continuous wall	Click on an unlimited number of points in the scan to create continuous walls. Click respectively on two points per wall segment. The individual walls will be automatically joined to each other.

VirtuSurv Tools - Wall Thickness

Used in:

Project   VirtuSurv

With this command already existing walls can have their wall thicknesses adapted by picking points on the second wall face. For this purpose there must be an existing wall in Revit within the preset search distance (see → **As-Built - Settings**).

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for placing the wall can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:

The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To adapt the wall thickness you have the following options to select from:

Icon	Command	Operation
	Adjusting a straight wall	Click on two points in the scan on the second side of the wall to customise the wall thickness.
	Adjusting a curved wall	Click on one point in the scan on the second side of the wall to adjust the wall thickness.

To adapt the wall face As-Built for Revit creates a new wall type with the measured wall thickness.

In Revit the wall width is always defined by the structure of the selected wall type. In order to adapt the width of an existing wall using the command "VS Wall Face", As-Built for Revit first makes a copy of the wall type of the existing wall, adapts the width of the load-bearing layer according to the measurement and applies this new wall type to the existing wall. The new wall type receives the wall thickness as name suffix (e.g. from the wall type "Generic - 200 mm" the new type: "Generic - 200 mm_365,0 mm"). Of course you may change the name in Revit afterwards at any time.

VirtuSurv Tools - Level

	Used in:	
	Project	VirtuSurv

With this command you can create a new level by picking a point in Revit.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The point that is required for the level can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

Simultaneously with the level a corresponding plan view will be created. The designation will be carried in accordance with the settings in → **As-Built - Settings**

VirtuSurv Tools - Window

	Used in:	
	Project	VirtuSurv

With this command you can place windows in the Revit project. For this purpose there must be an existing wall in Revit within the preset search distance (see → **As-Built - Settings**).

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for placing the wall can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:
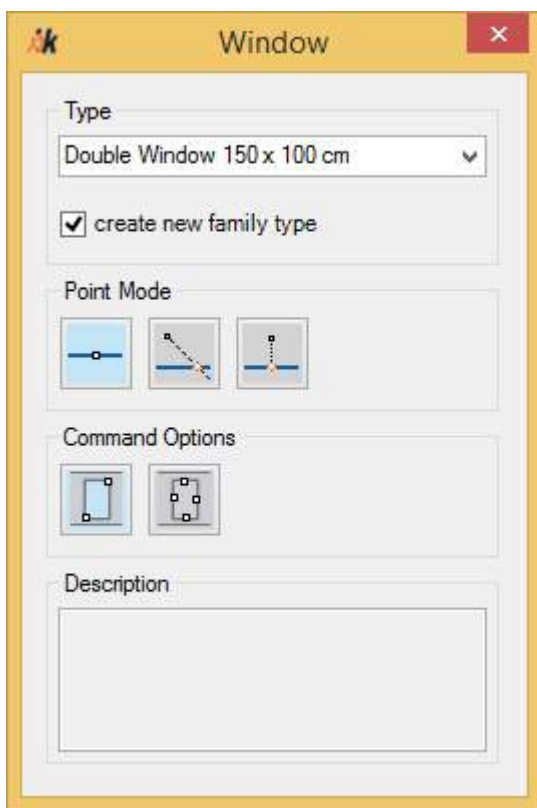
All of the currently loaded window families and their types are available in the Type drop down selection list. The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the wall you have the following options to select from:

Icon	Command	Operation
	Using diag-onals	Click on two diagonal points in the scan to create a window in an existing wall.

Icon	Command	Operation
	Using reveals	To create a window click on four points in the reveals or on the edges of the window opening.

If the option **"create new family type"** is activated, then the selected door type will be duplicated and its height and width dimensions will be adjusted.

If this option is not activated then the height and width dimensions will only be adjusted if they are defined as instance parameters.

VirtuSurv Tools - Door

Used in:

Project   VirtuSurv

With this command you can place doors in the Revit project. For this purpose there must be an existing wall in Revit within the preset search distance (see → **As-Built - Settings**).

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for placing the door can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:
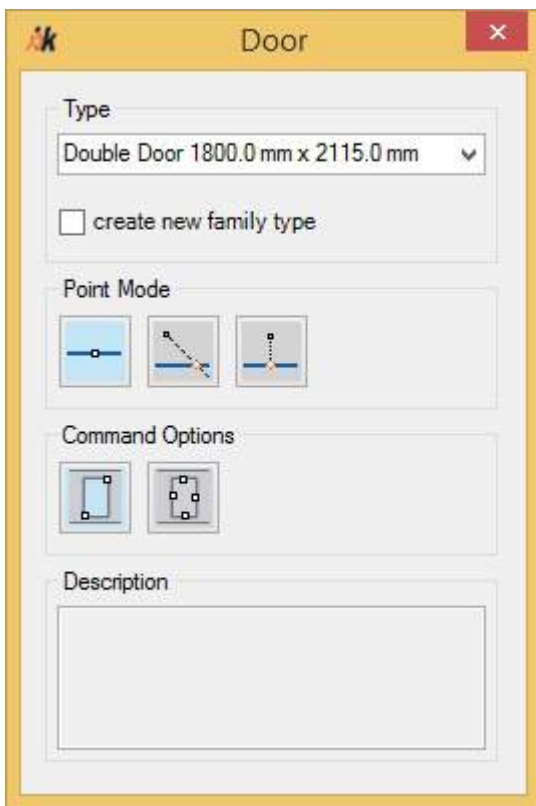
All of the currently loaded door families and their types are available in the Type drop down selection list. The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the door you have the following options to select from:

Icon	Command	Operation
	Using diag-onals	Pick two diagonal points in the scan to create a door in an existing wall.

Icon	Command	Operation
	Using reveals	To create a door pick four points in the reveals or on the edges of the door opening.

If the option **"create new family type"** is activated, then the selected door type will be duplicated and its height and width dimensions will be adjusted.

If this option is not activated then the height and width dimensions will only be adjusted if they are defined as instance parameters.

VirtuSurv Tools - Opening
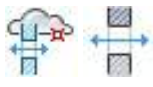
	Used in:	
	Project	VirtuSurv

With this command you can place openings in the Revit project. For this purpose there must be an existing wall in Revit within the preset search distance (see → **As-Built - Settings**).

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for placing the opening can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:
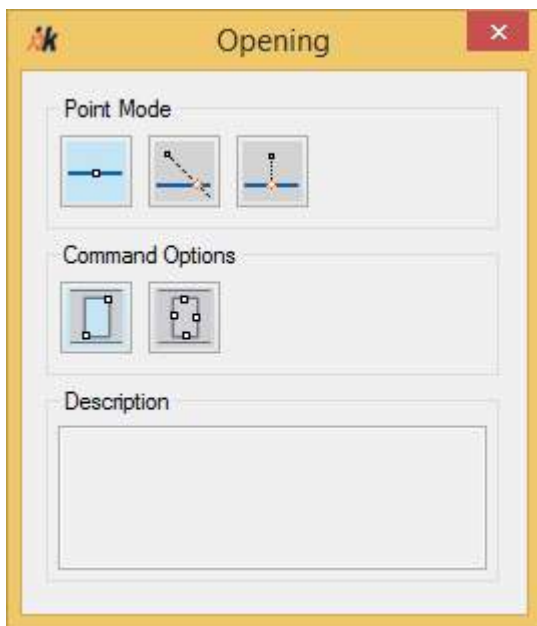
The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the opening you have the following options to select from:

Icon	Command	Operation
	Using diagonals	Click on two diagonal points in the scan to create an opening in an existing wall.
	Using reveals	To create an opening pick four points in the reveals or on the edges of the opening.

VirtuSurv Tools - Beam

	Used in:	
	Project	VirtuSurv

With this command you can place line based structural families (e.g. braces or drop down beams) in the Revit project, by clicking on 4 points in the scan or in the point cloud.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for the placement can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:

All of the currently loaded family types are available in the Type drop down selection list. The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the components you have the following options to select from:

Icon	Command	Operation
	Place in 3D	Click on four points in the scan to create an exemplar of the selected family type. The first two points define the orientation of the family on the front face (vertical aligned face). Points 3 and 4 define the orientation of the family on the upper or lower face.

Icon	Command	Operation
	Place hori- zontally	Analogue to placing in 3D, however the z heights of points 3 and 4 are averaged, so that the family exemplar can be aligned horizontally.

VirtuSurv Tools - Column

Used in:

Project   VirtuSurv

With this command you can place 2 level based family exemplars (e.g. columns) by picking from up to 3 points in the scan, or in the point cloud in the Revit project.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The points that are required for the placement can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:

In the frame "Levels" you define the lower upper constraints of the component. For more information on this, see Section **4.2.2** .

All of the currently loaded family types are available in the Type drop down selection list. The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

To place the components you have the following options to select from:

Icon	Command	Operation
	Using centre point	Click on a single point, as the base point, in the scan to create an exemplar of the selected family type.
	Using a rectangle	Click on three points in the scan to create an exemplar of the selected family type. In order to place the object, VirtuSurv determines the maximum extents of the family's geometry in the floor plan level (x-y level) to create a bounding rectangle. The first two points determine the orientation of the long side of the rectangle and the third point the position of the short side. The centre point of the rectangle forms the base point of the family.
	Using a circle	Click on three points in the scan to create an exemplar of the selected family type. The centre point of the circle drawn through the three points forms the base point of the family.

VirtuSurv Tools - CPoint

Used in:

Project   Family   VirtuSurv

With this command you can create a construction point by picking a point in Revit. You can use the command both in the project environment, as well as in the families editor.

The command can be called from the As-Built tab on the Revit ribbon, as well as from the Revit tab on the VirtuSurv ribbon. The point that is required can be picked in both the Revit point cloud and also in the VirtuSurv scan view.

After starting the command the following option window opens up:
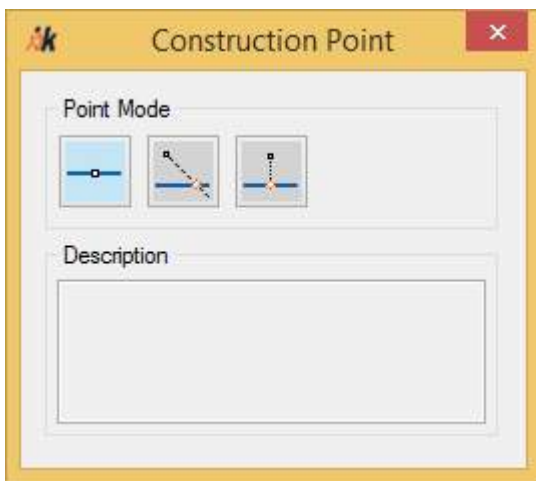
The Point Modes control how the picked coordinates are to be used. For more information on using the point modes, see Section **4.2.3** .

The designation will be carried in accordance with the settings in **→ As-Built - Settings**

VirtuSurv - Set Project Coordinates

	Used in:
	VirtuSurv Family

With this command you can specify the project coordinates, which you want to work with in your Revit project. This applies to both the project environment and the family editor. After starting the command the following option window opens up:
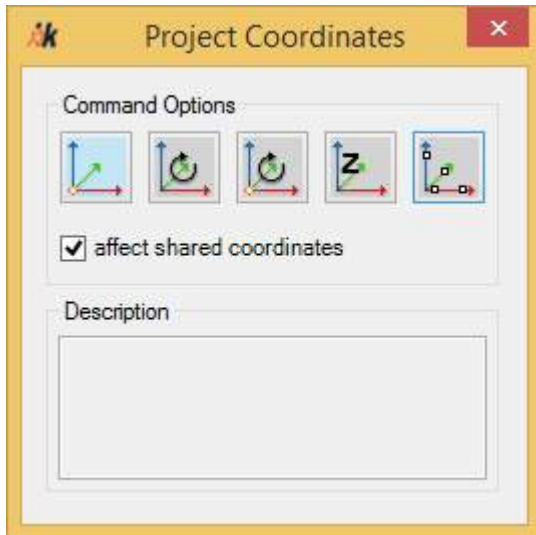
The various options allow the project origin and the project alignment to be defined.

Icon	Command	Operation
	Project origin	Click a point in scan: defines the origin of the project coordinate system in Revit.
	Project orientation	Click on two points in the scan to define the direction of the x-axis (east-west) of the Revit project coordinate system.
	Project origin and alignment	Click on two points in the scan to define the direction of the x-axis (east-west) of the Revit project coordinate system. The project origin is defined as the midpoint between the two points.
	Project z-height	Click on a point in the scan to define the z-height of the Revit project coordinate system.
	Set the project coordinates	Click on two points in the scan to define the direction of the x-axis (east-west) of the Revit project coordinate system. Click on a third point to fix the position of the y-axis (north-south) and click on a fourth point to define the z-height.

If the option **"affect shared coordinates"** is also selected, the VirtuSurv project coordinates will also be transferred into the Revit defined "Shared coordinate system". This is meaningful when the VirtuSurv project is for example in a superior reference coordinate system and you wish to also work in Revit in this coordinate system with other data (e.g. point clouds or layout plans). The "Shared coordinate system" ensures that all data in this reference coordinate system will be brought into Revit in the correct position. In the Revit Family editor this option has no meaning.

VirtuSurv - Work Plane

Used in:	
VirtuSurv	Family

With this command you can create the active work plane by picking three points in the VirtuSurv scan view.

This command is also available in the Family editor.

VirtuSurv - Detail line

Used in:

VirtuSurv

With this command you can create detail lines in Revit.

Click on a number of points in the VirtuSurv scan view to create detail lines in the active Revit view. A planar view must be active in Revit to be able to use this command.

View - Hide or Show PC

Used in:

Project

This command toggles the visibility of all point clouds in the current view on or off. Thereby the visibility of the category "Point Clouds" is controlled. Should individual point cloud objects be manually hidden in Revit, they will not be affected by this command.

View - Hide / Show PP

Used in:

Project

With the commands "Show PP" and "Hide PP" you can show and hide all protocol points in the active view. Protocol points are 3D point symbols that are inserted into the Revit project, on the position of the clicked point in the scan, every time each VirtuSurv command is called. These points are simply meant for monitoring purposes.

Note: If you have hidden the protocol points in a view and then call further VirtuSurv commands, the new protocol points will still be visible. To hide these points also, you must run the "Hide PP" command again.

As-Built - Getting Started

	Used in:
	Project Family

This command opens a dialogue box, from which you have access to tutorials, videos and further information relating to As-Built for Revit:
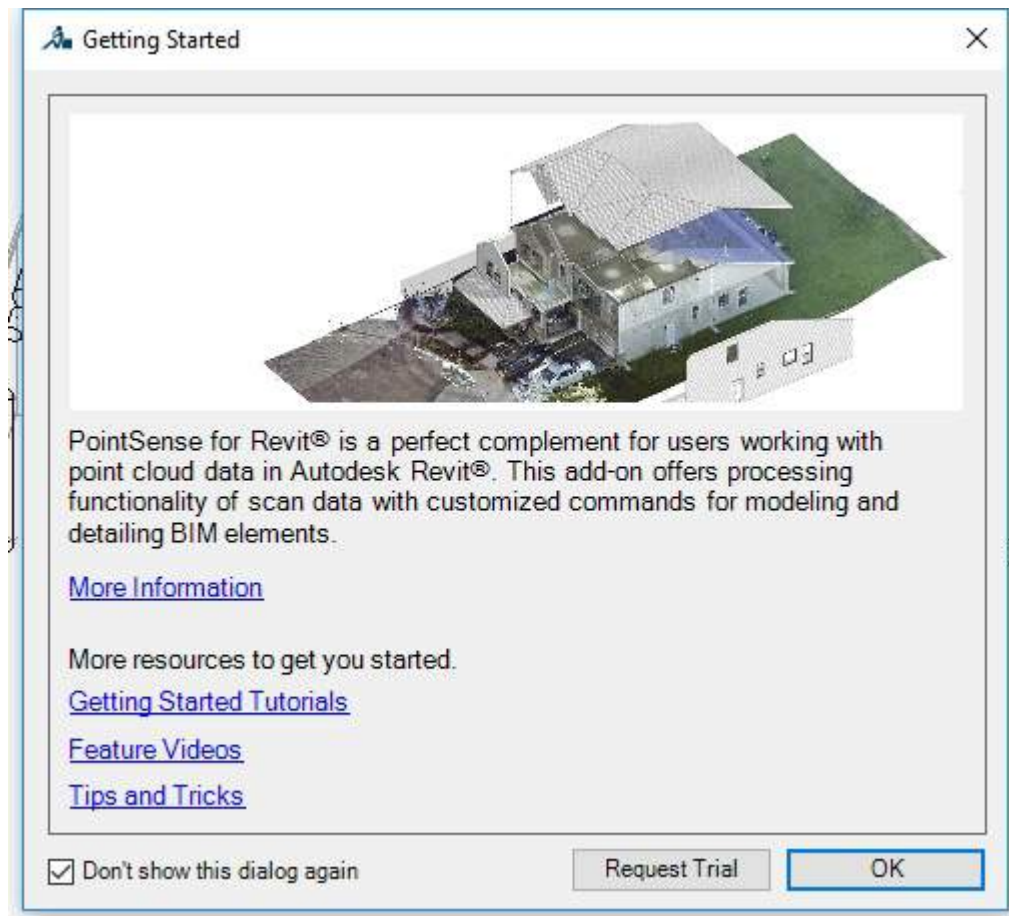
You can request a trial version of As-Built for Revit clicking the button **[Request a trial]**. The button only appears when no licence has been found.

The "Getting Started" dialogue box appears by default when starting Revit. By selecting the option "Do not show this dialogue again" you can switch off this behaviour for the currently installed program version.

As-Built - Cleanup

Used in:

Project   Family

This command is used together with its various options to clean up a As-Built for Revit project. This can be helpful when project data are to be passed on without As-Built for Revit content.
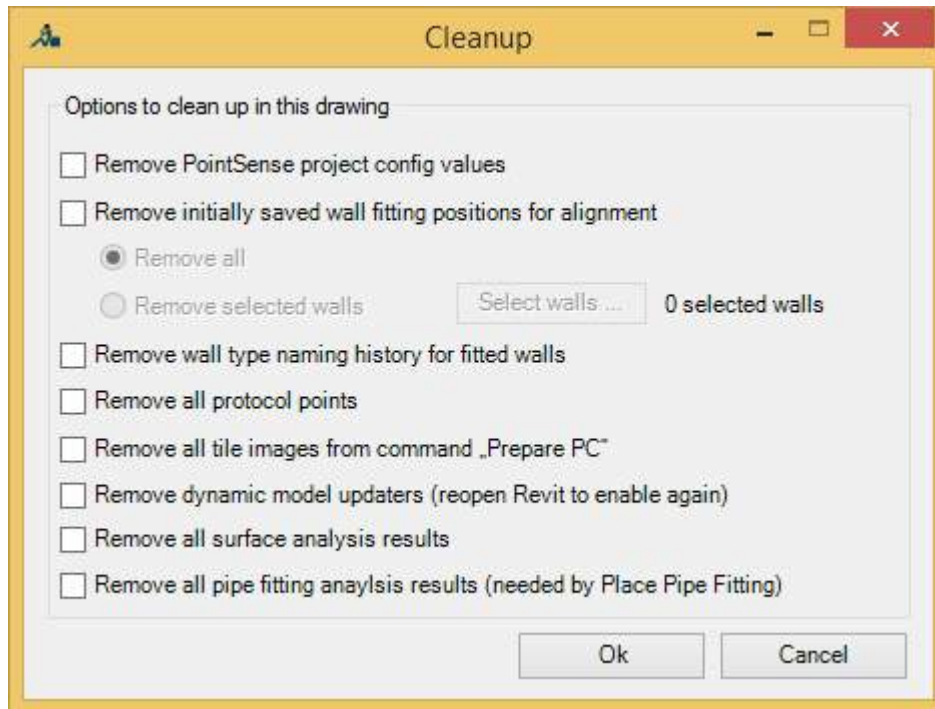
- "Remove As-Built project config values"

  Removes all configuration values that have been set from the project. That is, all values will be reset to their values in the default settings the next time the command is called.

- "Remove initially saved wall fitting positions for alignment"

  Each time before the command → **Alignment - Align Walls** is called, the original position of the selected walls will be saved, in order that the wall deviation calculation, and if necessary the walls can be reset to their position from → **Object Tools - Fit Wall**. It may be necessary to reset these values, if the wall position is to be manually changed and then the command "Align walls" is to be called. It is possible, using this option, to reset all of the walls simultaneously, or only a few selected walls, (using the button **[Select walls ...]**).

■ "Remove wall type naming history for fitted walls"

New Revit wall types will sometimes be created when calling the command → **Object Tools - Fit Wall**. These new wall types are saved under the names of the original wall type. Thereby, further derived wall types can always be named in the style of the original. This naming history can be removed from the project with this option.

■ "Remove all protocol points"

Deletes all protocol points in the project.

■ "Remove all tile images from command "Prepare PC""

Deletes all image tiles that have been created with the command → **Object Tools - Prepare Point Cloud**.

■ "Remove dynamic model updaters (reopen Revit to enable again)"

Removes all "Dynamic Model Updaters" from the project. These fulfil functions, such as switching the level of detail for pre-rendered image tiles from the command "Prepare Point Cloud". Likewise, with this technique in point cloud modifications, the pre-rendered images will be declared as invalid and deleted. The removal of these "Dynamic Model Updaters" can be important for sharing the project, it restricts however the usual method of working in As-Built for Revit. Therefore, when selecting this option this warning appears:
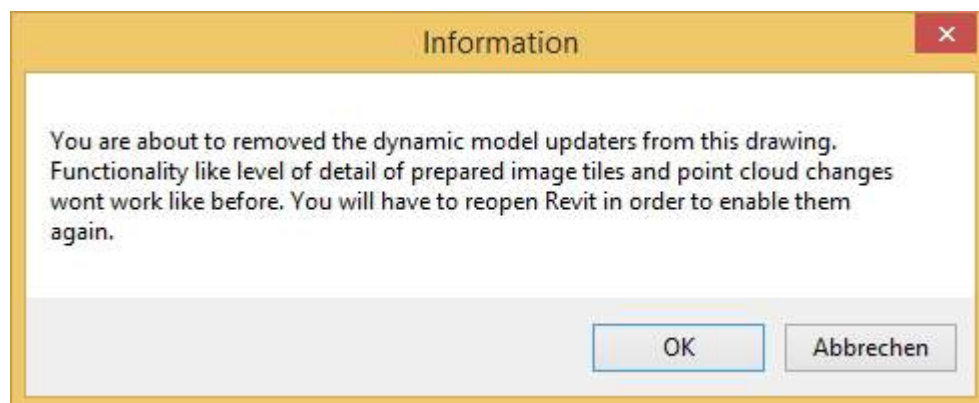
A new start of Revit is required to restore the usual method of working with "Dynamic Model Updaters".

■ "Remove all surface analysis results"

Removes all calculations of the command surface analysis, even when they are not currently displayed in the model.

■ "Remove all of the results from the pipe fitting analysis (from "Pipe fittings placement" required)"

Removes all of the results of the Families analysis, which is carried out on the first start of the "Placement of Pipe Fittings".

After you have selected the desired options in the "Clean Up" dialogue box, you click on **[OK]** button and the corresponding actions are carried out. Finally you will receive a summarised status message of the "Clean-up" actions that have been carried out:
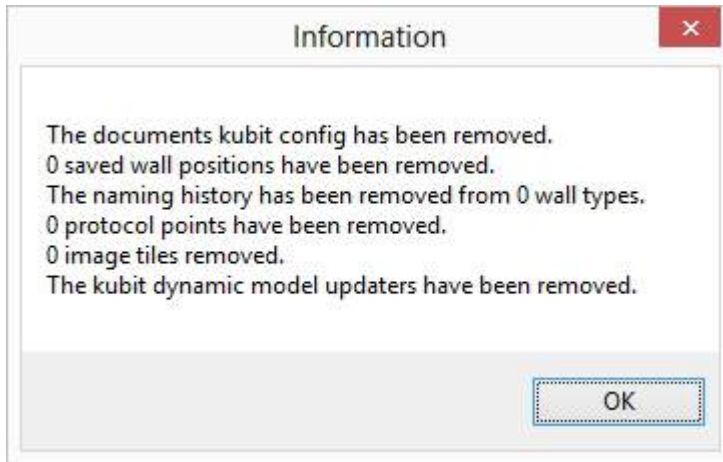
As-Built - Settings

	Used in:
	Project Family

This command opens the As-Built for Revit Settings dialogue box in which you can set project specific settings.

PointSense for Revit Settings   ✕

∨ Align Pipes	
Angular tolerance when connecting pipes	0.01°
Use strict constraints if the tolerance is not met or cancelled by user	Yes
∨ Cylinder Detection	
Cylinder search radius	50.0 mm
Radius tolerance in percent	15.00
∨ General	
Construction point number	1
Construction point prefix name	CPoint_
Inserted image size threshold value in MByte	256.00
Is getting started splash screen disabled	Yes
Is software auto update disabled	No
Language	English
Level number	1
Level prefix name	Floor_
Turn off inaccurate line/wall failure	Yes
Use point cloud section tools	Yes
∨ Object Creation	
Element search distance	500.0 mm
Rounding step for opening dimensions	5.0 mm
∨ Ortho Image	
Intensity factor for ClearView	1.00
∨ Plane Detection	
Contour smoothing deviation	40.0 mm
Plane search radius	1500.0 mm
∨ Point Cloud Parameter	
Average point distance	10.0 mm
Noise	10.0 mm
∨ Project Transformation	
Linked point cloud ID	
Project orientation	0.00°
Project origin X	0.0 mm
Project origin Y	0.0 mm
Project origin Z	0.0 mm
∨ Surface Analysis	
Maximum Surface Distance	300.0 mm
Mean Grid Size	50.0 mm
∨ Wall Creation	
Automatic wall extension	Yes
Max. wall thickness for fitting	1200.0 mm
Query if value of extension greater than	200.0 mm
Resolution for point cloud prepare	1.5 mm
Rounding step for wall thickness	5.0 mm
Wall search distance	250.0 mm

OK   Cancel   Apply

Parameter	Description
General	
Turn off Inaccurate Line/Wall Failure	Revit sends a warning when wall axis or lines slightly differ from the orthogonal project coordinate system. This warning can be switched off here.
Automatisches Software Update deaktivieren	Diese Optione deaktiviert die Suche nach Software Updates im Internet.
Level Number	Start number for new levels. This value is increased automatically.
Level Prefix Name	Name prefix for new levels
Inserted image size Threshold value in MByte	This parameter defines the file size threshold when inserting new images with As-Built for Revit. That mainly affects the functions → **Object Tools - Prepare Point Cloud** and → **Object Tools - Ortho Image**, which place the rendered images in your Revit project. If the threshold value is exceeded, a warning message is displayed, that the images which are to be inserted should be scaled using a definable factor. The background is that when Revit has many and/or large images in its database it will become difficult to use or could even lead to a crash. This should thereby be avoided.
Construction point Number	Start number for new construction points. This value is increased automatically.
Construction Point Prefix Name	prefix name for new construction points.
Language	Language of As-Built for Revit menus and dialogues
Erste Schritte Dialog deaktivieren	Diese Option deaktivert die Anzeige des "Erste Schritte" Dialoges beim Starten von Revit.
Oberflächenanalyse	
Durchschnittliche Rasterweite	Definiert die durchschnittliche Größe einer Rasterzelle bei der Oberflächenanalyse.
Maximaler Oberflächenabstand	Definiert die maximale Entfernung von der Modeloberfläche bis zu welcher Punktwolkenpunkte bei der Oberflächenanalyse berücksichtigt werden.
Object Creation	

Parameter	Description
Element Search Distance	The search distance used when searching for existing walls when creating openings, doors and windows using VirtuSurv commands.
Rounding Step for Opening Dimensions	The value that the parameters height and width should be rounded to for new windows, doors and openings.

Ortho Image

Intensity Factor for Ortho Image X-Ray Visualisation	With this value you can control the intensity of an X-Ray ortho image. The higher this value is, the more contrasty and darker appears the X-ray image.

Plane Detection

Point Cloud Iteration Block Size	This value determines the size of the quadratic and cubic point cloud blocks that can be selected at any one time. This value mainly influences the performance of pattern recognition. Wherever possible, a block should have a size between 1 million and 10 million points. The default value is 4.0 metres and that is the optimal value for working in an architectural environment. Only change this value when working in another structural size, e.g. when working with terrain models or in the millimetre range.
Contour Smoothing Deviation	This value determines by how much the resulting contour from the commands "Plane fitting with contour" and "Fitting model lines" should be smoothed. The larger this value is the fewer vertices has the resulting contour.
Search Radius for Planes	This value determines the radius from the picked point within which the points will used to find and form the plane. You will get the best results when this value is approximately the size of your structure. For example, if you want to fit a plane on a building facade that is 10 x 10 m, then use a search radius of 5 m. However, should you want to fit planes on room walls of approximately 2.5 x 4 m, then use a search radius of around 2 m. The search radius can be slightly larger than your structure size, but the plane fitting can, in certain circumstances, take a little longer.

Point Cloud Parameter

Average Point Distance	This value defines the average distance between the point cloud points. The value can be automatically determined by using the command →

Parameter	Description
	Point Cloud- Auto Measure Parameter.
Noise	This value sets how far points can lie from an ideal plane. The value depends on the characteristics of your scan data (noise of your scanner) as well as the quality of the surface and the curvature of the scanned surface as opposed to an ideal plane. The value can be somewhat larger than the actual value. The accuracy of the results can, however, only be assured within the set value. For example, with a set noise value of 5 cm, an accuracy can only be guaranteed to 5 cm.
	You can achieve the best results, for example, if you use 3 mm for room walls, but for facades you should use 1-2 cm and 5 cm for roof surfaces.
	The value can be automatically determined by using the command → **Point Cloud- Auto Measure Parameter**.
Project Transformation	
Project orientation	The rotation angle of the Revit project coordinate system relative to the VirtuSurv project coordinate system.
Project Origin X	X coordinate (east) of the Revit project origin in the VirtuSurv project coordinate system.
Project Origin Y	Y coordinate (north) of the Revit project origin in the VirtuSurv project coordinate system.
Project Origin Z	Z coordinate (height) of the Revit project origin in the VirtuSurv project coordinate system.
Linked Point Cloud ID	Element ID of the linked point cloud. If no link is set the value is "-1"!
Wall Creation	
Resolution for Point Cloud Prepare	This value specifies the physical size of an image pixel, in project units, to be used in the "Prepare Point Cloud" command. It defines the resolution of the Prepare image and thus also the relevant required computation time.
Preparation of the point cloud and fitting the walls	This value defines the maximum possible wall thickness for wall fitting. Should the wall fitting ever fail for very thick walls you can increase this value to achieve better results.
Automatic Wall	When fitting the wall face with the appropriate VirtuSurv command and

Parameter	Description
Extension	using additional points to define a longer wall, then the wall will be automatically extended.
Query if the Value of Extension greater than	If the "Automatic Wall Extension" parameter is in-active you may define from which length difference the software prompts if the wall shall be extended or not.
Rounding Step for Wall Thickness	Value to which the new wall thickness shall be rounded.
Wall Search Distance	Search distance for existing walls when adjusting the wall side.
Align pipes	
Angular tolerance for pipe connections	The angular tolerance at the pipe connection which should at least be met when using the command "Align pipe run"
Use strict constraints if the tolerance has not been met or if cancelled by the user	If this option has been selected, a different algorithm with stricter constraints will be used when using the command "Align pipe runs", if: ■ the angular tolerance has not been met after 10000 iterations, or ■ the user has prematurely terminated the computation by pressing the [Esc] key.

Attention: Changes made to the values for "Project Transformation" have no effect on the internal Revit "Shared Coordinate system".

As-Built - Work-sharing Settings

Used in:

Project

The work -sharing settings are available for you to use when your Revit project has been activated for work sharing over a central model. Otherwise only a warning message is displayed. In a work-sharing project you now see the following dialogue box:
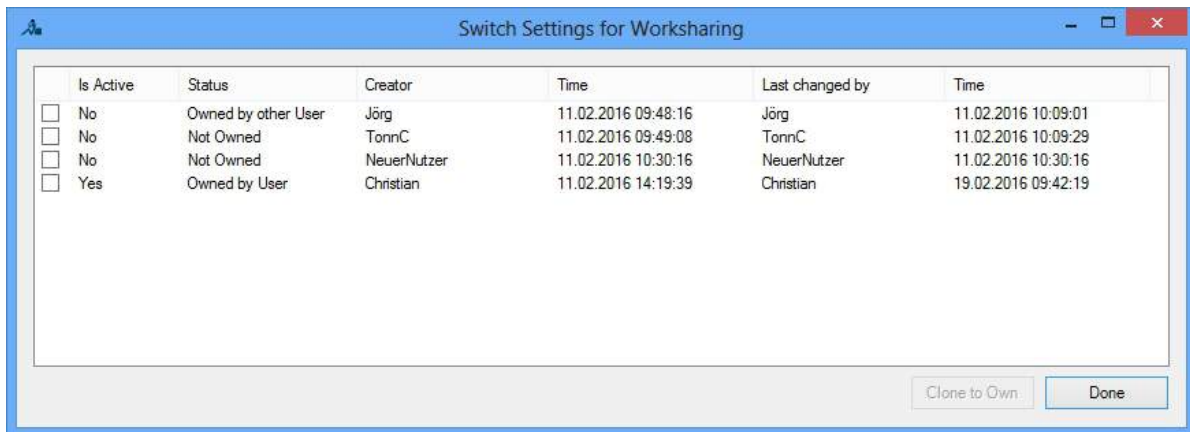
It will be shown which users have already have preference settings saved in the drawing(➔ **As-Built - Settings**). Each individual user works using their own personal preference settings. If you ever want to use the same settings as another user, place a tick in the foremost column of the user and click on the button "Copy". Now all of your own preference settings will be replaced by those of the other participating user. This can make sense when using team wide project standards.

As-Built - Licence Info

Used in:

Project

With this command you can check the license status of your software. You can also add, update and transfer licenses to other machines or you may collect the license information to a c2v file and update the license from a v2c file.
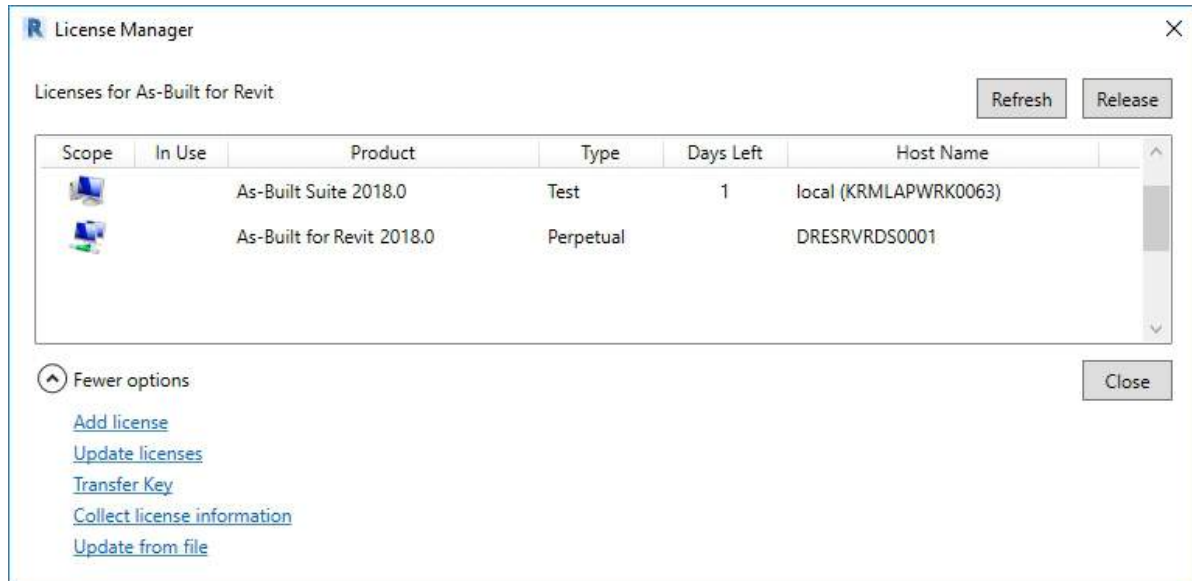
A valid license is necessary to allow the software to work, either for a trial period or for the terms defined in a purchased entitlement. A network license can be used by any computer on the network, while a local license is only locally visible. A license can be provided in a softlock or a hardlock (dongle).

The license manager shows all relevant product versions for the current application.

Each product has the following information:

■ **Scope** – Indicates from where the license is available: local, remote (network) or port-lock (dongle);

■ **In Use** – Indicates if the license is being used by the current application;

■ **Product** – Name of the product including version;

■ **Type** – Type of the license: Trial, Perpetual or Temporary;

■ **Days Left** – Number of days to the license expiration. Applies for trial and temporary types only;

■ **Host Name** – Indicates the server from where the remote license is available.

Add a license

To add a license click the according under More Options and enter your product key. To apply a product key, the computer needs to have internet access for communication with the licensing server.

Update Licenses

The license update is the process of updating an existing license from an older software version to a more recent one. This process needs internet access. You may check for license updates. The software will tell you if there is a more recent version of the software and if you are eligible for the update (with a valid maintenance contract). Applying the update in this case will not have effect until the correspondent version is downloaded and installed.

Transfer Key

The transfer key functionality helps moving a license key from one computer to another. If you click this option, a list of all transferable license keys in the network is shown. Select the desired key and click the button Transfer to move it from the network to a local computer.

Collect license information

Collecting license information is the first step of a manual procedure of adding or updating a license. If you click this option a Save File dialog opens. The generated c2v file (customer to vendor) is used either for product key activations or for license updates. Send the c2v file with your license request to FARO. You will receive a v2c file from FARO with the according license. This file is then used for "Update from file".

Update from file

Updating a license from a v2c file is the last part of a manual process of adding or updating a license. If you click the option Update from file, a Load File dialog opens for loading the v2c file you received from FARO.

Further information about licensing is to be found in the FARO Knowledge Base:

■ for as As-Built for Revit specifically

■ General (network licenses etc.)

Note: When you start the software without a valid license and you try to execute a feature, the software will show a message that informs you that you do not have a valid license. In the same dialog you are offered to select a time limited free of charge trial.

As-Built - FARO Knowledge Base

Used in:

Project

This command opens the default browser on the As-Built for Revit FARO Knowledge Base page

6 Licence Agreement

This is a Licence Agreement, not a Purchase Contract, between you (herein after known as the "Licensee") and FARO Technologies, Inc. (herein after known as the "Licensor" or "FARO"). By the opening of the sealed program package you are agreeing to the conditions of this Agreement. The rights to use, transfer and copy are limited by this Agreement.

1. Subject of this Agreement

The subject of this Agreement is the licensing of a FARO software package. It is delivered to the Licensee in the form of a software program. The Licensee is not entitled to claim the source code. The software package also includes a disc, manual and software copy protection.

The parties to this Agreement hereby agree that the program is eligible for protection and of being protected under copyright law. The Licensor or any third party have intellectual property rights on the goods supplied. So far as any third party is entitled to the rights, the Licensor has the appropriate rights of use.

2. Scope of Use

The Licensee is hereby granted for the duration of this Agreement the simple non-exclusive and personal right (licence) to use the computer programs on the disc subject to the following conditions.

The licence for the software may not be used by the Licensee at any one point in time on more than a single computer.

The Licensee has the right to make a backup copy of the programs. The program package and the company names, trade marks and copyright notices contained therein, must not be changed or altered in any way or manner. The Licensee is not entitled, in any way or form, to copy or otherwise duplicate the software and the written material, in whole or in part in its original or in any form merged with or embedded in any other software.

The Licensee hereby agrees not to disassemble or have disassembled the programs or any part thereof. Should the Licensee require information on how to achieve interoperability, then it will be supplied by the Licensor. Likewise it is not allowed to translate the written material or to amend or to produce derivative works there from.

The Licensee is not entitled to remove the software copy protection that is bound to the software or to bypass it in any other way.

The Licensee has to ensure that no programs, documentation or duplicates thereof can be accessed by any third party without the express permission of the Licensor. In particular the Licensee is not allowed, without the express permission of the Licensor, to transfer, to rent or to loan the software to any third party. Basically, a transfer is only possible if the third party has declared to comply with the terms and conditions of this Agreement. This must be expressly

pointed out to the third party. With an approved transfer all the rights of use of the transferor are forfeited.

Any further use is prohibited.

3. Warranty

The software has been developed with due care and diligence and with regard to accepted programming methods. With the currently best available technology it is not possible to completely rule out errors in the development of the software.

The Licensor accepts no liability for any errors and does not guarantee the specific suitability of the software to meet the Licensee's requirements. The responsibility for the correct choice of and the consequences arising from the use of the software as well as the intended or achieved results lies solely with the Licensee. Warranty claims are excluded, to the extent allowed by law.

The Licensor is in particular not liable for any indirect damages or any consequential damages caused by errors or defects in the licensed software. There shall be no limitation of liability for wilful intent, gross negligence and guaranteed quality.

4. Loss of the Dongle

FARO products are protected by a dongle. The dongle contains complete licensing information. Therefore your licence is irremovable embedded in your dongle. If the dongle is lost the Licensee is obligated to promptly report the loss of the dongle to FARO (Licensor). The dongle (the licence) will not be replaced by FARO.

5. Duration of the Agreement

This Agreement is open ended (none expiring). The Licensee's rights to use the software will automatically expire without notice, if he breaks any one of the conditions of this Agreement. Upon expiration of the right to use following a breach of this Agreement the Licensee is obligated to destroy all parts of the program package supplied with this licence, together with all copies including any modified copies.

6. Final Provisions

Should one or more of the provisions of this Agreement be or become invalid, or should the Agreement contain an omission, all of the remaining provisions shall retain their legal effectiveness. The parties agree that the invalid provision shall be replaced by a valid provision that come closes to the commercial effect intended by the parties hereto.

Any change to this Agreement must be made in writing. There are no ancillary verbal agreements.

This Agreement shall be subject to the laws of the United States of America. The executive place of jurisdiction for all disputes under this Agreement shall be the state of Florida, USA.