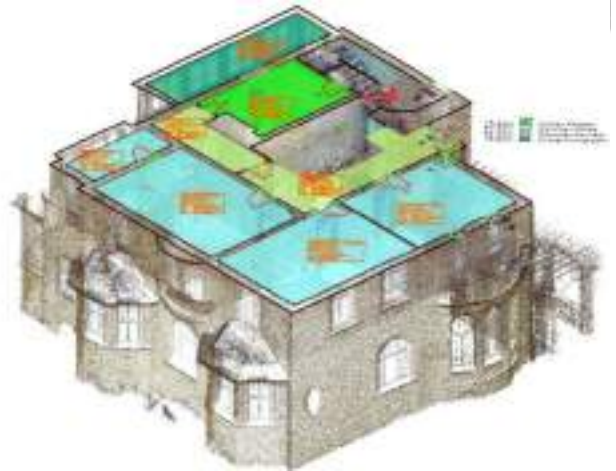
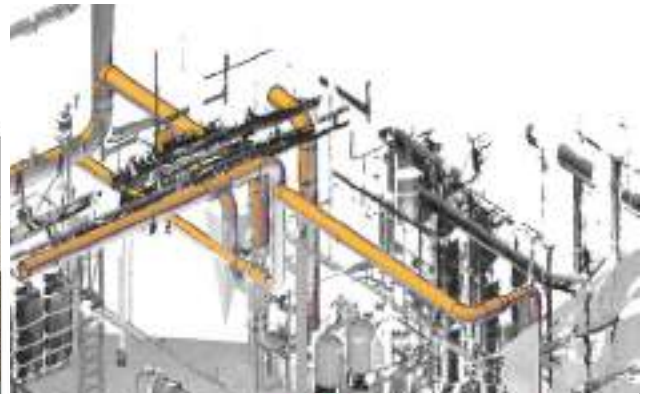
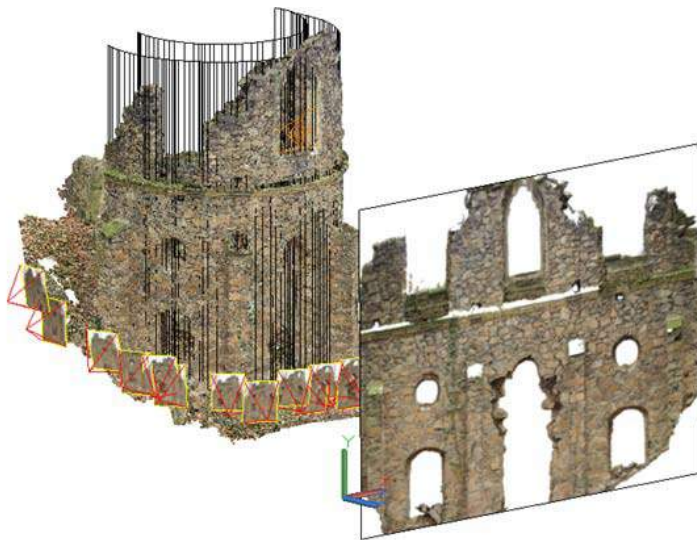




As-Built

for AutoCAD® Software

Manual



FARO

SYNERGY
POSITIONING SYSTEMS

For more information contact Synergy Positioning Systems or
visit the Synergy Positioning Systems website at www.synergypositioning.co.nz
All branches: Phone 0800 867 266 Email: info@synergypositioning.co.nz

As-Built for AutoCAD 2018.0

The information and directions in this manual are up to date. FARO Technologies Inc. does not accept liability for the use, violation of patent laws or rights of third parties resulting from the use.

FARO Technologies Inc. reserves the right to carry out product alterations serving the technical progress at any time without previous notice.

No part of this documentation may be reproduced in any form (print, photocopy, or any other procedure) or processed, duplicated or distributed with the use of electronic systems without the previous written license of FARO Technologies Inc.

As of Mai 2018

Contact

FARO Technologies, Inc.

www.faro.com

Knowledge Base:

<https://knowledge.faro.com/>

Trademarks

AutoCAD® is a registered trademark of Autodesk Inc.

As-Built for AutoCAD uses "Visualization Toolkit" (vtk), Copyright © 1993-2004 Ken Martin, Will Schroeder, Bill Lorensen All rights reserved.

The sample point clouds used in this manual and in the software were provided by Arctron (www.ARCTRON.de) and Riegl USA.

A few Words concerning this Manual

Dear users:

We would like to thank you for the trust you put into us and wish you much success in the daily work with FARO software. You can rest assured that we have done our best in order to develop As-Built into a system that enables you to complete your tasks much more easily.

In order to guarantee an optimal introduction into the work with As-Built for AutoCAD, this manual contains examples that will lead you step by step through the representation and analysis of point clouds in AutoCAD. Chapter 3 will explain how to import point clouds in AutoCAD and shows how to use simple tools to evaluate point clouds. In the following chapters you will get to know the comprehensive functionalities of As-Built for AutoCAD. The command reference helps you to become acquainted with the most important details of the single commands and enables you to process own projects. Therefore we suggest taking some time to read these chapters in advance.

In addition, the user documentation of As-Built for AutoCAD also includes the [As-Built Photo manual](#), the user [manual for feature data management](#) and a [manual for the total station interface](#).

Please feel free to contact our support team in case of any questions and problems. Your questions and advice concerning As-Built for AutoCAD are important to us and always welcome. A system like As-Built for AutoCAD depends on the ideas and creativity of its users.

On this note, we are looking forward to work with you and hope you enjoy working with As-Built for AutoCAD.

Content

1	What does As-Built for AutoCAD do?	1
2	Installation and Start of As-Built for AutoCAD	2
2.1	System requirements	2
2.1.1	Requirements for the work with images	2
2.2	Installation	2
2.2.1	Removal of a product that is older than version 15	2
2.2.2	Installation of As-Built	4
2.2.3	Integration into AutoCAD	5
2.2.4	Licensing	6
3	Working with As-Built for AutoCAD	7
3.1	Notes for working with AutoCAD point clouds	7
3.1.1	General advice	7
3.1.2	Limited downward compatibility	10
3.1.3	Preparing point clouds using Autodesk ReCap	11
3.1.4	Export of RCP projects from SCENE	14
3.2	The Point Cloud Object	15
3.3	First steps	16
3.4	Working with point cloud sections	19
4	Modeling with As-Built for AutoCAD	20
4.1	kubit Planes	20
4.1.1	Wireframe model	21
4.1.2	Surface model	25
4.2	Flatness analysis – terrain and other surface models (volume computation)	26
4.3	kubit Cylinders	27
4.4	kubit truncated cones	28
4.5	Cylinder analysis and analysis of frustums of cones	29
5	Working with the Plant Tools	31
5.1	The As-Built Plant Workflow	31
5.2	The Plant Model (PSPM)	32
5.2.1	Overview of the Plant Model	32
5.2.2	Editing and adding items	32
5.3	Configuration of Parameters	34

5.4 Walk The Run	34
5.4.1 Understanding the WTR Interface	34
5.4.2 Probability and Verification	35
5.4.3 Proceeding Through the Run	35
5.4.4 Pre-calculate cylinders	37
5.5 ApplyConstraints.....	38
5.5.1 Why ApplyConstraints?	38
5.5.2 Using ApplyConstraints	39
5.5.3 Setting Links Between Objects.....	40
5.6 Exporting.....	40
5.7 Troubleshooting.....	41
5.8 Tutorials for pipelines, steel beams and more	41
6 Tagging in 3D space	43
7 Tutorial – Generating a floor plan.....	47
8 Area management – Introductory tutorial	48
9 Working with images	49
10As-Built for AutoCAD Command Reference	50
10.1 As-Built Modeler Tools	51
Point clouds – Insert point cloud	51
Point clouds – Set current point cloud.....	54
Point clouds – Set point cloud parameters... ..	55
Point clouds – Reindex AutoCAD point cloud	57
Point clouds – Export LAS file.....	58
Point clouds – Convert to AutoCAD 2015	59
Point clouds – PTC format – Convert PTC point cloud to AutoCAD point cloud ...	60
Point clouds – PTC format - Load point cloud.....	61
Point clouds – PTC format - Unload point cloud.....	62
Point clouds – PTC format - Merge point clouds.....	63
Point clouds – PTC format - Crop current point cloud	64
Point clouds – PTC format - Set grip position.....	65
Point clouds – PTC format - Display settings	66
Point clouds – PTC format - Show / hide bounding box	68
Point clouds – PTC format - Show / hide grip.....	69
Point clouds – PTC format - Set point size.....	70
Import – Create AutoCAD point cloud (ReCap).....	71

Sections – Define slice (UCS)	72
Sections – Define horizontale slice	73
Sections – Define slice (aligned with planar AutoCAD object)	74
Sections – Shift slice.....	75
Sections – Shift slice upwards	76
Sections – Shift slice downwards	77
Sections – Change slice thickness	78
Sections – Define multiple slices	79
Sections – Define rectangular.....	84
Sections – Exclude rectangular	85
Sections – Define clipping box.....	86
Sections – Define circular	87
Sections – Exclude circular.....	88
Sections – Define circular shell.....	89
Sections – Define clipping polygon.....	90
Sections – Exclude polygon.....	91
Sections – Previous section definition	92
Sections – Section Manager [for old PTC point clouds].....	93
Sections – Section Manager [for Autodesk/ReCap point clouds].....	103
ScanNav – Create ScanLabel	109
ScanNav – Scan navigation on.....	112
ScanNav – Scan navigation off.....	114
ScanNav – Open linked VirtuSurv scan.....	115
ScanNav – Specify direction of view for ScanNav	116
Digitizing tools – SmartSnap.....	117
Digitizing tools – UCS from slice.....	123
Digitizing tools – UCS from plane	124
Digitizing tools – Vertical UCS	125
Digitizing tools – UCS from image	126
Digitizing tools – 3D orbit (with center of rotation)	127
Digitizing tools – Set elevation (from point).....	128
Digitizing tools – Align cursor.....	129
Dimension – 3D distance dimension	130
Measure – Measure horizontal distance	131
Measure – Measure height difference	132

Measure – Measure Z difference	133
Create ortho image	134
Flatten drawing.....	139
Modeling – Fit outline plan	140
Modeling – Fit polygon	145
Modeling – Fit a polygon with variable number of vertices.....	148
Modeling – Fit polygon in multiple slices	149
Modeling – Fit profile.....	154
Modeling – Draw circle (3 points).....	160
Modeling – Draw arc (3 points)	162
Modeling – Plane – Fit plane.....	163
Modeling – Plane – Plane (1-click).....	166
Modeling – Plane – Draw plane	170
Modeling – Plane – Intersection line (2 Planes)	171
Modeling – Plane – Intersection point (3+ Planes).....	172
Modeling – Plane – Intersection lines (3 Planes)	173
Modeling – Plane – Perpendicular projection of points onto plane.....	174
Modeling – Plane – Extend (2 Planes)	175
Modeling – Plane – Automatic rim.....	176
Modeling – Plane – New Rim	183
Modeling – Plane – Modify Rim.....	184
Modeling – Cylinder – Fit cylinder using 2 click points	185
Modeling – Cylinder – Fit cylinder using visible points (clipped).....	187
Modeling – Cylinder – Fit cylinder	189
Modeling – Cylinder – Draw cylinder.....	191
Modeling – Cylinder – Generate AutoCAD 3D solid.....	192
Modeling – Truncated cones – Fit circular truncated cone.....	193
Modeling – Truncated cones – Fit elliptical truncated cone.....	195
Modeling – Truncated cones – Create a truncated cone.....	197
Modeling – Truncated cones – Explode to AutoCAD 3D solid	198
Modeling – Fit 3D solid.....	199
Help – Manual	200
As-Built – About	201
As-Built – Settings.....	202
As-Built – Getting Started	210

As-Built – Knowledge Base...	211
As-Built – Software Update.....	212
As-Built – Licence management ...	213
As-Built – Customize User Interface.....	215
As-Built – Version Manager.....	217
10.2 As-Built Analysis Tools.....	218
Analysis - Clash Detection.....	218
Analysis – Analysis of arbitrary surfaces/solids	224
Analysis – Colorize an analyzed point cloud.....	227
Analysis – Create Legend.....	229
Analysis – Plane – Analysis of planes in a point cloud	230
Analysis – Cylinder – Analysis of cylindrical shapes in a point cloud.....	240
Analysis – Analyze frustum of a cone in a point cloud.....	249
10.3 As-Built Plant Tools.....	259
Prepare – Settings.....	259
Prepare – Edit Plant Model.....	264
Prepare – Define a block for a plant object.....	275
Pipes – Walk The Run.....	276
Pipes – WTR – Pre-compute Cylinders	287
Pipes – Exchange a plant entity	291
Pipes – Set link.....	292
Pipes – Remove link.....	293
Pipes – Set new links.....	294
Pipes – ApplyConstraints.....	295
Pipes – Change insulation	298
Pipes – Select pipe run.....	299
Pipes – Configure plant object’s appearance	300
Pipes – Fit bent pipe.....	301
Beams – Fit beam	303
Beams – Edit beam	310
Beams – Show the alignment of the beam:	311
Beams – Switch customisation of the beam	313
Beams – ApplyConstraints for steel.....	318
Beams – Adjust steel.....	321
Beams – Extend steel to steel	322

Beams – Extend steel to point.....	323
Beams – Intersect beam axes.....	324
Beams – Visualize beam axes	325
Beams – Align beam axes automatically.....	326
Beams – Align beam axes manually	328
Tie-in points – Determine tie-in points.....	329
Export – Export	333
10.4 As-Built Feature Data Management	341
Feature Data Management	341
10.5 As-Built Building Plans	342
Walls – Align walls	342
Building Elements – Door.....	346
Building Elements – Window.....	351
Building Elements – Staircase.....	356
Building Elements – Segments	363
Building Elements – Rectangle	364
Building Elements – Measure Recess/Ledge.....	370
Building Elements – Draw grid	372
Edit – Fix lines to corner.....	375
Edit – Perpendicular return.....	376
Edit – Horizontal extend	377
Edit – 3D extend.....	378
Edit – Perpendicular extend	379
Dimension – Absolute height.....	380
Dimension – Relative height.....	381
Dimension – Room height.....	382
Dimension – Convert relative to absolute height	383
Dimension – Convert absolute to relative height.....	384
Dimension – Set relative reference datum	385
Dimension – Subsequent editing – Set reference height of height dimension.....	386
Dimension – Subsequent editing – Show properties of height dimension.....	387
Dimension – Subsequent editing – Set absolute reference datum.....	388
Dimension – Subsequent editing – Define blocks	390
Dimension – Subsequent editing – Adjust blocks to UCS	396
Dimension – Create rotated arch plan.....	397

Dimension – Window dimension annotation	399
Dimension – Staircase dimension annotation	403
Drawing – Simple HELMERT transformation.....	405
Drawing – Advanced HELMERT transformation.....	406
Drawing – Set text height.....	409
Drawing – Plan analysis – Find double lines	410
Drawing – Plan analysis – Find gaps.....	411
Drawing – Plan analysis – Find short lines	413
Drawing – Plan analysis – Trace outline.....	414
Drawing – Plan analysis - Purge.....	418
Drawing – Clipboard – Copy.....	419
Drawing – Clipboard – Cut.....	420
Drawing – Clipboard – Paste	421
10.6 As-Built Photogrammetry Tools	423
Photogrammetry Tools - As-Built Photo.....	423
11 Adaptation and Configuration of As-Built for AutoCAD	424
11.1 Predefined blocks	424
11.2 Working with template files	425
11.3 Configuration with company guidelines.....	425
11.4 Useful AutoCAD Settings.....	428
12 License Contract	429

1 What does As-Built for AutoCAD do?

As-Built for AutoCAD® provides powerful tools to derive plans, image plans and 3D models from point clouds for as-built data collection and planning in architecture, construction and engineering.

Plant components and piping systems can be automatically detected and recorded with all their parameters. Exact connection points can be determined without modeling.

Analysis tools serve for quality control of the modeled object as well as for the determination and visualization of deformations.

The integrated [feature data capture](#) is useful for facility management and real estate management, as well as for component lists from piping systems or for damage mapping on facades of historical buildings.

[Photogrammetry tools](#) are a valuable addition to the inventory and documentation of historical buildings in particular in historic preservation and building research.

An [interface to total stations](#) complements the range of functions and enables the staking out of points from the design into reality. When measuring the total station becomes a 3D mouse. Measurements are transferred directly to AutoCAD.

2 Installation and Start of As-Built for AutoCAD

The installation of As-Built for AutoCAD consists of a number of steps which have to be carried out in sequence. Administrative rights are required for the installation.

2.1 System requirements

As-Built for AutoCAD 2018.0 requires:

Software

- AutoCAD 2015 to 2019
- the operating system according to the system requirements of the chosen AutoCAD version: Windows 7, 8, 8.1 or 10 (all 64bit).
- The photogrammetry tools, feature data management and total station interface also support AutoCAD LT 2015 to 2019.

Hardware

- Graphics card as recommended or certified by Autodesk (<http://usa.autodesk.com/adsk/servlet/syscert?siteID=123112&id=18844534>),
- RAM at least 8 GB, better 32GB and more
- SSD for larger projects (swap file)
- 64bit processor at least 2.5 GHz, better 3-4 GHz and 4-8 cores

In order to be able to have the queries to a point cloud processed with high performance (e.g. selection of points within a prism) you should also make sure you have a high CPU performance

2.1.1 Requirements for the work with images

We suggest the use of graphic cards certified by Autodesk!

You will find information regarding suitable graphic boards and drivers on the support&download pages of Autodesk. You may always search for updates and downloads within the current certification list. You may use the command "GRAPHICSCONFIG" to check on graphics performance. AutoCAD is automatically searching for graphic cards updates.

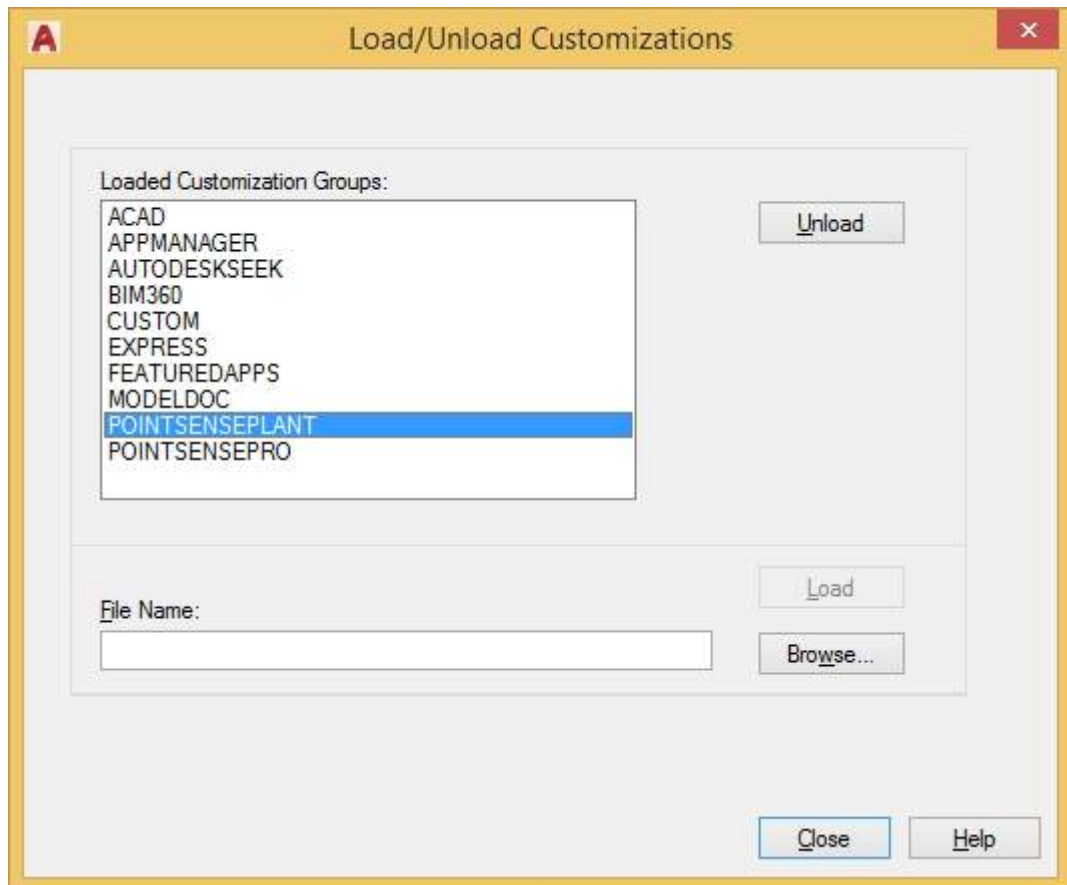
2.2 Installation

The installation of As-Built for AutoCAD consists of several steps that you have to perform one after the other. For the installation you need administrator rights.

2.2.1 Removal of a product that is older than version 15

In the case where you already have previous versions of FARO applications installed on your computer, first remove them from your AutoCAD (LT) environment.

- Remove the appropriate product menus (e.g. POINTSENSEPLANT) by entering the command "menuload" 1 in the command line. The following dialogue box appears

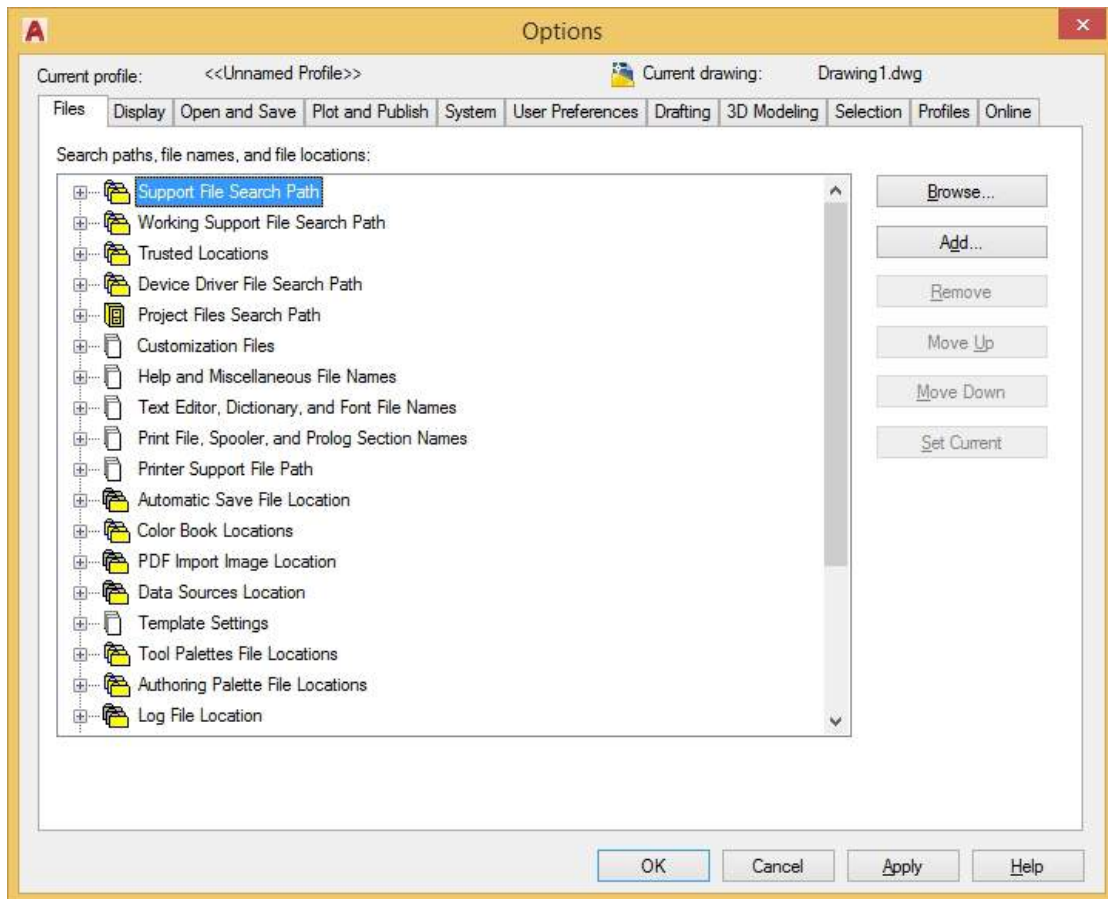


→Load/unload Customisations → Loaded Customisation Groups:

[Product name] → Unload → Close

- Remove the directory entry of the old versions (e.g. PointSense) from the search path for the support files. Enter the command "config" in the comm1 The following window is displayed. Remove the support file path.

Options → Files → Support File Search Path: Path for the Product → Remove → OK



- Close AutoCAD.
- Only when using **AutoCAD LT**: Run the file LTUNINST.EXE, to be found in the folder of your previous installation. Administrator privileges are required to do this.
- Start AutoCAD again. After the new start all products are no longer embedded.

2.2.2 Installation of As-Built

The latest setup for As-Built for AutoCAD can be downloaded from the [FARO website](#). (alternatively also setups.faro3dsoftware.com)

Double-click the setup file and follow the instructions of the installation wizard.

During the installation process, you will be prompted to select the AutoCAD version for which you want to install As-Built.

If you want to install multiple versions of As-Built for AutoCAD, create a separate folder for each version. This allows you to use the different versions (e. g. for AutoCAD 2015 and 2019) in parallel. The setup wizard supports you in this.

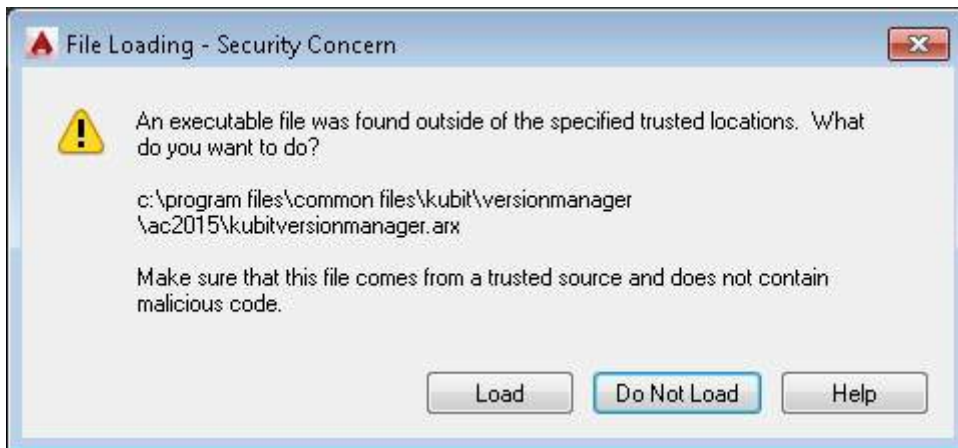
Note – Planar view of scans

If you – in addition to point clouds – want to use the more intuitive photo like planar view of scans for you evaluation, please also install VirtuSurv. The setup is included in the As-Built for AutoCAD setup. The use of VirtuSurv's link to AutoCAD is included in your As-Built for AutoCAD license.

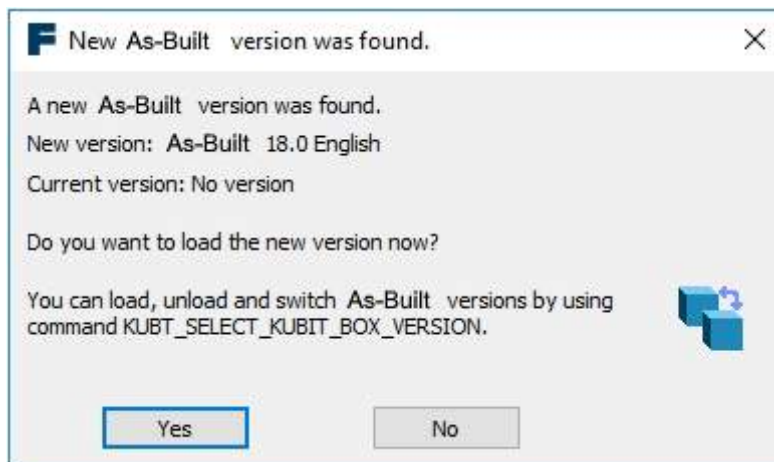
Your As-Built for AutoCAD license also allows you to use the SCENE DrawToAutoCAD App. With this App you can directly send coordinates from SCENE scan views into the AutoCAD command line without additional data conversion. DrawToAutoCAD is available for [download here](#).

2.2.3 Integration into AutoCAD

After a successful installation you can start AutoCAD (LT). Should the following security message be displayed you can simply confirm it by clicking [Load].



Should a message be displayed saying that a new As-Built for AutoCAD version has been found and asks if you want to load it, Confirm the message with [Yes]. Confirm you adaptation with [YES].



Should you want to use several versions of As-Built for AutoCAD, you can at any time by using the →Version Manager, switch back and forth between the various installed versions.

Should the message not be automatically displayed then please start the →**Version Manager** from the AutoCAD (LT) command line (command: KUBT_SELECT_KUBIT_BOX_VERSION).

As-Built for AutoCAD has now been successfully embedded in AutoCAD (LT) and is ready to be used. The associated menus will be automatically loaded. These can be seen in the automatically inserted tool bars or on the ribbon or the corresponding entry in the menu bar.

If the menu does not load automatically, enter the command `ABMENU` in the command line of AutoCAD (LT).

Ribbon or classic menu

Since version 2009 AutoCAD has offered the user a user interface with the ribbon. In all previous versions the user had a main menu bar with the points File, Modify, View, etc. which has been replaced in current versions with other structured tabs and panels (a so called ribbon). In some cases it may be meaningful to additionally display the classic main menu bar, so that specific AutoCAD commands can be more quickly found.

In order to display the main menu bar we proceed as follows:

Enter the command `MENUBAR` in the command line.

```
Command: menubar
```

```
Enter new value for MENUBAR enter <0>: 1
```

(confirm with ENTER)

The menu bar now additionally appears above the ribbon, the one that you know from the older versions of AutoCAD.

2.2.4 Licensing

As-Built for AutoCAD is protected by a softlock against unauthorized use. On explicit request, a dongle can also be provided. Without a license, As-Built for AutoCAD can be tested for a few days. If you wish to extend your trial license, please contact FARO or a FARO reseller.



With the →**As-Built – License Management** you can check the license status of your software and request a trial.

For more information about licensing, see the [Knowledge Base](#) and the command reference.

3 Working with As-Built for AutoCAD

After important remarks on the different point cloud formats that are supported by As-Built for AutoCAD, this chapter introduces you in point cloud objects and shows you how to work with As-Built for AutoCAD by the help of a simple example. A thorough description of all commands can be found in chapter **10 As-Built for AutoCAD Command Reference**.

3.1 Notes for working with AutoCAD point clouds

In the following sections specific properties of the individual AutoCAD versions with respect to AutoCAD's own point cloud format will be examined.

3.1.1 General advice

Set the *point cloud parameters* with the command →**Point clouds – Set point cloud parameters**.... The parameter value that has been set will be saved in the AutoCAD drawing, in contrast to old PTC point clouds where this parameter value is saved in the PTC file. That means when you insert an Autodesk point cloud into a new drawing this parameter must always be reset.

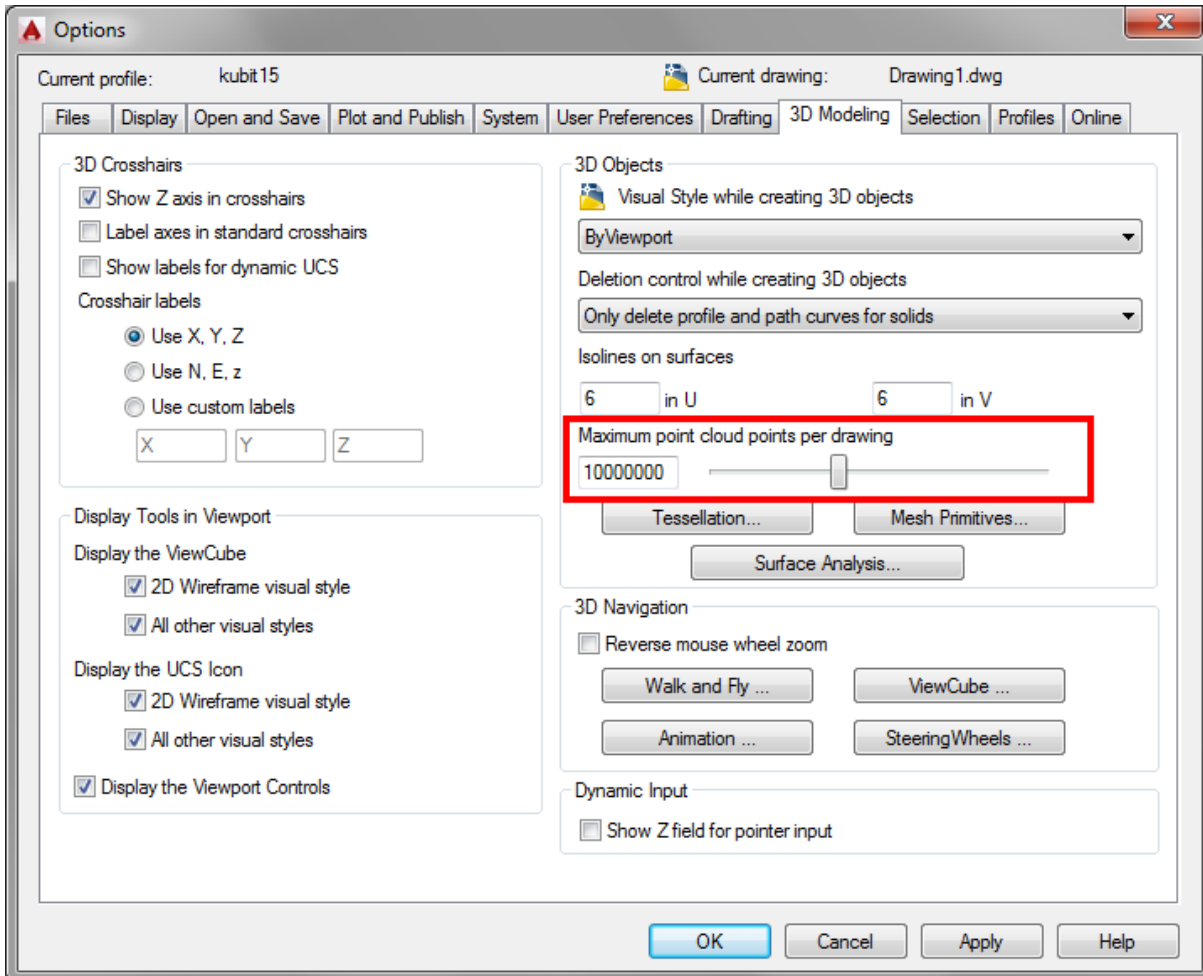
You can *work with multiple Autodesk and PTC point clouds in a single drawing*. If you are working with PTC point clouds you need to set the active point cloud using the command →**Point clouds – Set current point cloud**.

You can set the proportion of points that will be used by the pattern recognition (fitting) commands. To achieve that use the command →**Point clouds – Set point cloud parameters**... and set a value for the point density, between 1 and 100, that will be used in the alignment. A value of 1 means that only a small fraction of points will be used (fastest). A value of 100 means: Use all of the points (highest reliability).

You can physically *clip* the loaded *AutoCAD point cloud* to the currently visible region *or union* several loaded AutoCAD point clouds and then convert them with the command →**Point clouds – Reindex AutoCAD point cloud** . In both cases a new AutoCAD point cloud (RCP/RCS file) is formed.

With the AutoCAD system variable **POINTCLOUDBOUNDARY** (value: 0 or 1) you can determine whether the *bounding box* of a point cloud should be visible (1) or not (0).

The *system variable* **POINTCLOUDPOINTMAX**, allows the maximum number of displayed points to be increased to 25 million. You reach this setting in the AutoCAD "Options" dialogue box (tab: 3D Modeling, here in the example 5 million).



By increasing this value the quality of the displayed point cloud will be significantly improved. **Warning:** Should your graphics card not be capable of displaying the entered number of points, it is possible that no points at all will be any longer visible. You must then again reduce the value for **POINTCLOUDPOINTMAX**.

In the **Point Cloud tab on the ribbon** (contextual, that appears after selecting on a point cloud) various settings can be set that affect the appearance of the displayed point cloud:



Group "Display"

Point size [in pixels]: Set the size with which the points should be displayed

Detail accuracy [between 1 and 10]: Set how many points are to be displayed. The higher the value is, the better is the detail accuracy. With smaller values you can possibly become a better display performance. Each step approximately doubles the number of displayed points.

Additionally, you have in the bottom row direct access to various navigation commands.

Group "Visualization"

Here you can set various visualisation styles for the selected point cloud. The most important four examples of the visualisation styles are shown in the figure below. Further details on the visualisation styles and the lighting options (right hand column) can be found in the AutoCAD help.



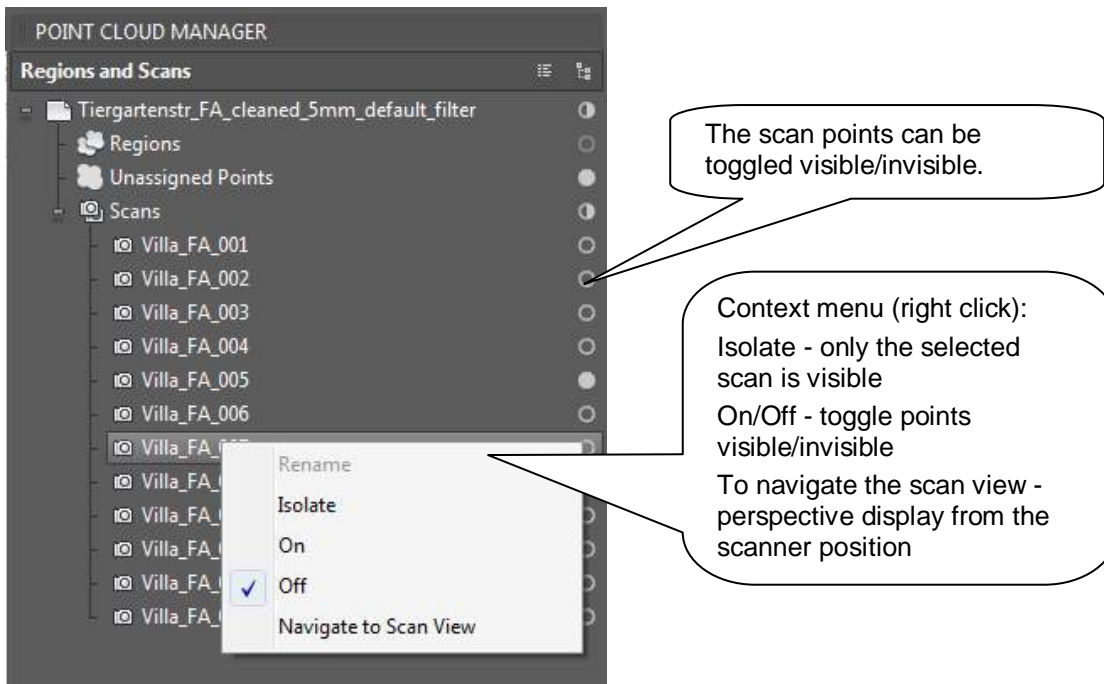
Figure 2 Visualisation styles (from left to right): RGB real colours, intensity (spectral colours), normal appearance, elevation slices

Group "Cropping"

AutoCAD itself offers three commands for cropping point clouds. But as As-Built offers a comprehensive selection of sections and clipping commands, we recommend that they be used. As-Built created cut outs or sections can be comfortably managed with the Section manager.

Group "Options"

The *Point Cloud Manager* offers as a supplement to the Sections Manager access to the referenced scan files (RCS) in the ReCap Point cloud project (RCP) and with ReCap defined regions. You can switch the individual scans on and off, isolate them or navigate the scan view.



"External Reference" opens AutoCAD's External Reference Manager. There you can attach and detach point clouds, reload or remove them from the drawing.

3.1.2 Limited downward compatibility

In **AutoCAD 2015** the embedding of ReCap point clouds has been fundamentally revised. The simultaneous retention of the backwards compatibility to older drawings and PCG point clouds hides a few interesting features, which will be looked at in this section.


Point clouds (PCG or ReCap) in a drawing, that has been created in an older version of AutoCAD, are managed in a sort of compatibility mode. *All of the As-Built features function*, that are suitable for working with AutoCAD point clouds, but AutoCAD displays *no contextual ribbon tab* for these point clouds, that means that you cannot change the appearance of the point cloud.

We recommend to convert such a drawing by using the As-Built command **PCCONVERTDWGTO2015** (→Point clouds – Convert to AutoCAD 2015) in a standard AutoCAD 2015 drawing.

Following is important for **AutoCAD 2016 and above**. Old drawings which contain point clouds in PCG format can still be opened and the point clouds are displayed, also the As-Built functions still work for them BUT the AutoCAD Object Snap „node“ is no longer available for PCG point clouds which makes working with them very hard.

For AutoCAD 2016 and above we recommend to work with ReCap point clouds only! Please convert the old point clouds into the new format.

3.1.3 Preparing point clouds using Autodesk ReCap

Please, start Autodesk ReCap and click on „New Project“ for creating point clouds in ReCap format. You can start ReCap directly from inside AutoCAD using following Button  .



Please enter a project name and directory then click on [proceed]:

create new project

first, we need a project name

new project

then a place to save your files

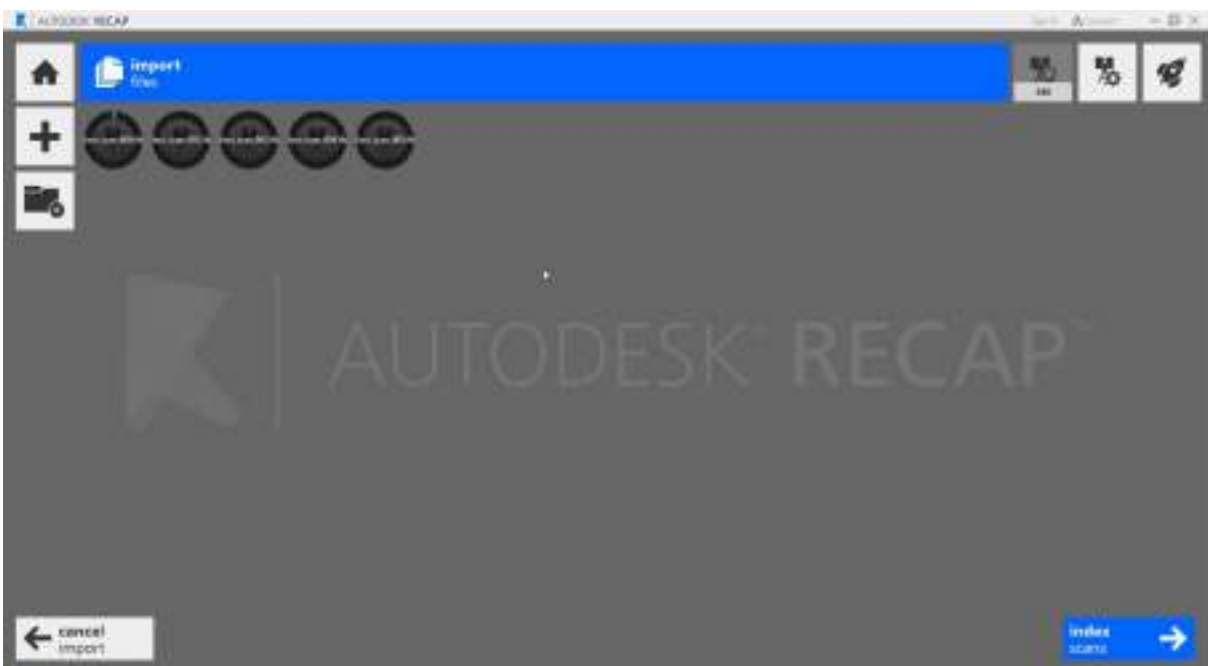
C:\Users\jana\Desktop

cancel proceed

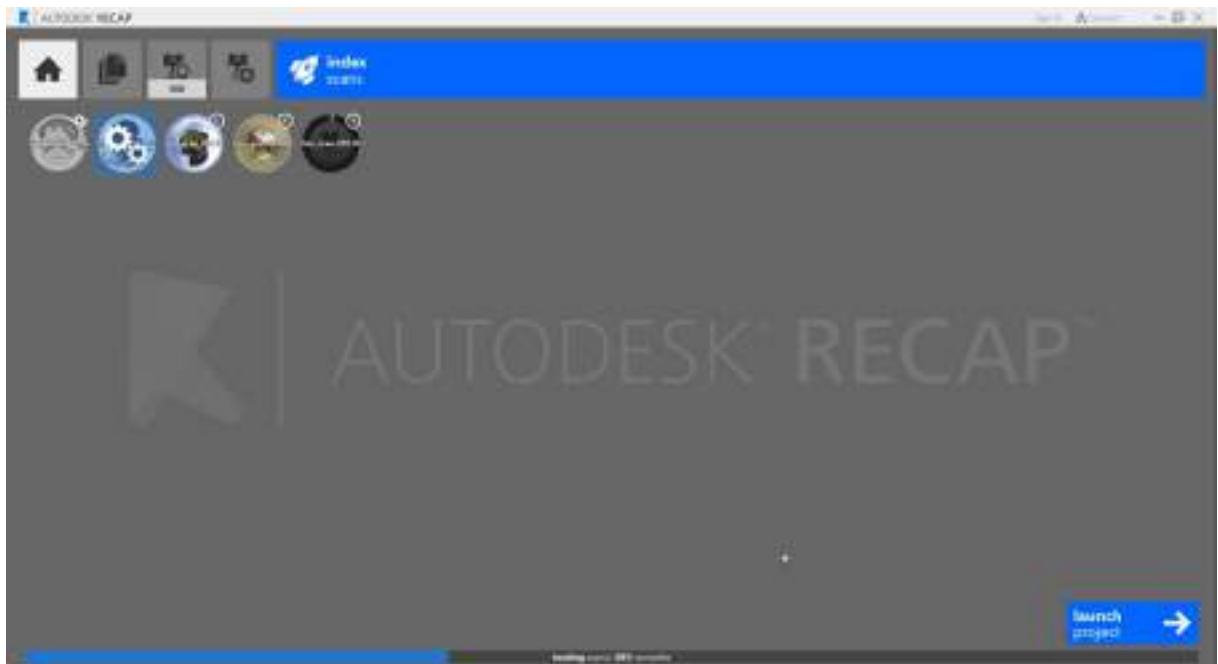
Select the scan files for import. ReCap supports most current scan data formats also the ASCII format PTS, the E57 standard format and the older Autodesk PCG format. You may drag&drop scan files into ReCap, select a folder containing scan data or add import files via single file selection.



After selecting the import files the scans are loaded automatically. You will see the progress at the scan thumbnails (here the first scan is already loaded):

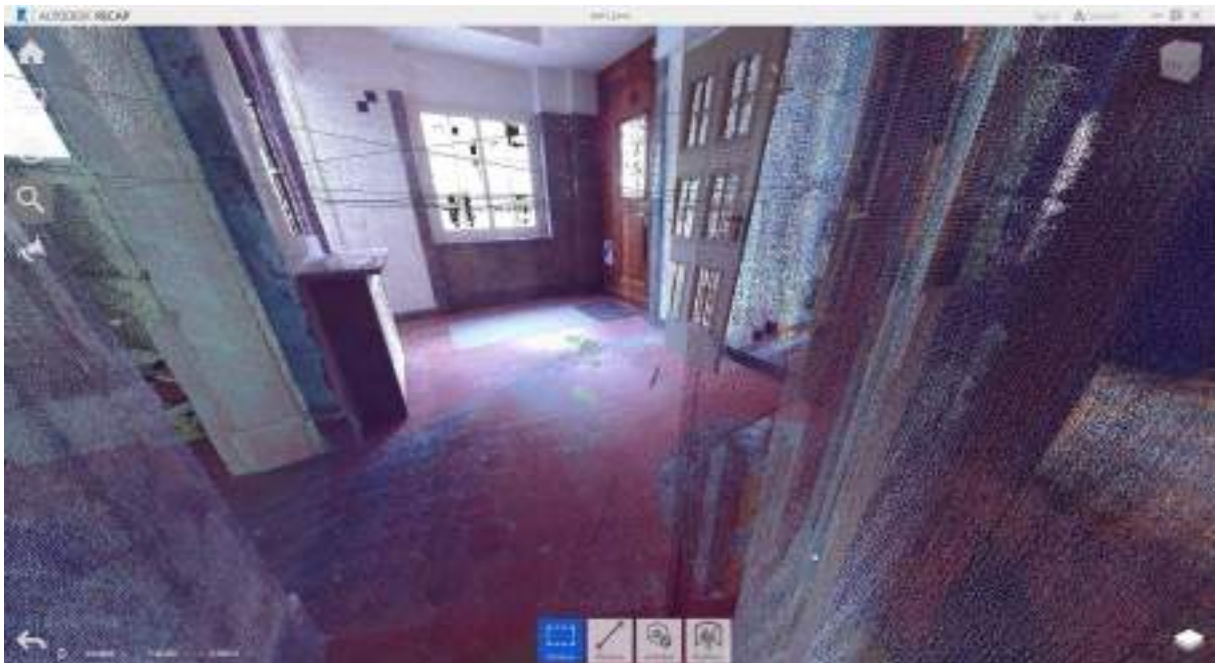


When the first scan is loaded you may start the import by clicking [index scans→]. The progress of import is displayed:



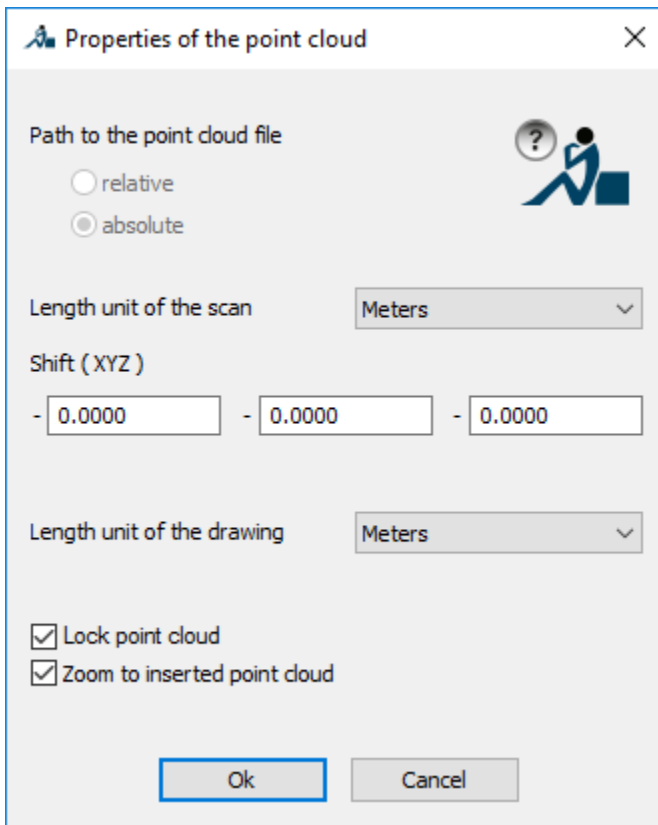
After the first scan has been imported (“indexed”), you may open the point cloud view ([launch project →]). In the background the other scans will be imported.

After the import is finished you should save the project.



A RCP will be created which references all imported scans (RCS files).

The RCP files as well as the single RCS files can be inserted in AutoCAD using the As-Built command PCLOAD. Please note that ReCap point clouds are always saved in meters. Therefore please set the „Length unit of the point cloud“ accordingly.



For more detailed information on Autodesk ReCap refer to:

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

3.1.4 Export of RCP projects from SCENE

You may directly export Autodesk point cloud projects (RCP) from our scan registration software SCENE. Please refer to the SCENE manual for further detail.

3.2 The Point Cloud Object

You may directly load a point cloud object into AutoCAD by inserting a point cloud in the PTC file format or in the Autodesk file format (RCS/RCP).

The real point cloud data is only referenced into the drawing. Thereby the DWG file stays small. Please note that when passing on a drawing with point clouds in it, that the referenced point cloud files must also be passed on. Otherwise the point clouds cannot be displayed by the person receiving the drawing.

Point clouds are characterized by their size (number of points), their noise (n) and the average distance of points (a). Because different modeling and fitting algorithms need these parameters for correct functioning, you have to specify values for n and a .

Values for n and a are saved:

- with PTC point clouds in the PTC file itself
- with Autodesk point clouds in the drawing

When you insert an Autodesk point cloud into a new drawing the parameter must be set (→**Point clouds – Set point cloud parameters...**, PCSETPARAMS). The default value for the point cloud parameter is held in the configuration file.

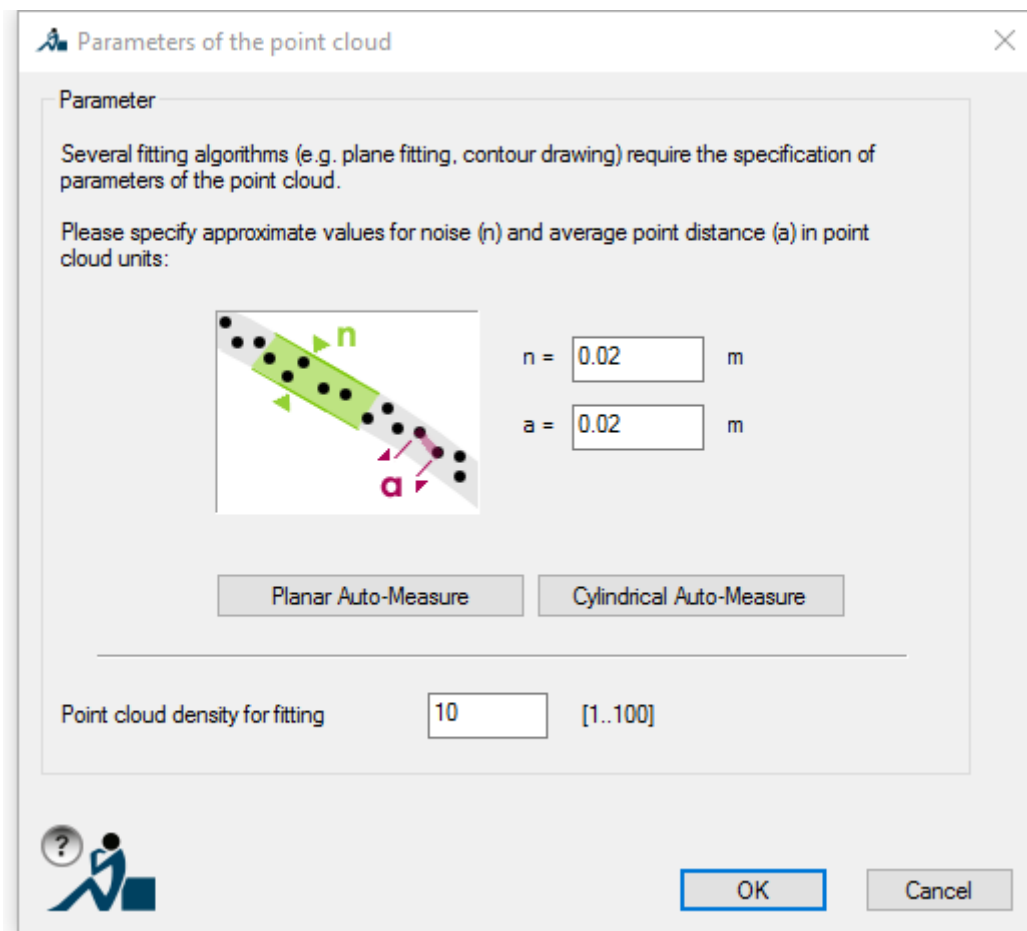


Figure 3 Dialogue box for setting the point cloud parameter

3.3 First steps


We recommend that you carry out the following realistic example step by step in order to get a first impression of the potential of As-Built.

A completely installed version of AutoCAD is required in which the corresponding As-Built application was included. If you have not done this yet please refer to section 2 **Installation and Start of** of this manual.

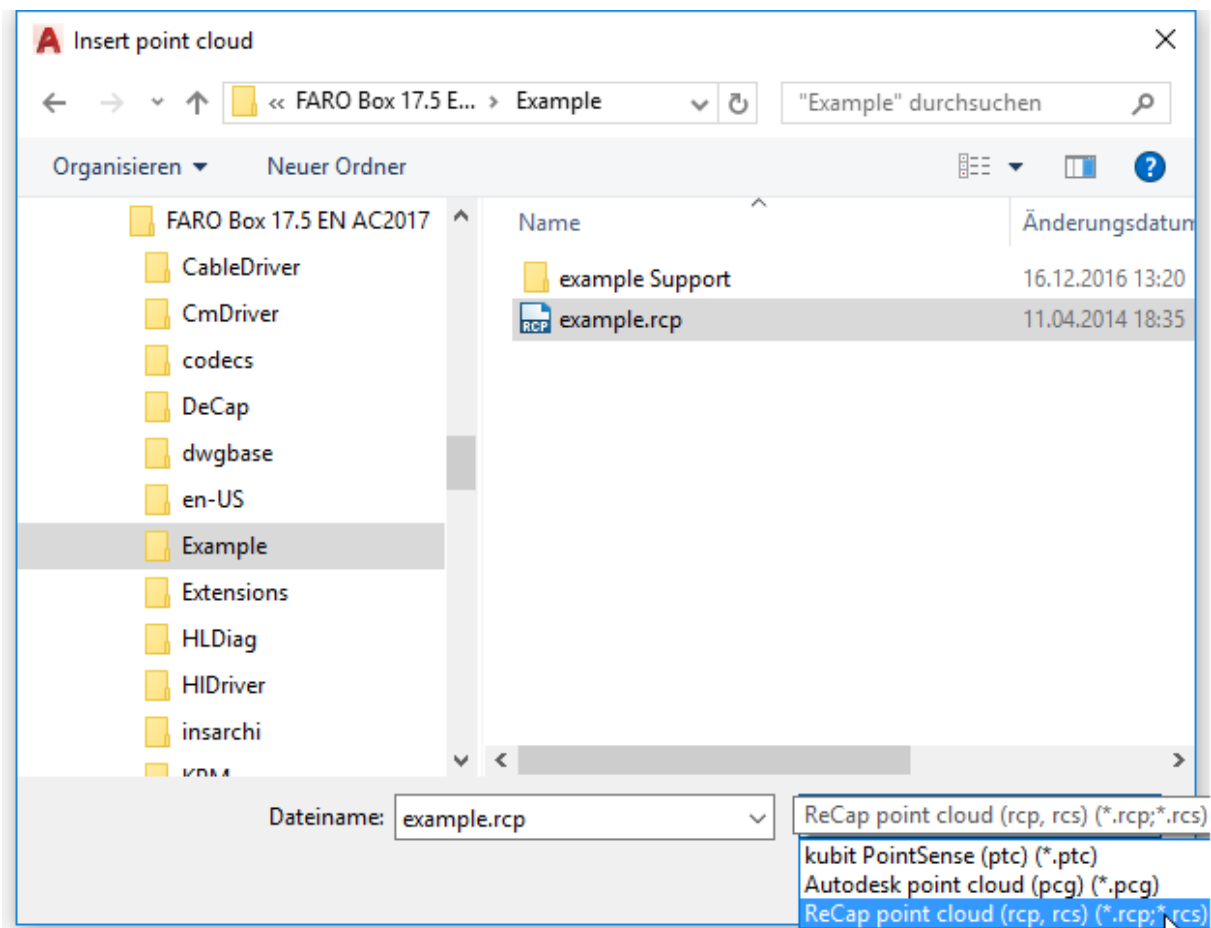
During the first step we will load a point cloud object into an AutoCAD drawing.

The loading of point cloud files is faster than the import of ASCII files (e.g. *.TXT, *.PTS, *.ASC) or data of other formats (e.g. Leica PTZ format). For that reason this example uses already existing AutoCAD point cloud files.

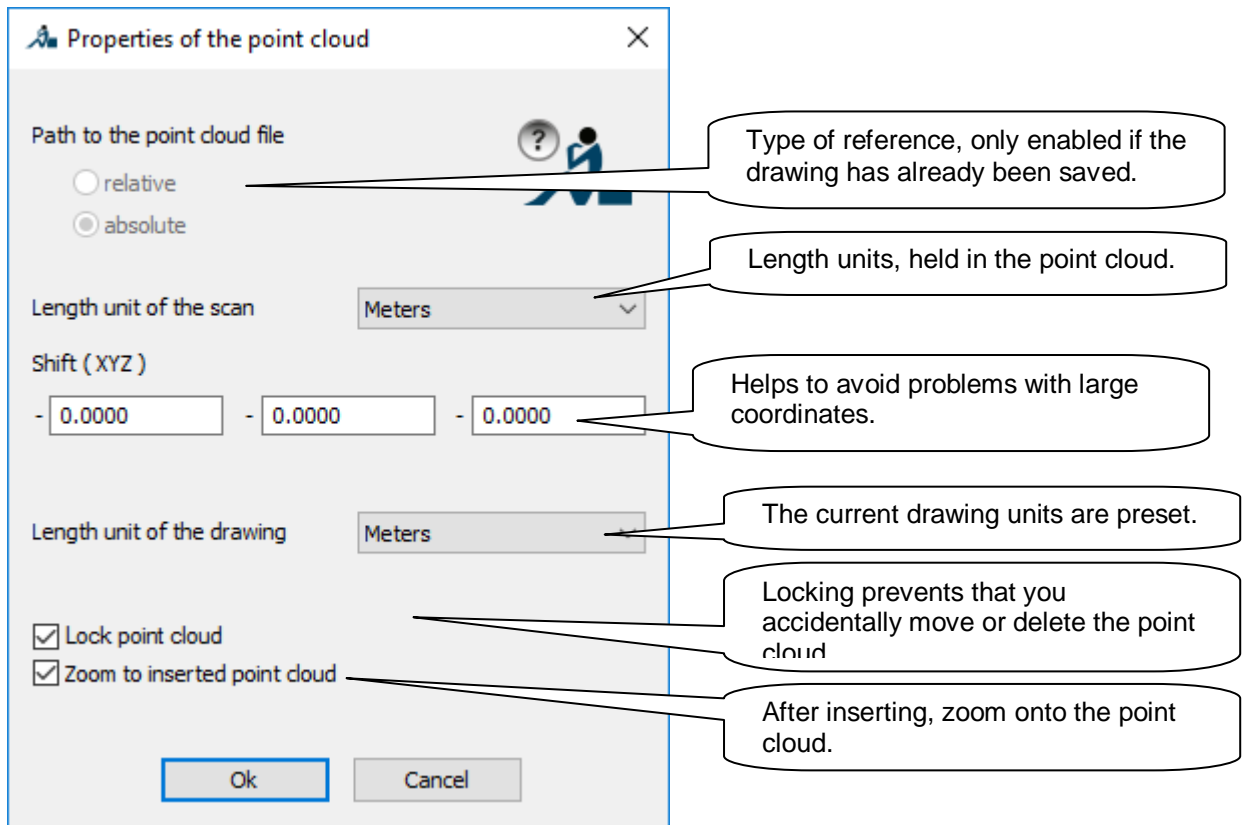
Start AutoCAD and open an empty drawing.

Select the As-Built-command → **Point clouds** – Insert point cloud (**PCLOAD**) or use the button  from the As-Built toolbar or ribbon.

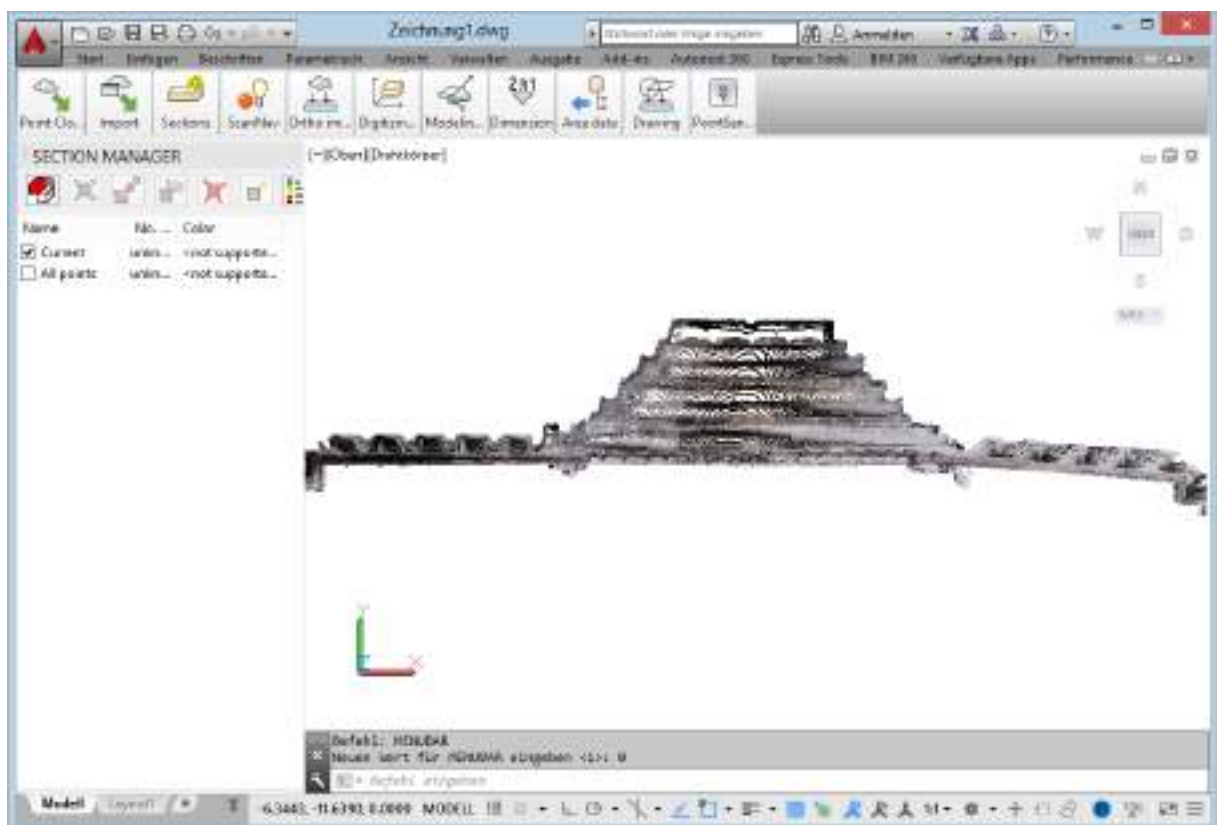
The following window appears.



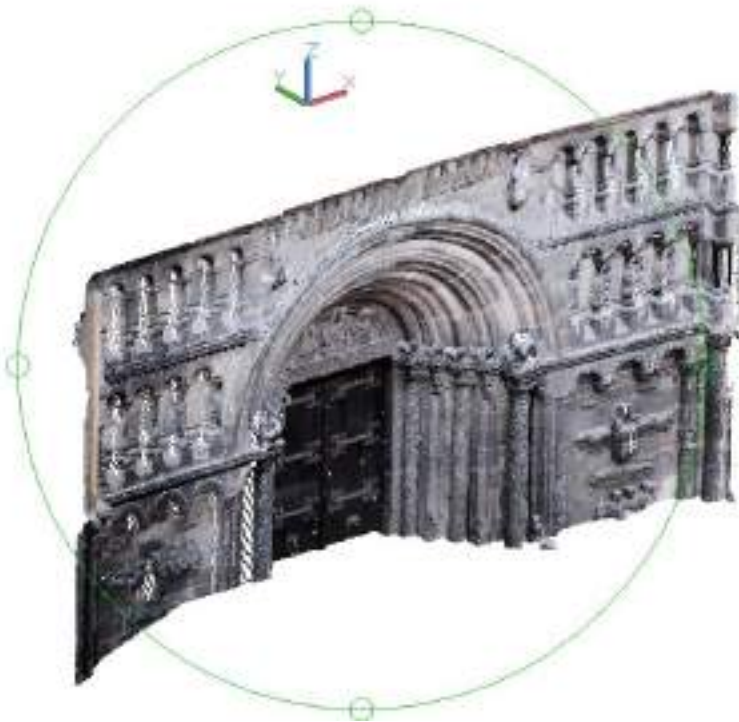
Browse to the installation directory. Under \EXAMPLE select the point cloud file EXAMPLE.RCP. Confirm your selection with [OK].



With the AutoCAD command "ZOOM → EXTENTS" you may scale the view of the point cloud within the borders of your workspace. You should now be able to see the point cloud within AutoCAD. The bounding box shows the borders of the point set and serves, as well as the grip of the point cloud, as handle for the point cloud object.



By using the AutoCAD command **3DFORBIT** or pressing the corresponding button in your AutoCAD user interface you can look at the point cloud from different viewpoints.

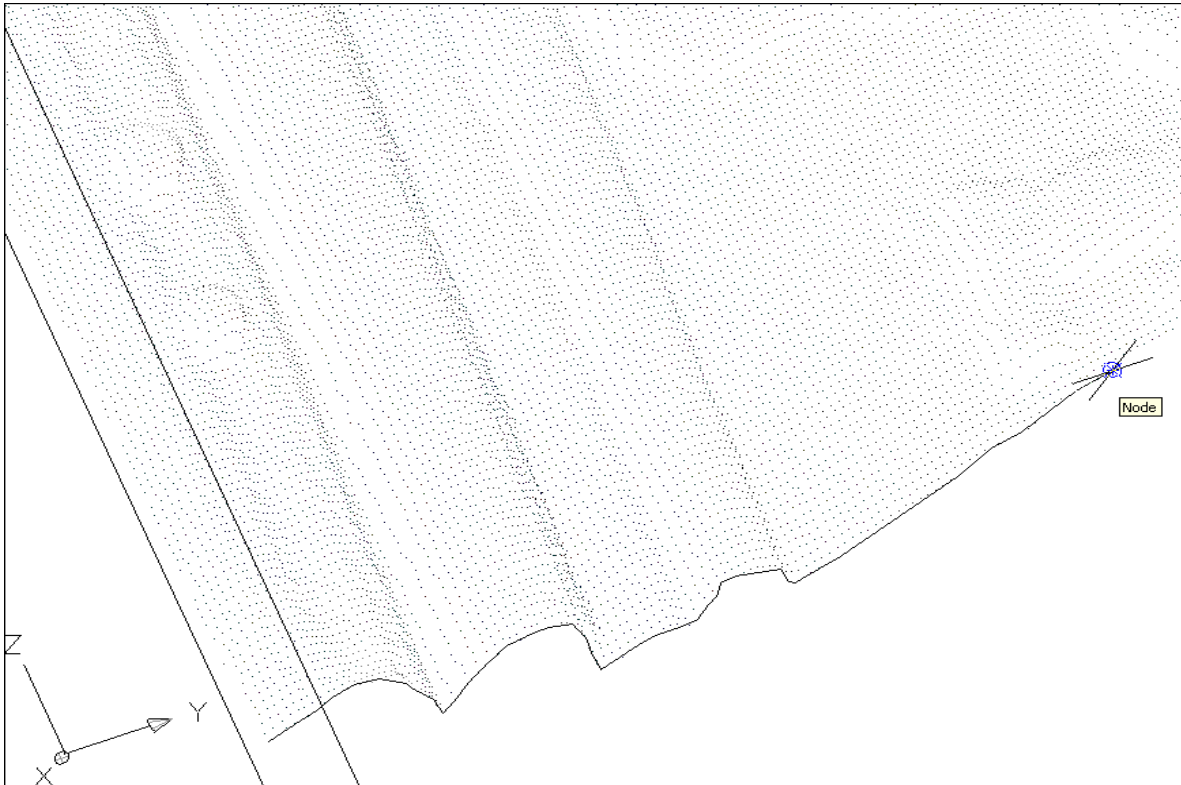


By using the AutoCAD function "Object snap" (**OSNAP**) you may place the mouse exactly above the coordinates of a single point. Thus, line connections for example, are possible. When working with As-Built, you may snap that node from a point cloud that is closest to the mouse pointer, by using the 3D object snap setting "Node of point cloud".

Just try it. With the AutoCAD command "Object snap" (**OSNAP**) you can check whether the object snap has been activated.

The AutoCAD point cloud object snap is set with the command **3DOFANG**.

Try to draw simple line runs.



With this tutorial you got a first insight into As-Built. You can find all As-Built commands in chapter **10 AS-BUILT FOR** AutoCAD Command Reference. There you will find information on specific point cloud related object snaps → **SmartSnap**, which make modeling more efficient.

Note: As-Built also supports the commands “redo” and “undo” ().

3.4 Working with point cloud sections

For the interpretation and modeling of an object it is often suggestive, to split the point cloud into well defined sections. By defining slices of a set thickness and with a certain gap, profiles can be generated and digitized. You can get an overview of a complex point cloud when coloring the individual sections.

Within the As-Built menu, under "Sections" you will find many commands for the definition and the work with slices and sections of point clouds. Next to the possibility to define sections, you may use the **Section Manager** (→ **Sections – Section Manager**). With its help you can manage the individual sections and parts of a point cloud.

Thus, the export of cut areas into a separate point cloud file is possible. Or you may transfer sections, which you have defined within a point cloud, to adjacent point clouds. This is especially useful, if an object to be interpreted consists of several scans or point clouds, which are very large and which are better loaded separately for detail interpretations.

Further interesting commands for editing point clouds are provided within the As-Built Modeler ribbon under “Point clouds”.

You may find further information regarding the commands for the work with point clouds (sections) within the → **As-Built for** AutoCAD Command Reference.

4 Modeling with As-Built for AutoCAD

The evaluation of point clouds with standard AutoCAD tools is not always simple. As-Built for AutoCAD provides powerful functionality which simplifies the creation of line drawings and generalized surface models. There are powerful algorithms for fitting geometry like polygons, planes, cylinders.

For example, after you defined a slice of the cloud you can best fit polygons or lines to that slice. The commands **→Modeling/Construction – Fit outline plan** and **→Modeling/Construction – Fit polygon** are described in detail in the command reference. Such a fitting leads to more accurate results than digitizing manually.

As-Built defines some new object classes (custom entities) inside AutoCAD, like the kubit plane, the kubit truncated cone or the kubit cylinder or the kubit pipe. These custom entities serve as surfaces and construction tools for modeling and digitizing inside point clouds.

4.1 kubit Planes

Planes are created by the commands **→Modeling – Plane – Draw plane**, **→Modeling/Construction – Plane – Plane (1-click)** and **→Modeling – Plane – Fit plane**. With the latter you can create an average plane according to the method of Least Squares by using the points of a point cloud, of AutoCAD objects or single selected points.

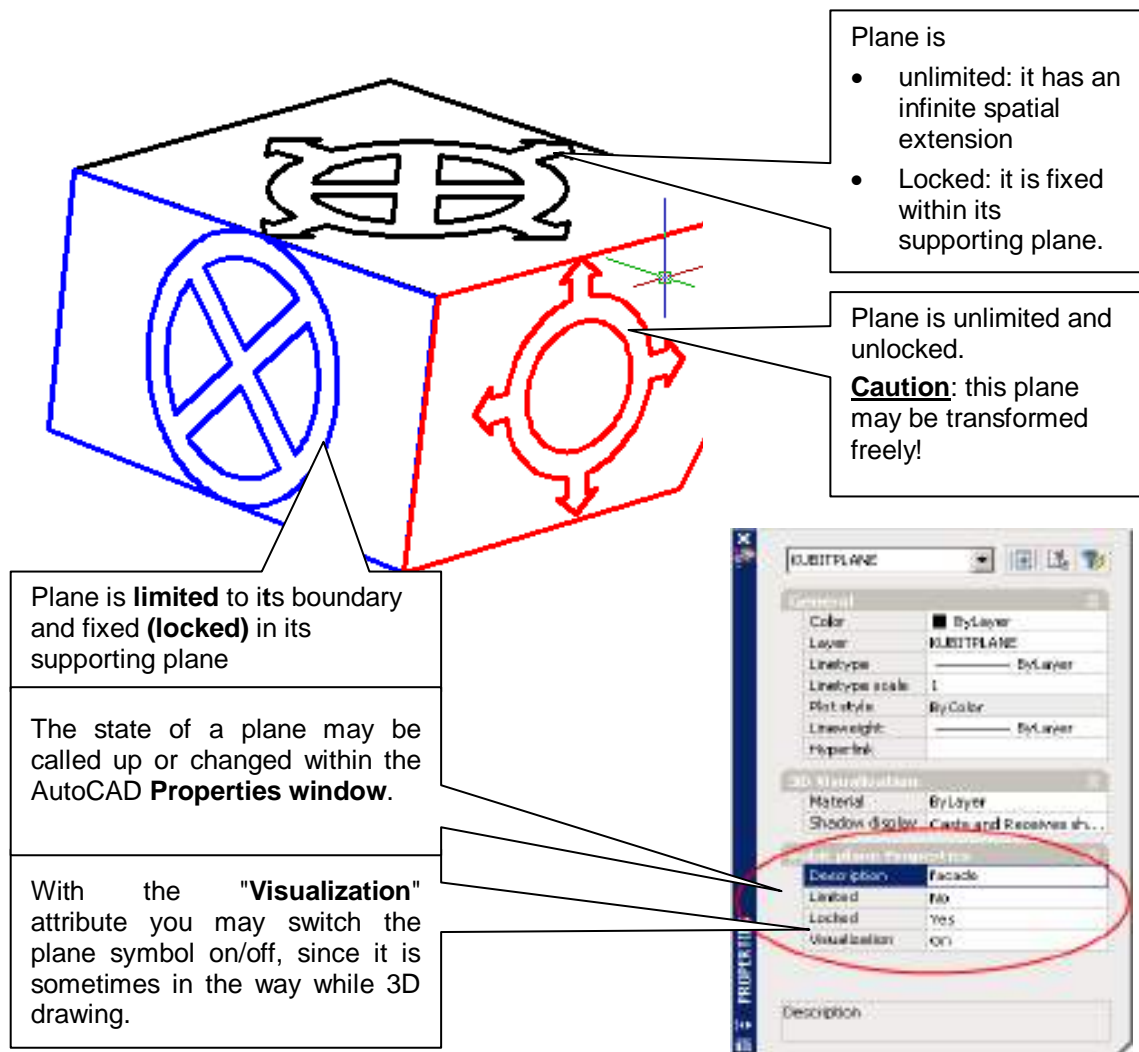
With the **→Modeling/Construction – Plane – Automatic rim** command you may let an (existing) plane grow to its boundaries in the point cloud (Region Growing). For further detail on this powerful command please, refer to the command reference.

Planes are surface objects and construction tools. They can be edited and intersected with each other. This way you can create intersection lines and points. the appropriate commands are described in more detail in the command reference.

Furthermore it is very easy and fast to generate a user coordinate system (UCS) from a plane with As-Built for AutoCAD (**→Digitizing tools – UCS from plane**).

Kubit planes are visualized by a plane symbol and may have different states. The current state is indicated by the symbol:

- **limited and unlimited:** Limited planes stretch out only to their visible boundaries. Outside the boundaries the plane is not defined. Unlimited planes stretch out infinitely, that means they stretch beyond their visible boundaries.
- **locked and unlocked:** Locked planes are fixed within their supporting plane. By gripping and moving the symbol the boundary will change but not so the mathematical supporting plane. In contrast, unlocked planes may be transformed freely, that means, they may be moved spatially.



All of these plane attributes (description, boundary, locked, visualization) may be configured within the AutoCAD Properties window (see figure)

When exploding a plane (AutoCAD command "EXPLODE") a 2D polyline is generated.

It is recommended to use only limited planes for the 3D drawing. That way you make sure, that the 3D objects are only generated within the visible plane boundary and not – as it would be for an unlimited plane – on the (indefinite) mathematical supporting plane.

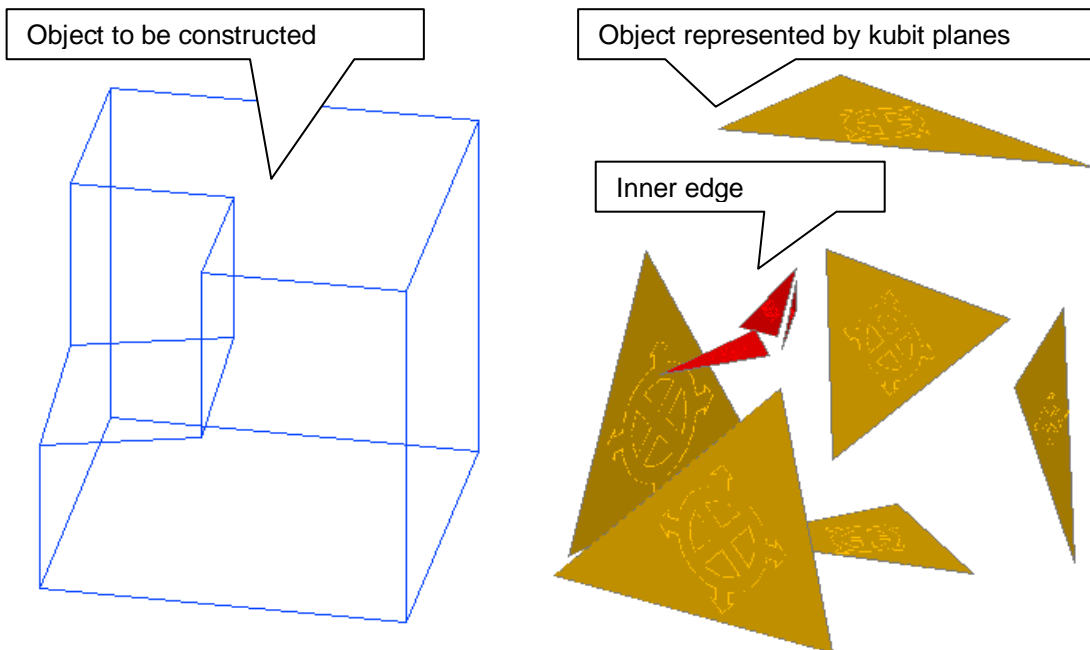
The following chapters will give you advice on how you can use kubit planes for creating 3D models (wireframe or surface model).

4.1.1 Wireframe model

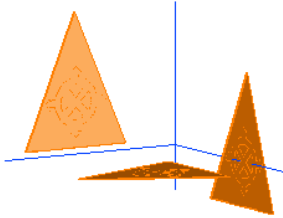
To construct the wireframe of an object you first create all planes you need by using the **→Modeling – Plane – Fit plane** command. This command also supports multiple image evaluation. The planes are drawn on the current layer.

When all planes of the desired object are created you may use the AutoCAD orbit commands to get an overview of the position of the planes to each other. It may also help if you assign different colors to the planes or groups of them. Additionally you

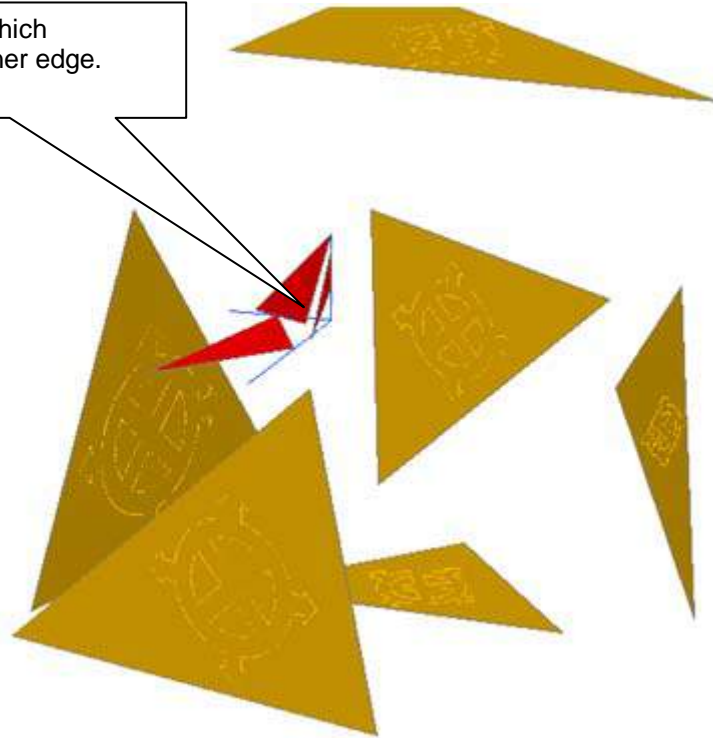
may toggle the visual style from wireframe to conceptual or realistic to improve the display of the planes. As-Built for AutoCAD provides some of these commands in its own ribbon in the 'View' panel to make them available also in perspective views (camera navigation).



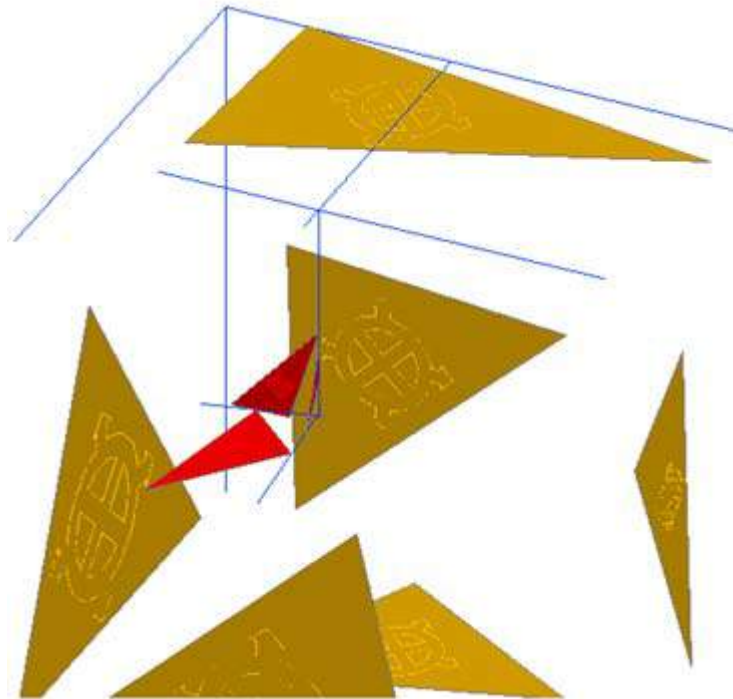
When the needed planes are created, you intersect three planes at a time using the **→Modeling – Plane – Intersection lines (3 Planes)** command. A triad is being created, which represents the corner and the three belonging edges. The length of the edges is defined by the displayed boundaries of the kubit planes. Due to this fact it may happen that the edges of the triad do not end at the intersection point - as one might expect (see figure on the left).



Here is the triad which represents the inner edge.

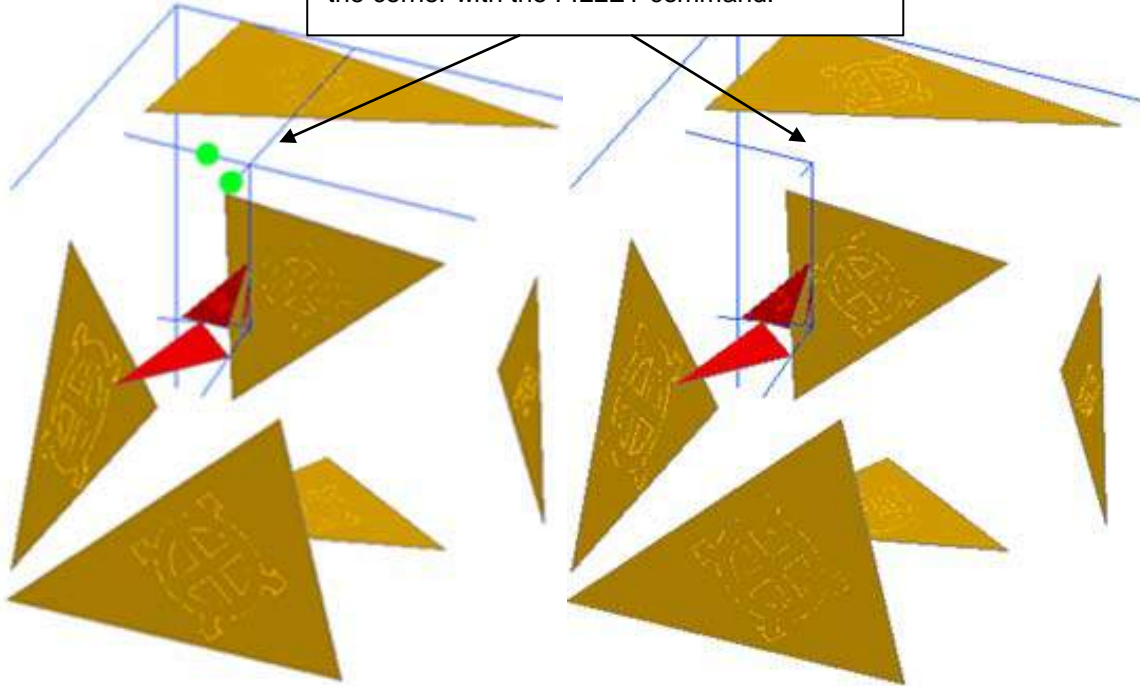


To keep track during modeling we recommend generating just a few corners at once and then continuing.



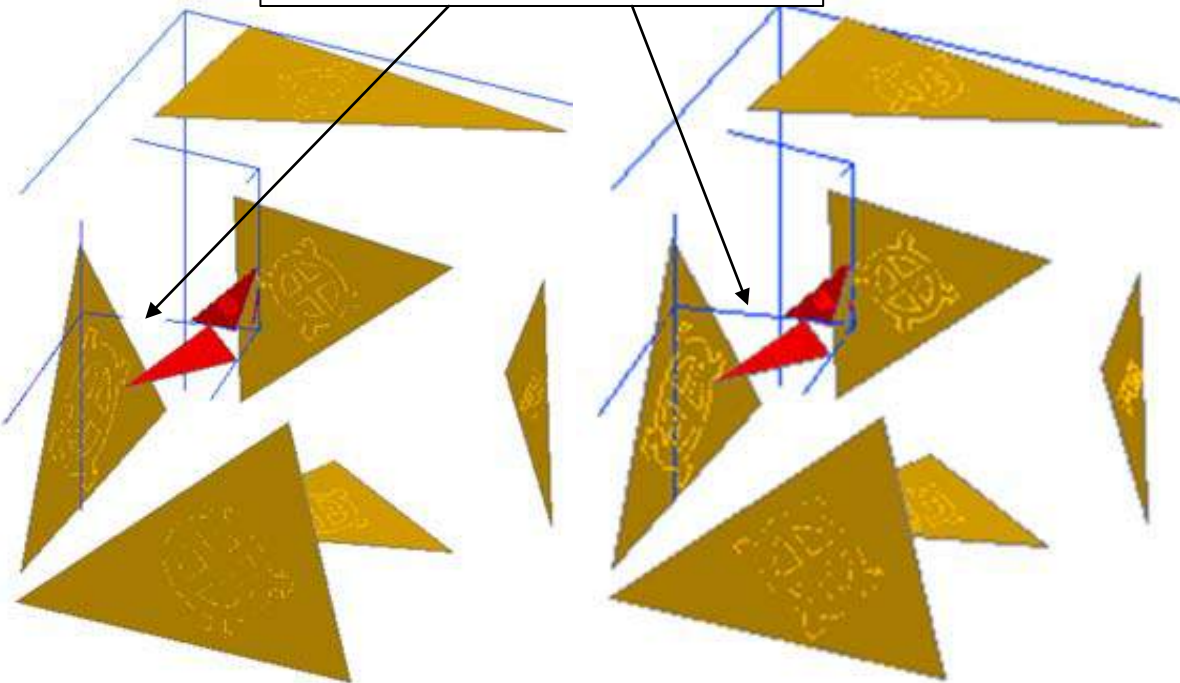
Use the AutoCAD command `FILLET` to exactly intersect the edges to a corner. Ensure that the mode is set to `Mode = TRIM` and the radius to `Radius = 0.0000`. You have to click the edges (lines) at the end you want to keep. Refer to following figure:

Click these two points on the edges to create the corner with the FILLET command.

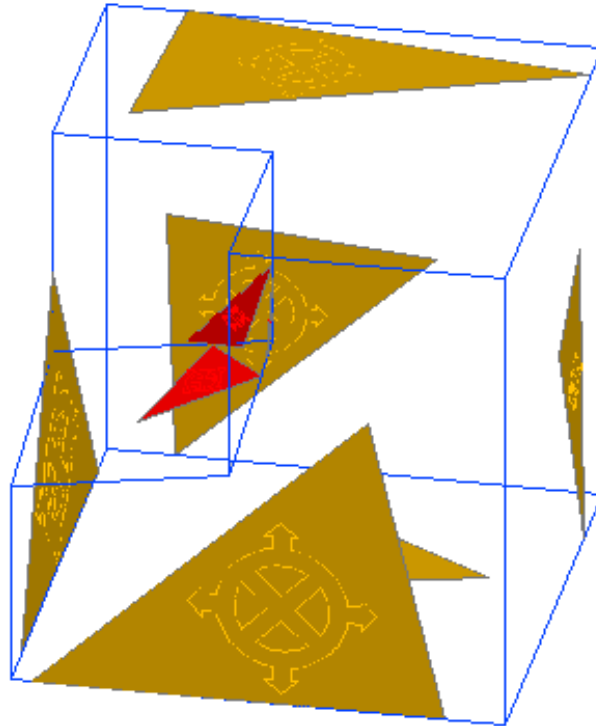


If two edges of two opposite triads point to each other, you can use the AutoCAD command JOIN. It joins two collinear lines to one. It does not only work for closing gaps but also if the two lines overlap.

When joining two collinear lines you get one single line.

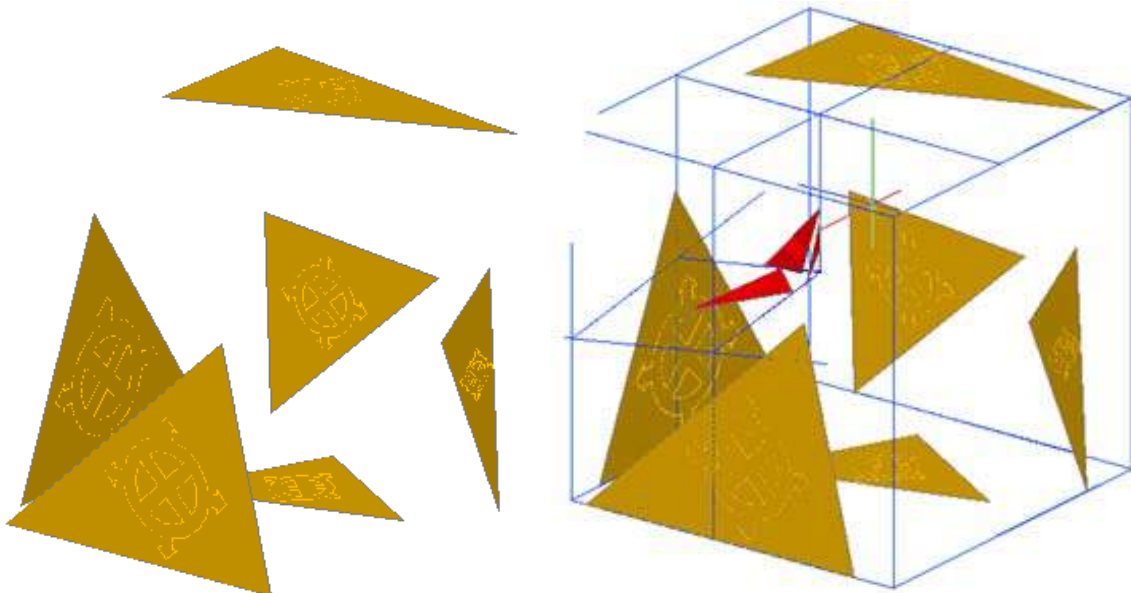


Repeat these steps until you completed the whole wireframe model:



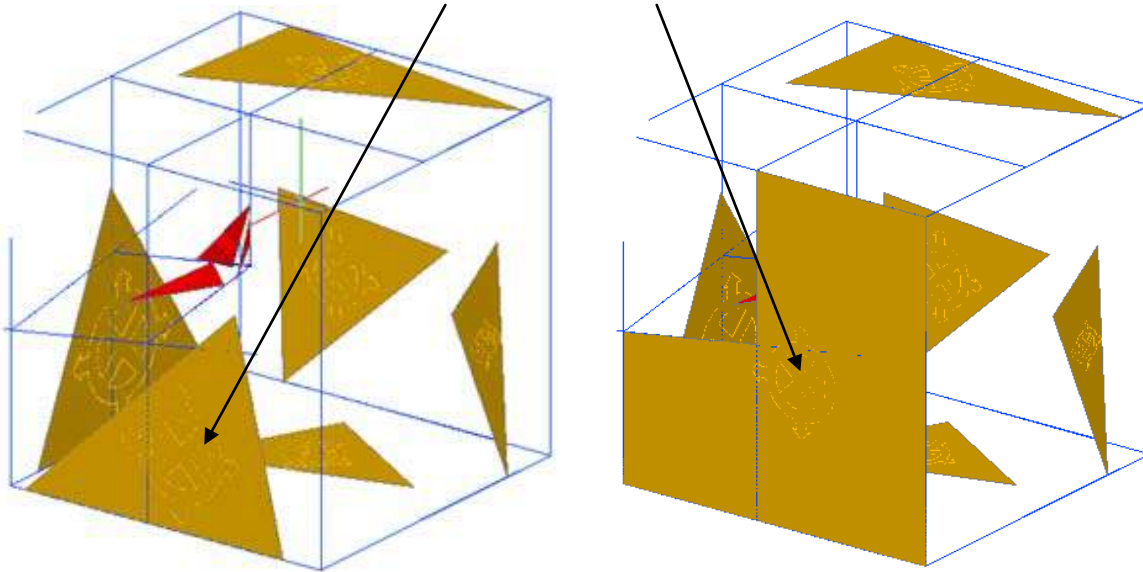
4.1.2 Surface model

To generate the surface model of an object you first create all planes you need by using the **→Modeling – Plane – Fit plane** command, like you would do for the wireframe. Then you intersect three planes at a time using the **→Modeling – Plane – Intersection lines (3 Planes)** command or **→Modeling – Plane – Intersection point (3+ Planes)**..



After having created all corners of the model you change the displayed boundaries of the planes to the true boundaries by using the **→Modeling – Plane – New Rim** command and snapping the corners:

The boundary of the plane is changed by snapping the corners (OSNAP = endpoint) of the plane to the corners of the triads.



If you do not only want to use the surface model in As-Built, you need to convert it to pure AutoCAD objects. You can use EXPLODE to convert the kubit planes into polylines. A polyline can easily be converted into planar surface (AutoCAD command `_PLANESURF`) or a region (AutoCAD command `REGION`).

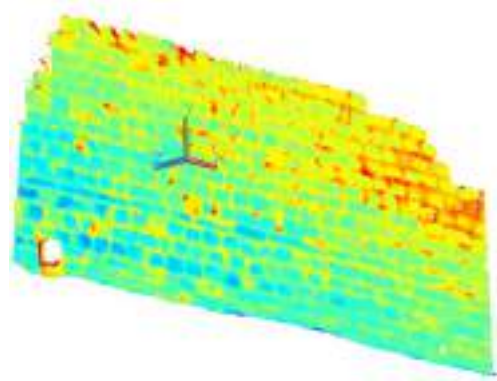
4.2 Flatness analysis – terrain and other surface models (volume computation)

As-Built for AutoCAD has a very powerful tool, with which regions of a point cloud can be analysed and modelled, with reference to a datum plane (e.g. walls, floors, excavations or **terrain models**).

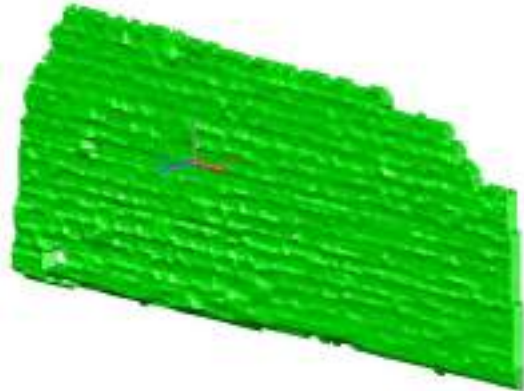
Deformations can be determined and displayed. It also allows you to create **AutoCAD solids** from those objects that deviate above or below the optimum plane and display their volumes. The AutoCAD command `_MASSPROP` can be used to **determine the volume** of the solids.



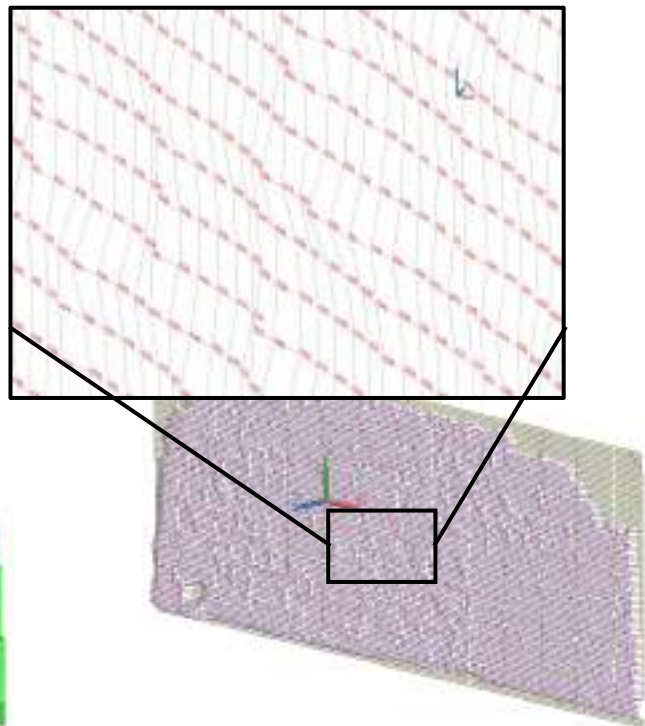
Original point cloud



Coloured in accordance with the deviations



Modelled AutoCAD solid



Variances displayed as 3D polyline grid

A detailed description of this analysis tool can be found in the Command Reference → **Modeling/construction – Analysis of planes in a point cloud.**

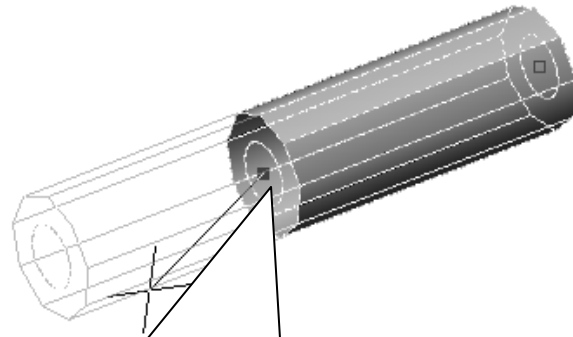
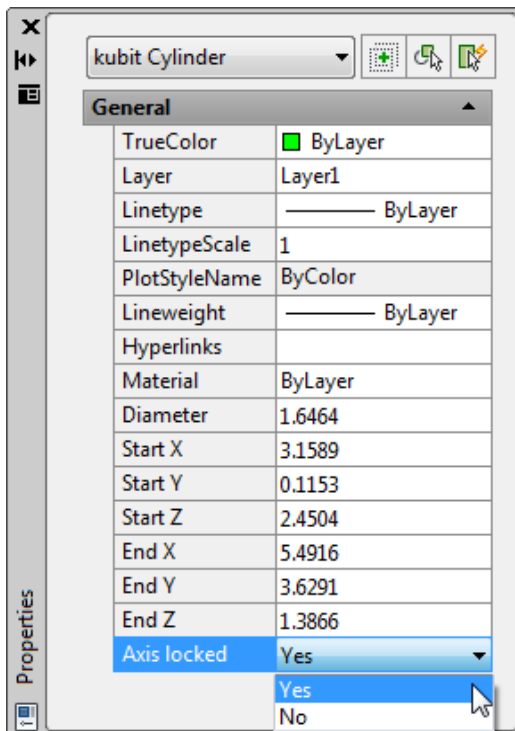
4.3 kubit Cylinders

Similar to the kubit planes the kubit cylinders are an object class which is also used in other FARO products.

Cylinders are created with the help of the commands → **Modeling – Cylinder – Draw cylinder** or → **Modeling – Cylinder – Fit cylinder using 2 click points**. With the latter command you may generate an average cylinder according to the method of Least Squares using points of point cloud.

Besides the geometrical properties like diameter and coordinates of the starting and end point, kubit cylinders do have an additional attribute, which states, whether the

cylinder axis is freely changeable or locked. This may be set within the AutoCAD properties window.



Every cylinder has a “grip” at each of its ends. By clicking on the grip with the mouse, you can change the length of the cylinder by “dragging”. However, the position of the cylinder centerline in space remains fixed.

For drawn cylinders the axis is by default unlocked, for best fit cylinders it is locked. The axis of a fitted cylinder is a valuable information gained from a point cloud and thus should usually not be changed.

You may use cylinders as surface objects when 3D drawing with the help of oriented images.

For modeling plants with a larger number of pipes and complex pipe runs we recommend to use the piping commands of the Plant ribbon tab.

There are different commands for editing kubit cylinders. These are described in more detail within the command reference.

4.4 kubit truncated cones

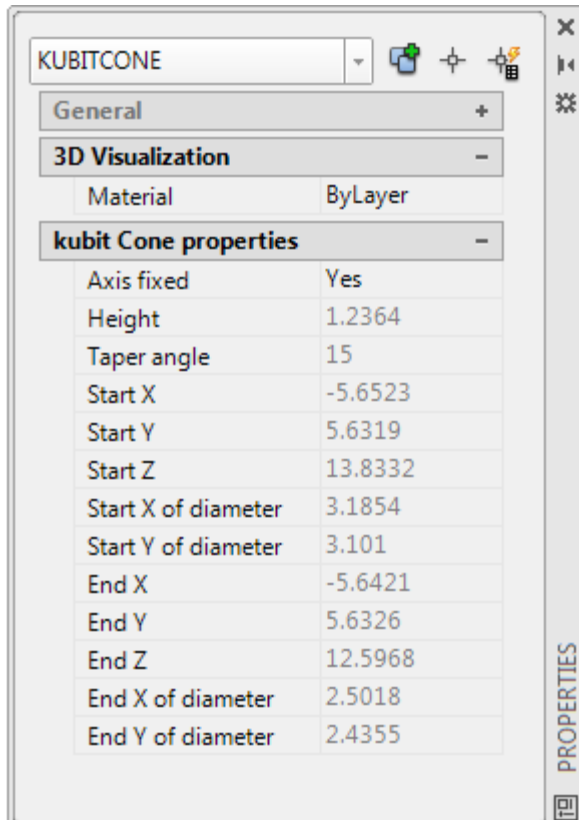
Similar to the kubit planes or the kubit cylinders kubit truncated cones are a specific object class introduced by FARO.

Truncated cones can be created on the basis of a circle or ellipse.

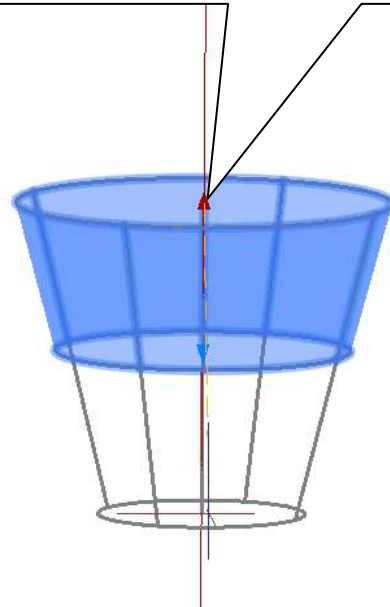
This is carried out by using the commands **→Modeling/Construction – Truncated cones – Create a truncated cone**, **→Modeling/Construction – Truncated cones – Fit a circular truncated cone** or **→Modeling/Construction – Truncated cones – Fit an elliptical truncated cone**. With both of the last two commands by snapping onto points in the point cloud, a fitted truncated cone can be created using the method of least squares.

Besides the geometrical properties like diameter and coordinates of the starting and end point, kubit truncated cones do have an additional attribute, which states,

whether the cones axis is freely changeable or locked. This may be set within the AutoCAD properties window.



Every truncated cone has a “grip” at each of its ends. By clicking on the grip with the mouse, you can change the length of the cone by “dragging”. However, the position of the cones centerline in space remains fixed.



For drawn truncated cones the axis is by default unlocked, for best fit cones it is locked. The axis of a fitted cones is a valuable information gained from a point cloud and thus should usually not be changed.

kubit Truncated cones can be exploded, using the AutoCAD command EXPLODE, to convert them into simple AutoCAD solids.

For a detailed description of the commands for creating truncated cones, please consult the As-Built for AutoCAD Command Reference.

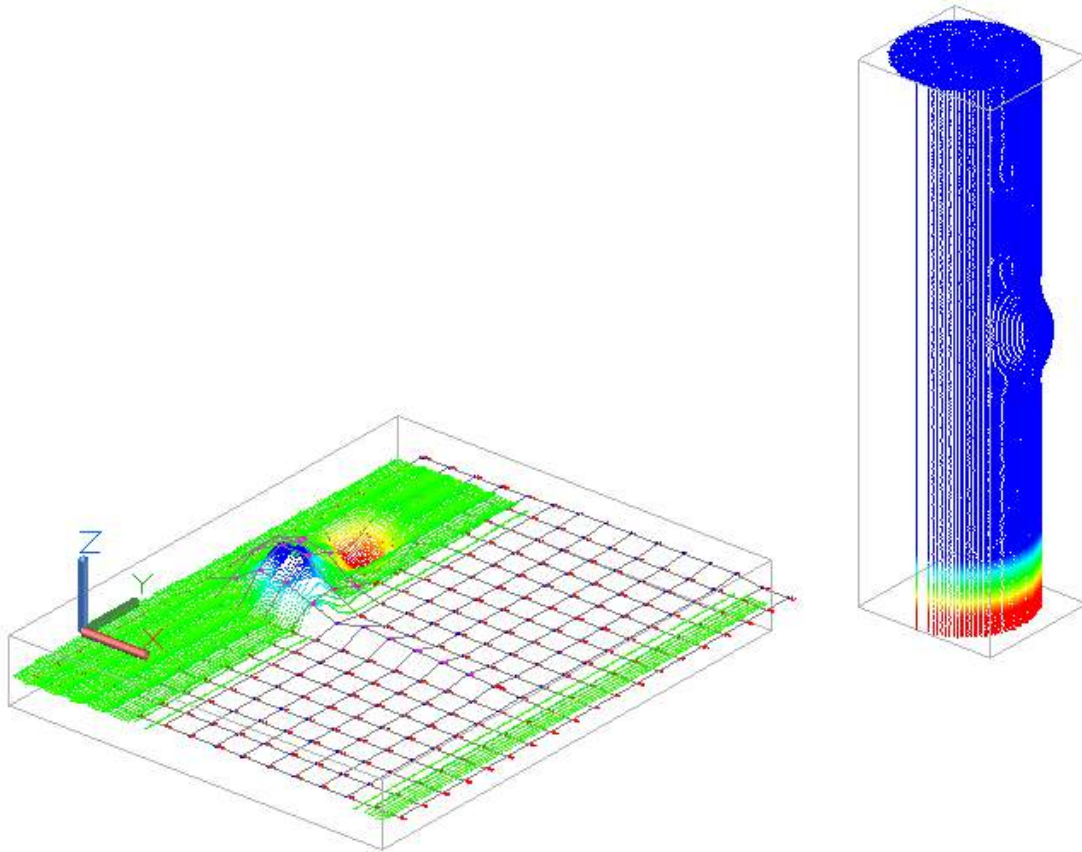
4.5 Cylinder analysis and analysis of frustums of cones

As-Built f provides a very powerful tool for analyzing cylindrical or conical shaped parts of a point clouds (e. g. tanks, towers, wells and the like) in various ways.

Deformations can be detected and visualized. **Volume calculation** based on the actual shape (including deformations and ‘deadwood’) of the object can be performed. It can also be used to simply create a nice **AutoCAD solid** of the object that deviates from the ideal cylinder or frustum of a cone.

In the command reference you will find a detailed description of the cylinder analysis tool (→**Analysis – Cylinder – Analysis of cylindrical shapes in a point cloud**

, →**Analysis – Analyze frustum of a cone in a point cloud**).



By unwrapping the cylindrical shaped part of a point cloud and subsequent use of the →**Ortho image** command you may create an image map of the cylindrical object.

5 Working with the Plant Tools

This chapter introduces you to the Plant features of As-Built for AutoCAD and takes you through some of the features. For a more thorough explanation of these features, please refer to the As-Built for AutoCAD **Command Reference**.

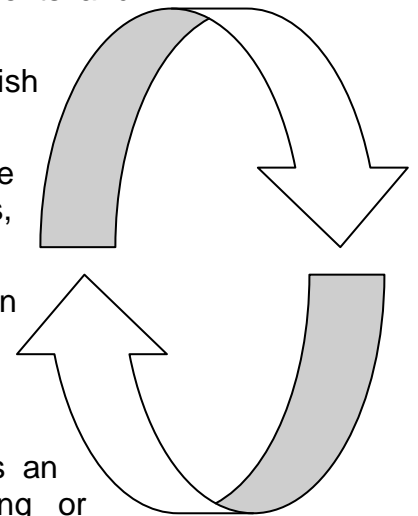
As-Built for AutoCAD allows you to use angle and shape detection to fit various components used within the plant environment. It has functionality to create a best-fit centerline with intelligence that can be exported to use with other Plant software. Using the new Walk The Run feature, a user can select the first pipe of the run (by selecting any two points on the pipe) and then use the available options to assist our software in detecting all components along that run.

The user has standard catalogs that can be imported/exported, as well as an easy interface to create and edit new/existing catalogs, as needed. Out-of-spec or uncommon items can easily be added by running a simple routine and then selecting a relative area in the point cloud data.

5.1 The As-Built Plant Workflow

To get from point cloud data to accurate and intelligent centerlines, we have created a straight-forward workflow; each step with its own purpose. To put it simply, the workflow consists mainly of these parts :

- Edit PSPM² – Importing/Exporting the correct components and settings for the current project.
- Walk The Run – Following each pipe run from start to finish to create intelligent plant objects.
- Adjust and Verify – Use our post-edit tools to set/change links between objects and make changes to existing runs, if needed.
- ApplyConstraints – After the components have been created for each pipe run, this step creates the best-fit centerlines and correctly adjusts different areas on the pipe run.
- Export – Exporting components in a pipe run creates an intelligent centerline, which can be used for routing or modeling in a desired plant product, if needed. Optional: Direct generation of pipes and components for special target systems such as AutoCAD Plant 3D or Advance Steel. To enable the correct export to special target systems, the corresponding catalogs must be imported from those systems before modeling.



We will get into these features in greater detail throughout the following chapters. For detailed information on all command options please refer to the command reference. We also recommend to go through the tutorials to get used to the process and its features.

² PSPM = PointSense Plant Model, formerly KPM

5.2 The Plant Model (PSPM)

The Plant Model (PSPM, formerly KPM) is a vital aspect of the piping functionality. It contains all patterns and dimensions for each component to be used in the drawing. You can load and edit a Plant Model using the command →**Pipes – Edit Plant**

Model .

We prepared some plant models for you. You find them in the installation folder of the FARO Box in the subdirectory \PSPM:

Americas_full_inch.txt is derived from catalogs with American standards.

ISO_full_mm.txt is derived from ISO catalogs and it is in millimeters.

ISO_selection_mm.txt is a selection derived from ISO catalogs.

StructuralPSPM.txt is the catalog for steel beams.

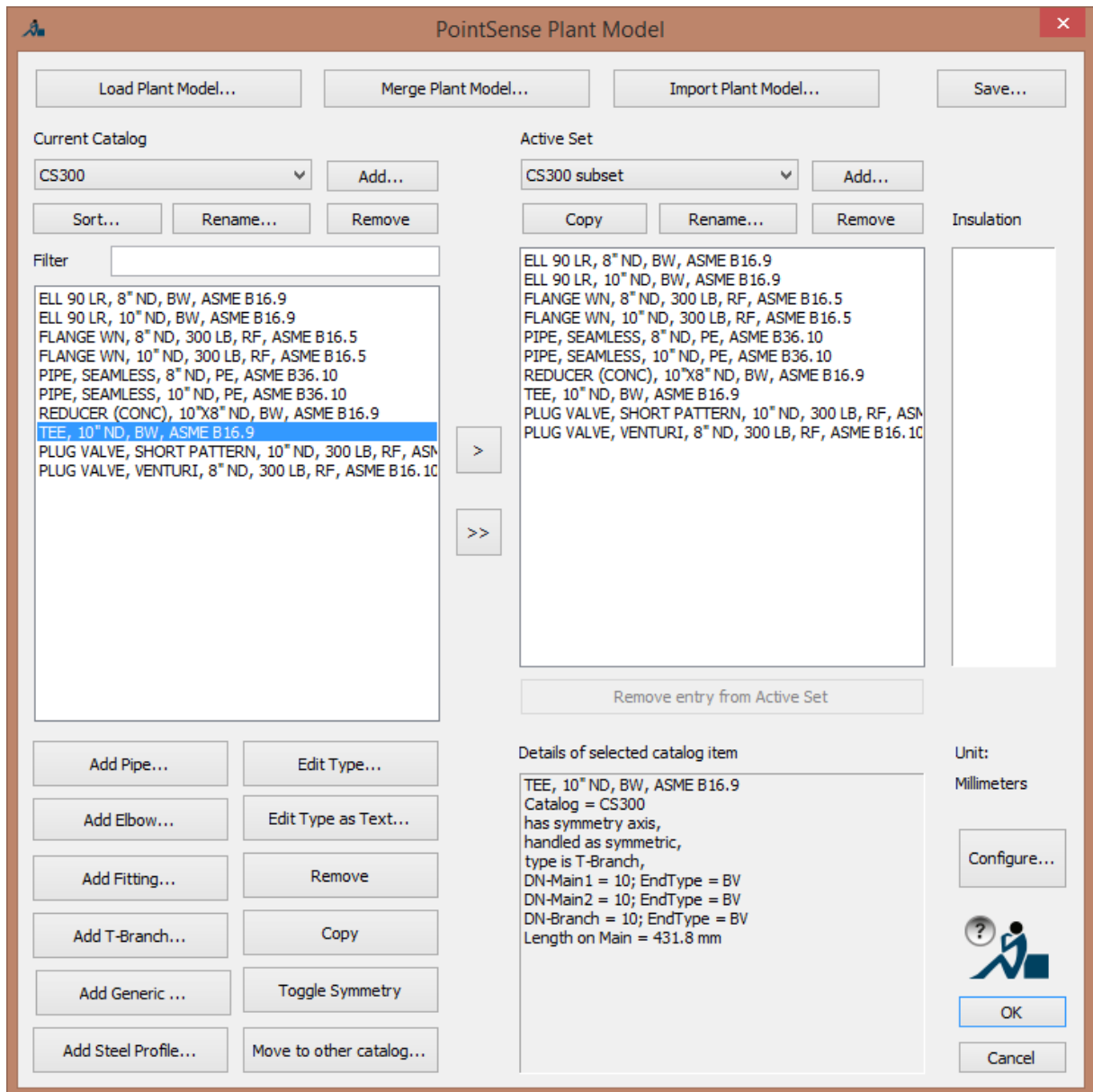
Try which plant model meets your project requirements and adjust it further if needed.

5.2.1 Overview of the Plant Model

The Plant Model data is divided into two sections: Catalogs and Sets. Catalogs (in the following also used as a synonym for pipe classes) contain all the available components, and the sets are derived from different component choices from these catalogs. The Active Set is the set of components currently being used by As-Built for AutoCAD. To adjust the Active Set, use the [**Remove Entry From Active Set**] and [**>>**] buttons.

5.2.2 Editing and adding items

In the Plant Model window, you will have buttons for adding components (i.e. **Add Pipe**). These buttons are to assist you in creating new patterns for Walk The Run to detect. Each class (Pipe, Elbow, Inline Fitting, and T-Branch) have their own unique characteristics, so be careful of which class you choose to add a new item.




Components can be added to the catalog/set by clicking the appropriate **[Add]** button. For the Measured Diameter and Nominal Size, you can manually enter the information, or you can use the **[From CAD]** button to enter these values from a selected section of the point cloud. Both options are very useful and should be utilized in the correct situations. When using the **[From CAD]** option, click any two points along a desired pipe. This will measure the diameter and enter that value into the appropriate parameter. Please note that for some components, other parameters can be automatically calculated by the **[From CAD]** option, including angles and lengths.

To edit items that already exist in a catalog, highlight that item and choose **[Edit Type]**. This will bring up the same dialog box as before, except with all the saved parameters for that component. Once you are done making changes, click **[OK]** to save the new parameters.

You also have the ability to add and save insulation thickness options to the PSPM. Simply type the insulation thicknesses in the dialog box and click **[OK]**. After that is completed, these insulation options will be available during Walk The Run.

For detailed information on all options, please refer to the command reference.

5.3 Configuration of Parameters

To begin the parameter wizard, select the As-Built command → **Settings** . This will display the first of four parameter windows.

The first window is to correctly configure the unit of the plant model (PSPM). Once you have both units set correctly, click **[NEXT]**.

The second window is to specify the noise and average distance between points, as well as setting the point cloud density. The noise and average distance can be calculated using the **[Auto Measure]** button. This feature will require the user to crop out a segment of pipe, in order to correctly calculate the parameters. It is very important to be precise in this cropping, so the parameters can be aligned with the current point cloud. It is also advised to manually verify these measurements, as best as possible.


'Point cloud density' is the value of detection used by the software. The higher the value, the better the detection and slower the performance. The reverse is true for a lower density value.

The third window is generally for experts who truly understand the algorithms and parameters being used, with the possible exception of Search region size. The larger the value, the further the software will attempt to detect components in the point cloud.

The fourth window is for experts too. Here you may change the default layer names for geometry created by As-Built and set thresholds for certain tolerance parameters.

5.4 Walk The Run

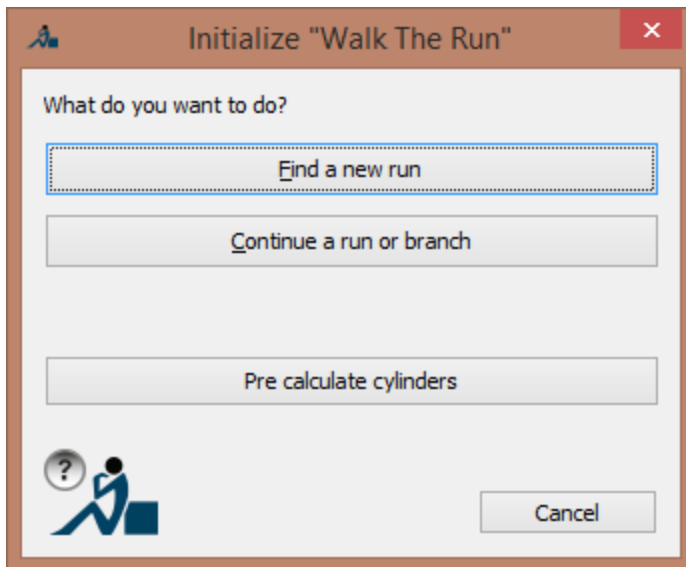
Walk The Run (WTR) automates component recognition and allows the user to step through the process of creating solids and centerlines throughout an entire pipe run.

To begin this process, select the As-Built command → **Pipes – Walk The Run** .

5.4.1 Understanding the WTR Interface

The WTR interface has two dialog boxes; the second one having two similar phases.

The first dialog box gives you the option to start a run or continue from an existing run.



The second dialog box is the main interface for the Walk The Run process. It has two phases that appear nearly identical. The first phase is constantly updating the detected fittings while the software is searching for component matches. The second phase occurs once the software is finished searching.

The main dialog box has all options for continuing through the WTR process, as well as buttons to Undo, Pan/Zoom, etc. The list box in the center shows all available component options at any time in the run. Items in this section can be highlighted or double-clicked to insert them into the drawing. You can find more on this in the Command Reference. In addition we recommend that you work through the tutorials at least once.

5.4.2 Probability and Verification

The entries in the top component list are sorted in accordance with their probability to be a match. The black entries in the lower list would fit on the current free connection, but there was no continuation pipes for it recognized in the point cloud.

The probability value is determined by a few different criteria, including its diameter, connection point, and angle to the previous and possible next components.

In many cases, the highest probability is the best choice to insert, however, it is always recommended that the user visually verifies this component to be the correct choice. For better overview the highlighted component is temporarily inserted into the drawing, in order to see the item in its rotation and placement. If the end of the pipe run is not displayed in the screen section you may use the [Pan Zoom] button to navigate to it.

5.4.3 Proceeding Through the Run

Once a user starts Walk The Run, there are a few different ways to proceed through the process. The first choice is starting a new run or continuing from an existing run. Both options have their differences and similarities.

Starting a New Run

To begin a new run, choose [**Find a new run**]. This prompts the user to click two points along a pipe to start the run. The user can select any two points on the pipe, as long as the points are picked in order of flow direction. For example, if the user wishes to follow the run to the left, they should choose the two points from right to left.

Continuing a Run

To continue from an existing run, choose [**Continue a run or branch (Select existing connection point)**]. The user will then be prompted to select an existing plant object. If the object has only one open connection point, the run will continue from that point. For example, if a tee only has the branch connection open, the software will understand that point as the most obvious continuation point. If there is more than one open connection point, the user will be prompted to click any point near the desired continuation point.

The other thing to note on continuing a run is that the new piping components will be automatically linked to the component that was initially selected. In the case that a user wants to link objects from multiple new runs (using **Start a new run**), they can use the **Set Link** (→Pipes – Set link) routine to manually perform the action.

Changing Insulation

At any point during Walk The Run, you can change the current insulation thickness. By default, this value is at 0 (no insulation). There are two places where a user can change the insulation thickness: the dialog box for Walk The Run or the command line (by typing I and pressing ENTER) during any Walk The Run command. When the insulation thickness is a value besides 0, the measured diameter will take this value into account before searching for detected components. Please note that the insulation is not modeled at this point, however, you will see a wireframe version of the insulation around the components, in order to accurately see the true diameter of the data. After you are finished with the line, there are ways to display the insulation as a 3D object.

Extend Pipe to Point

If the last inserted component during WTR was a pipe, the user has the option to [**Extend pipe to point**]. This option prompts the user to select a point along the pipe to extend towards. There are two toggle options that control the behavior of this:

- **Grow** – This option will attempt to extend the pipe as far as it can detect it in the cloud, based on the user's selected point. If this option is turned off (noGrow), the routine will only extend the pipe to the exact selected point.
- **Fit** – If fit is turned on, the pipe will try to re-align itself with the point cloud on every new point it is extended to. In the case of longer pipes, having this turned off (noFit) may result in pipes straying away from the point cloud, as they will simply extend on their originally-detected axis.

After the pipe is finished extending, it will attempt to detect components from its new connection point. It will display these components in the Walk The Run. It is also

useful to note that the user can usually extend the pipe backwards, in the case that they skipped over components or strayed too far from the cloud.

Specify Other Pipe

A very powerful option during WTR is called **Specify other pipe**. This feature prompts the user to click two points on any other pipe on run and will attempt to detect which components are located in-between the selected pipe and the last-inserted component. For example, if the last-inserted component is a pipe, the user can use this option and then select two points on the next pipe, in order to insert an elbow that is in-between them. This option can save a lot of time, as well as allow the user to detect branches while staying on the current run.

When using this option, many times the new pipe will draw as a small segment that is not connected to the component before it. This visual effect will be corrected during **ApplyConstraints**.

Please note that during insertion of certain components (i.e. reducers, flanges, valves, etc.) you will be prompted where to place this fitting. At this time, the fitting will follow your cursor and you can click to insert the component at a specific location. You also have the option of using the automatically detected location by simply hitting ESC. The manual-placement method is very effective when used with the planar view (run VirtuSurv³).

Manual Insertion of Components

There are certain cases where a user will need to manually insert a component. This could happen if the component does not exist in the point cloud or it isn't being detected, among other reasons. To perform this insertion method, simply highlight any item with black letters in the suitable item list. For each component type, the next prompt will be similar with a few differences. The user can either manually enter values or choose to select the values from points in the point cloud.

5.4.4 Pre-calculate cylinders

The third button in the window "Walk The Run" is the button "Pre-compute cylinders". With the help of pre-computation of cylinders, pipes and cylinders can be automatically computed in the current part of a point cloud. The generated cylinders are placed on a layer, whose name is defined under

Settings > Layer > Pre-computed cylinder

If the layer does not exist, it is created and is not visible. The pre-computed cylinders can be used by "Walk the run" so that it can work faster.

A computation that is currently being carried out can be interrupted by pressing the Esc key. If the pre-computation is started again, an interrupted computation can in the same session be taken up again and continued.

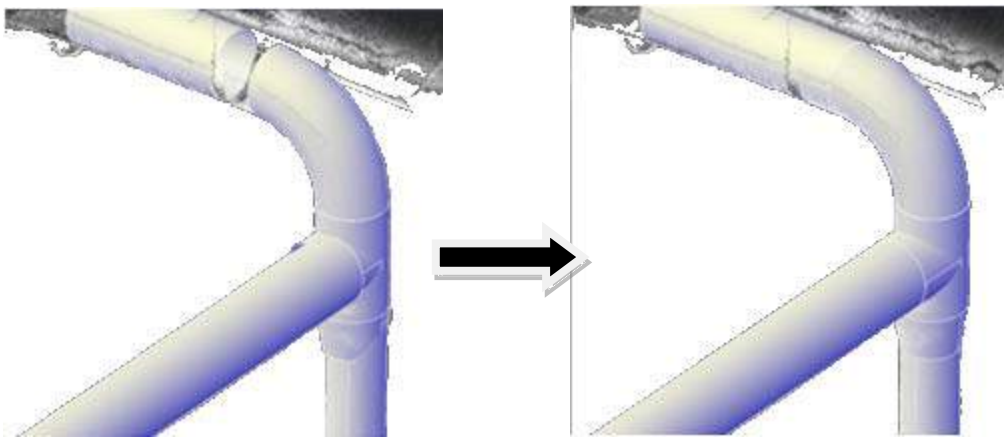
³VirtuSurv is FARO's software for displaying scan data in a planar view and is included in the As-Built Modeler license.

5.5 ApplyConstraints

In As-Built for AutoCAD the established term “ApplyConstraints” means a process for automated correction of plant object positions. During Walk The Run, many of the components will not connect perfectly. They may have crooked connections, overlap each other, etc. ApplyConstraints reads all attached components, determines the best fit scenario, and then applies it.

5.5.1 Why ApplyConstraints?

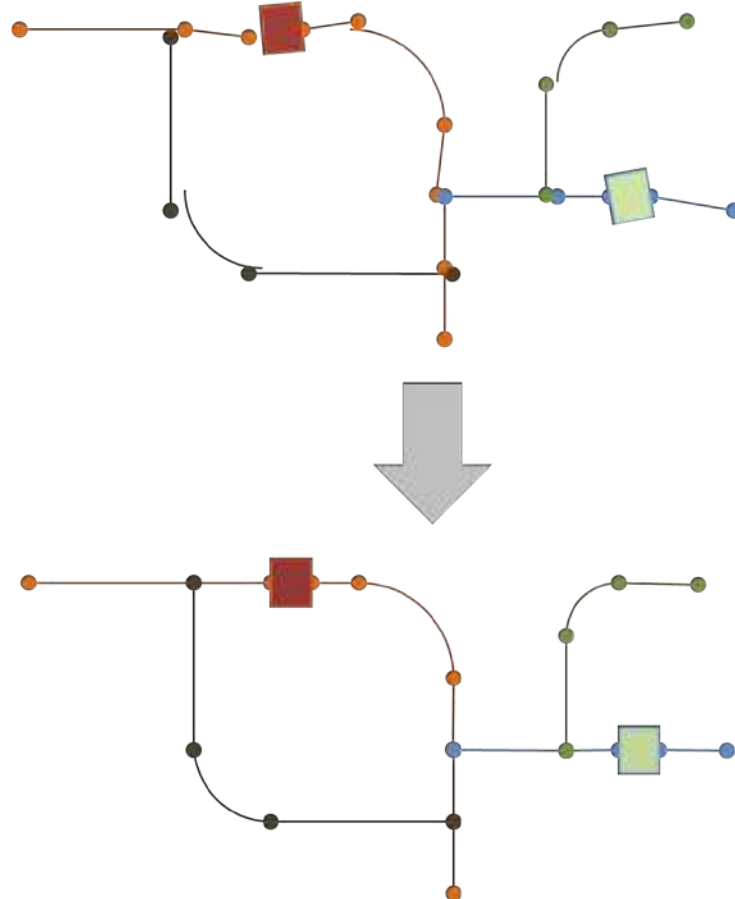
When dealing with real-world data in point clouds, it is important to remember that components will never have perfect dimensions.



Pipes and components can bend and fluctuate, elbows could be mitered, point cloud data could be slightly off, etc. For these reasons, the post processing command ApplyConstraints is necessary. The result from applying constraints is:

- to keep the components as close to the as-built data (point cloud) as possible
- to fulfill the constraints of a well defined plant construction (coaxiality, parallelism etc.)

The following figure illustrates ApplyConstraints adjusting the original fit to the final result.



In most cases, the function should be performed once the user is finished with the pipe run, in order to account for all components when determining the best fit.

5.5.2 Using ApplyConstraints

To use ApplyConstraints, select the command **→Pipes – ApplyConstraints**. Once the dialog box appears, click **Select Plant Objects** to continue. The user will then be prompted to select an object from the drawing and press ENTER. All objects that are connected to the selected component will be highlighted along the run. At this point, the user can verify that all objects are linked correctly. In the lower-left corner of the dialog box, it will show how many plant objects are connected to the selected object. This is how many components are being used to determine the best fit scenario. Click the **[Start ApplyConstraints]** button to finish the process. All connected components will be re-aligned to their new locations and angles.

Although the process is capable of correctly adjusting most of the pipe runs of practical use, it may happen on rare occasions or with uncommon sequences of plant objects, that the automatic process does not manage to fulfill all of the constraints. For this reason, you have the choice to activate the checkbox „*highlight violated constraints with colored lines*“ in the dialog. Hence, in the results of ApplyConstraints, three problem classes are being identified and all of the occurrences are going to be visualized by lines of the colors green, red and blue.

Problem Class	Line Color	Graphical Representation
Significant movement at the end of a pipe axis	green	a single line is drawn perpendicular to the resulting axis for showing the track of the movement

Two connection points are not coincident	red	two lines represent the directional vectors of the connection points, a third one is connecting the coordinates of the connection points
Two connection points are coincident but not collinear	blue	two lines represent the directional vectors of the connection points

Regarding the first problem class, you are able to configure yourself which amount of axis movement you want to be considered as „significant“ change. You may use the command **→Prepare – Settings** to configure the corresponding value on page 4 of that dialog.

5.5.3 Setting Links Between Objects

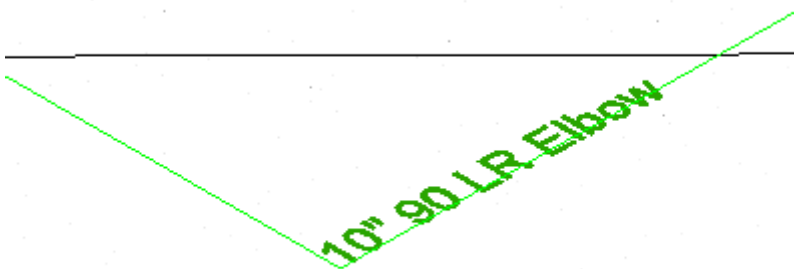
Objects need to be linked, in order for ApplyConstraints to work correctly. These links are automatically performed during Walk The Run, as well as when continuing a run from an existing object. In the case that objects are not linked, the user can use the command **→Pipes – Set link**. The following command-line prompt will ask to connect or disconnect objects and then ask the user to select the two plant objects to perform the action on.

During a Connect process, if there is only one open connection point on an object, the software will automatically choose that as the point to connect to the other object. If there is more than one open connection point, the software will determine the connection points based on proximity to the other object or by explicit selection.

5.6 Exporting

One of the main goals for this functionality is to create intelligent centerlines which can be used in other piping products to create a finished deliverable. To do this, use the command **→As-Built Plant – Export** and then select any pipe along the run or export all runs in the drawing. The routine will place a message on the command line displaying how many objects were created and the matching number of vertices. Each component along the run has the following information:



- Nominal Size
- Component Name
- Correct Start/End Points
- Insulation Thickness, if applicable




More opportunities are the exports as tagging objects or 3D solids. If you work in Plant 3D you can also create Plant 3D objects.

5.7 Troubleshooting

There are some common issues that a user may come across while using the functionality of As-Built for AutoCAD. This guide will provide the most common solutions for these issues.

- “No Pipe Found” dialog box – This usually occurs when a user chooses two points along an object that is not a pipe. If the component is correctly a pipe, then the cause could possibly be related to the parameters. Use the command **→Prepare – Settings**  to adjust the values or perform Auto Measure.
- “No Suitable Pipe Defined” dialog box – If your current Set (defined in the PSPM) does not include the currently-detected pipe size, this message will appear. To resolve this issue, add the correct pipe size(s) to the current set by using the command **→Edit Plant Model** .
- Objects are not adjusting during **ApplyConstraints** – There are a few reasons why this could happen, however, the most common reason is the objects are either linked incorrectly or not linked at all. To resolve this, try using the command **→Pipes – Set link** to connect the two objects that are not adjusting correctly. Also, please note the number of plant objects detected during the **ApplyConstraints** routine. If the number is too low, it is most likely a missing link.
- A much smaller pipe draws inside of the desired pipe – This issue is caused when parameters are not set correctly; usually the Average Point Distance (a) value. Use the command **→Prepare – Settings** to adjust the values or perform Auto Measure.
- Pipes draw too far or not far enough - This issue is caused when parameters are not set correctly; usually the Noise (n) value. Use the command **→Prepare – Settings** to adjust the values or perform Auto Measure. Higher Noise values cause the pipes to draw too far; vice-versa for lower Noise values.

5.8 Tutorials for pipelines, steel beams and more

There are a number of tutorials for a quick start into working with the As-Built Plant tools. You can access them via the Getting Started command  on the As-Built ribbon tabs. It provides links to the tutorials.

To complete all Plant tutorials a dataset consisting of a point cloud and drawings has to be downloaded. The tutorial videos lead you step by step through the various workflows.

Tutorial: Prepare a Drawing

After this tutorial you will be able to:

- Load catalogs (Plant Models)
- Add steel to an Active Set

- Configure units and parameters
- Insert a point cloud (RCP file)

Tutorial: Pipes

After this tutorial you will be able to:

- Use scan navigation
- Create pipe runs efficiently (Walk The Run)
- Apply Constraints
- Export the results
- Create an ISO drawing

Tutorial: Steel

After this tutorial you will be able to:

- Add steel to an Active Set
- Fit steel beams to the point cloud
- Copy beams
- Extend beams

Tutorial: Tie-in Points

After this tutorial you will be able to:

- Create and annotate tie-in points

Tutorial: Tank Analysis

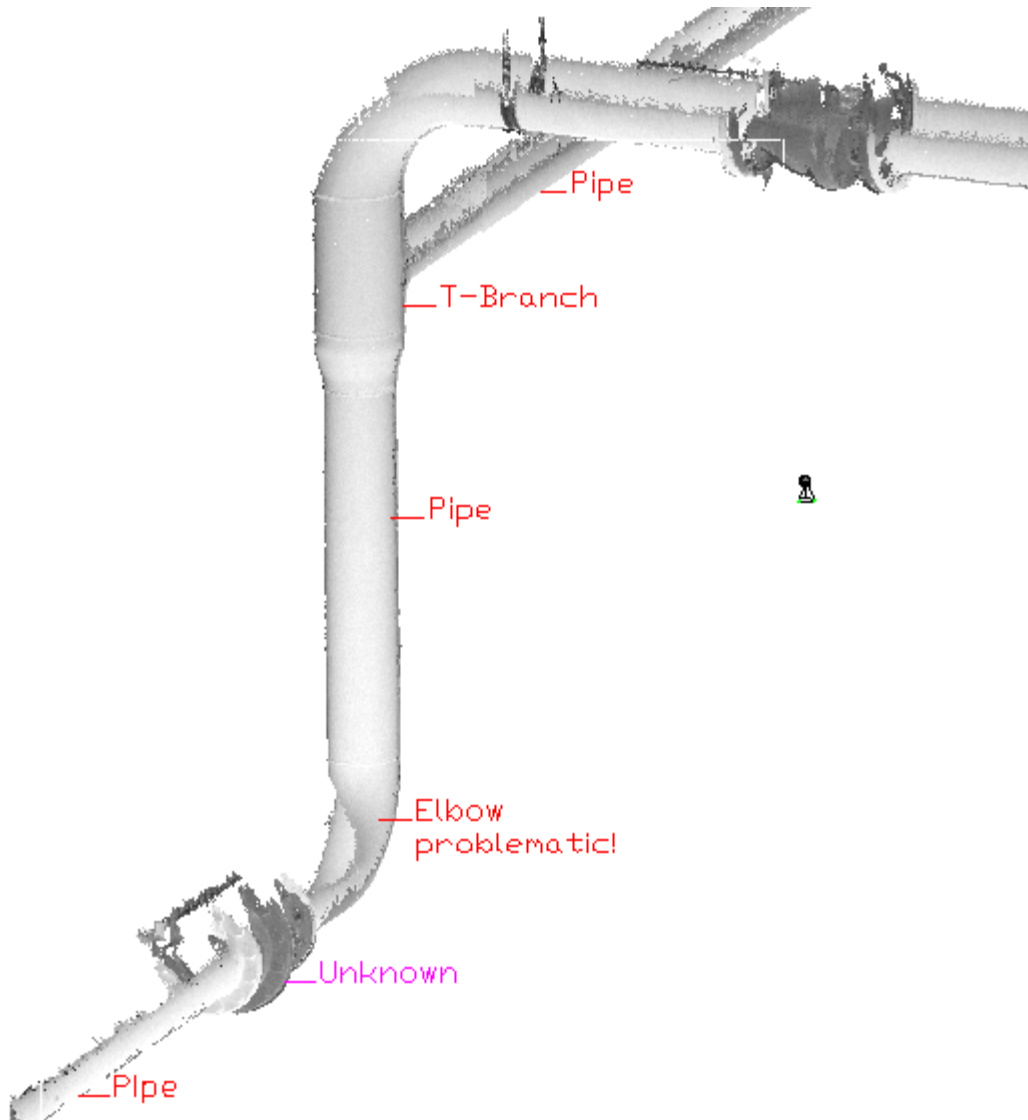
After this tutorial you will be able to:

- Define point cloud sections
- Fit a cylinder
- Run a tank analysis
- Import the result to Excel

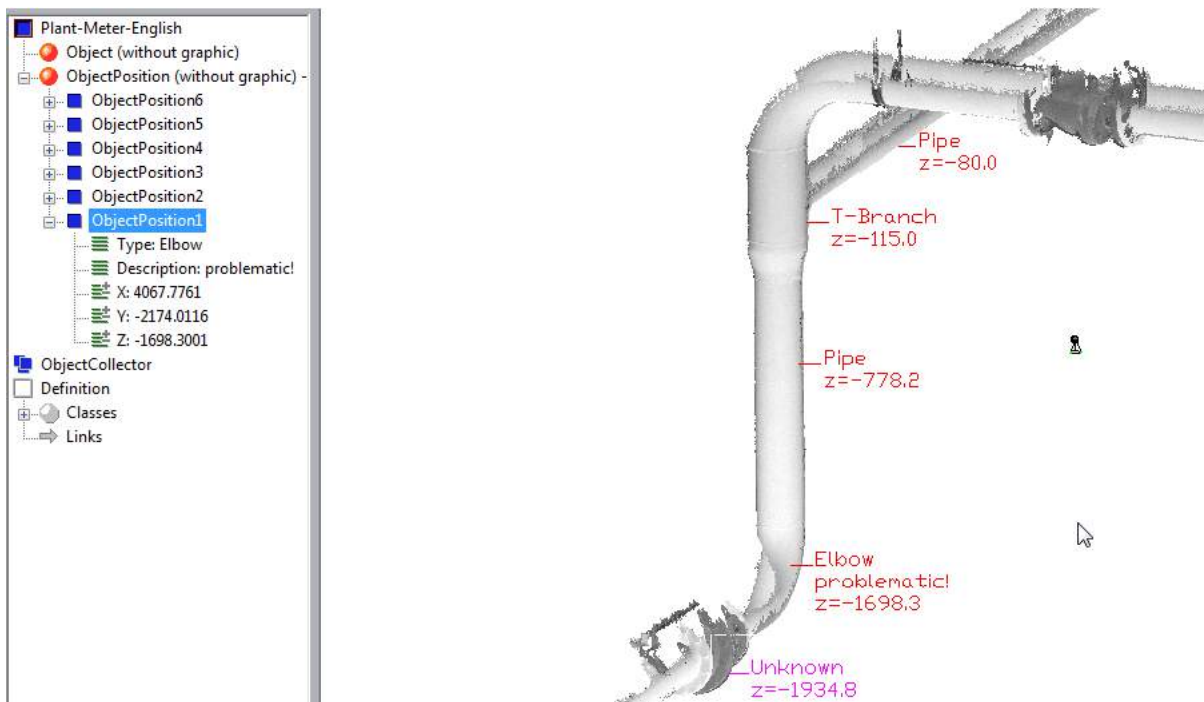
6 Tagging in 3D space

Often, certain parts of the point cloud are to be marked with text marks. These texts should be readable from any direction. This form of tagging is possible using the feature data management tools.

"Labels" serve the visualization of additional (non-geometric) data the user wants to add to the point cloud.



Parallel with the tagging in 3D space the technical data and attributes are displayed in a clearly arranged tree structure. The content of the „Labels“ can be controlled via the tree structure.

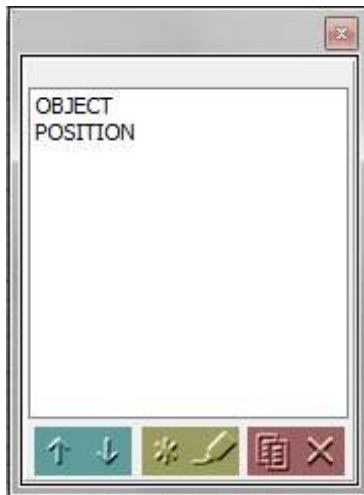


The attributes (rows of text in a Label) are completely free definable and they are typified in a way that supports their use in databases. You can define numbers, text, lists or links to external files as attribute. You find the complete documentation of this extremely powerful structure definition of attributes in the PointSense command reference.

To create tags use the “Fast Capture” button from the “Feature Data” ribbon of As-Built for AutoCAD (HYLASFASTREC). This opens the tree structure and provides some predefined commands to place tags.

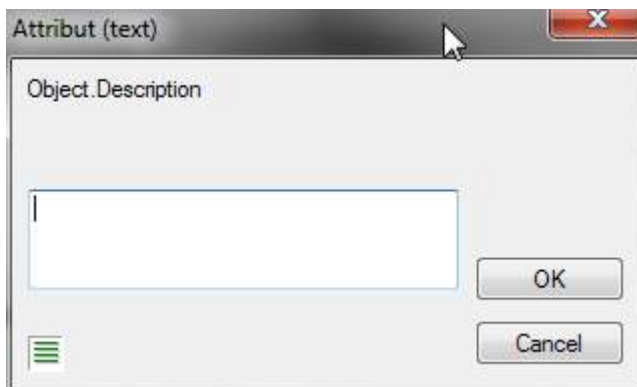
When running the command HYLASFASTREC in a drawing that has never been tagged with As-Built for AutoCAD before, you are informed that you have to load a structure definition first. Close the message with OK and the “Load structure definition” dialog opens. Click on the “Select directory...” button and select the \Plant directory under \Templates. Then simply select one of the files in the “Load Structure Definition” dialog box that are offered in the list on the left and confirm with OK. Ensure you select a definition that fits the units of your drawing otherwise the tags might be too small or too large.

Now the structure tree opens on the left. Its functionality is explained in the [Feature Data Management manual](#). Besides the structure also the box with two predefined tagging commands opens. Note: In a drawing that was tagged with As-Built for AutoCAD before this box opens directly.



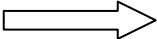
The box contains some predefined tagging commands. You run them by double clicking the according entry. Then just click an insertion point for the Label (tag) as you are prompted in the command line.

You will be prompted to fill in the description attribute. You can enter any text you like to tag your drawing with. Confirm your entry with OK.

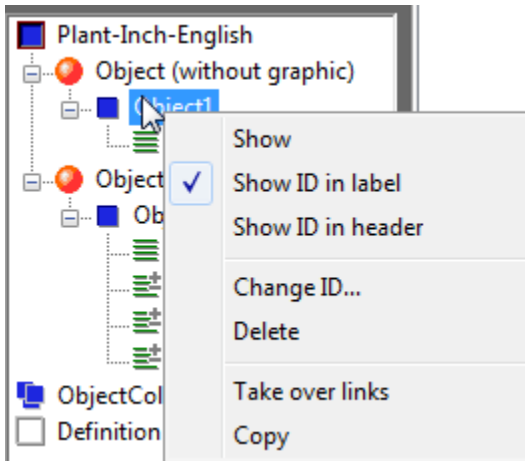


As-Built for AutoCAD then places the Label at the previously clicked position. And depending on which predefined command you chose creates either an object or an object position entry in the structure tree.

You have various possibilities to change the appearance and content of the Labels by using the context commands of the structure view. You may for instance hide the object ID in the label and display only the description attribute.

— **Object1**
any description text you like  — **any description text you like**


Just click on the Label. The corresponding entry will open in the structure view. There you right click on the object and in the context menu you un-check “Show ID in label”



For more detailed information please refer to the [Feature Data Management manual](#).

7 Tutorial – Generating a floor plan

This tutorial demonstrates the typical workflow of creating a building floor plan from a point cloud.


The data and video for the tutorial can be accessed via the Getting Started command  on the As-Built ribbon tabs. This provides links to the various tutorials.

Download the data. Unpack the zip archive. It contains drawing files (DWG) as well as point cloud data. Follow the procedure shown in the video step by step.

8 Area management – Introductory tutorial

By means of an easy example this chapter inducts you to the work with the area management (feature data management). Only the basic principles will be explained here. A detailed description of all commands can be found in the [manual for the Feature Data Management](#), which is part of this installation.

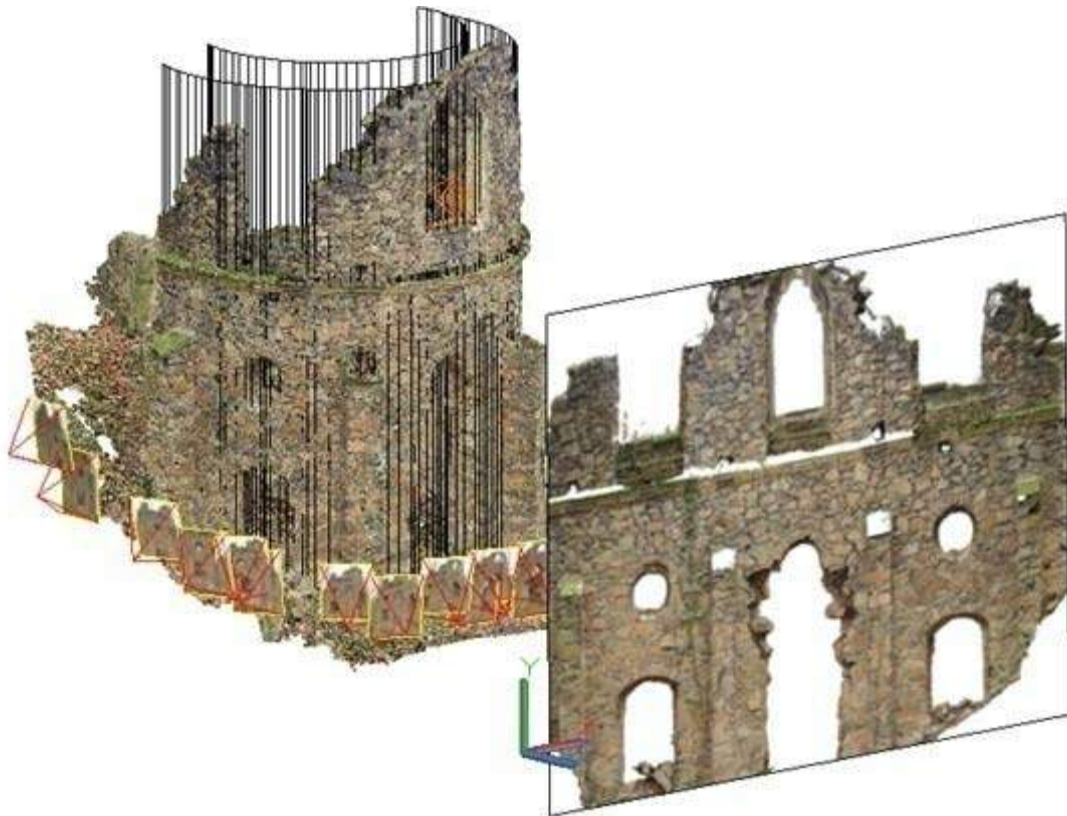
We suggest that you follow the tutorial step by step in order to get a first impression of the possibilities which are offered by the feature data management tool.

The video with a step by step description for the tutorial can be accessed via the Getting Started command  on the As-Built ribbon tabs. This provides links to the various tutorials. The tutorial drawing can be opened using the Area Data Tutorial command in the As-Built Feature Data ribbon tab.

9 Working with images

As-Built for AutoCAD allows a fundamentally new way of working by means of combined evaluation of oriented images and point clouds. The software provides extensive tools for photogrammetric evaluation. They enable the true-to-scale rectification of digital photographs, maps and plans as well as their subsequent interpretation and evaluation. Facade photographs, pictures of wall and ceiling paintings or floors can be processed, but also old maps and plan material can be brought back into a true-to-scale form. The results are true-to-scale image plans that combine photographic condition documentation with precise geometric information.

Other tools allow you to orientate photos by means of control points and to use these oriented photos for both 2D and 3D image evaluation. Photos of non-planar facades can be processed into true to length image plans. This creates three-dimensional models or 2D image plans.



Further information about the photogrammetry tools of As-Built for AutoCAD can be found in the [As-Built Photo manual](#), which is part of your installation. You may find a link on the As-Built Photo ribbon tab under “Help”.

10 As-Built for AutoCAD Command Reference

In the command reference all commands are listed in a similar order as in the As-Built for AutoCAD ribbon. The description of each command is structured as follows.

Ribbon text		
Button	Command Name	
	Ribbon tab	

Ribbon text

Under this title you will find the command in the As-Built for AutoCAD ribbon.

Button

Via the symbol given here you may call up the command in the toolboxes and ribbons of As-Built for AutoCAD.

Command Name


Here you find the command name as it should be entered in the command line of AutoCAD.

Ribbon tab

Specifies on which of the various ribbon tabs of As-Built for AutoCAD the command can be found.

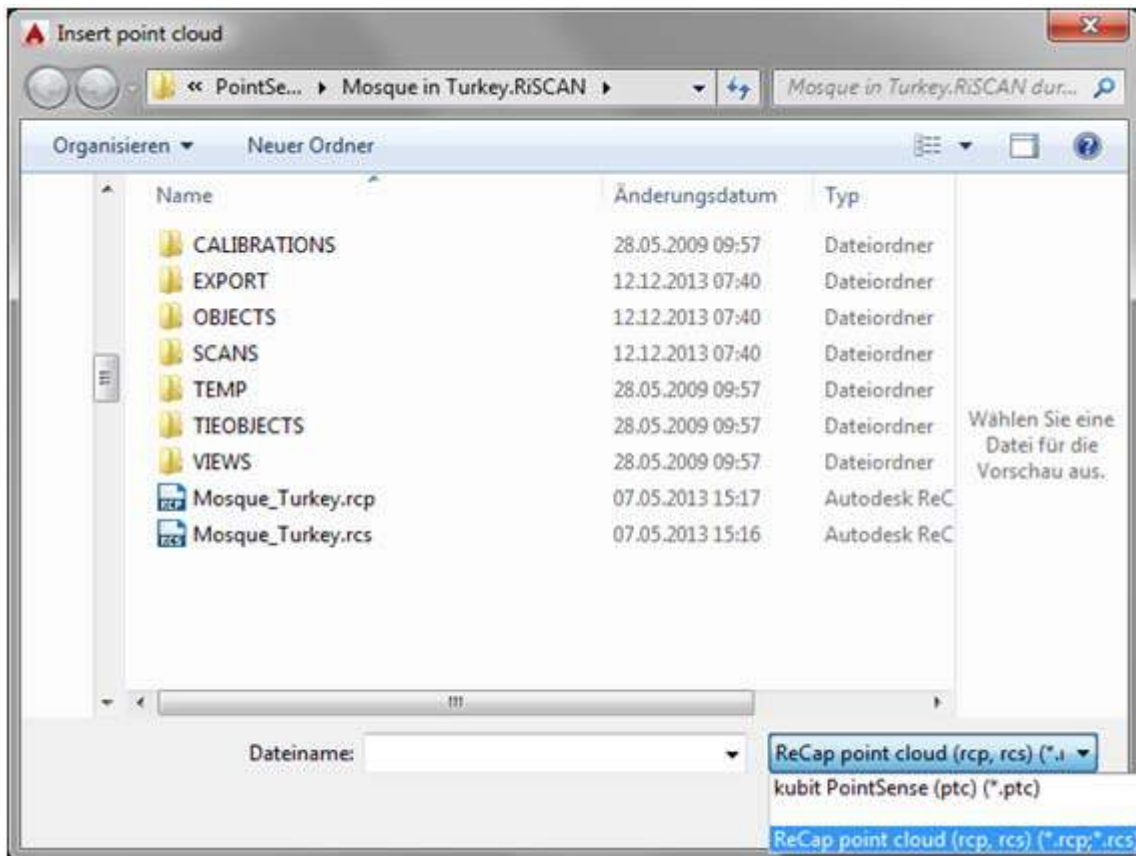
10.1 As-Built Modeler Tools

Point clouds – Insert point cloud

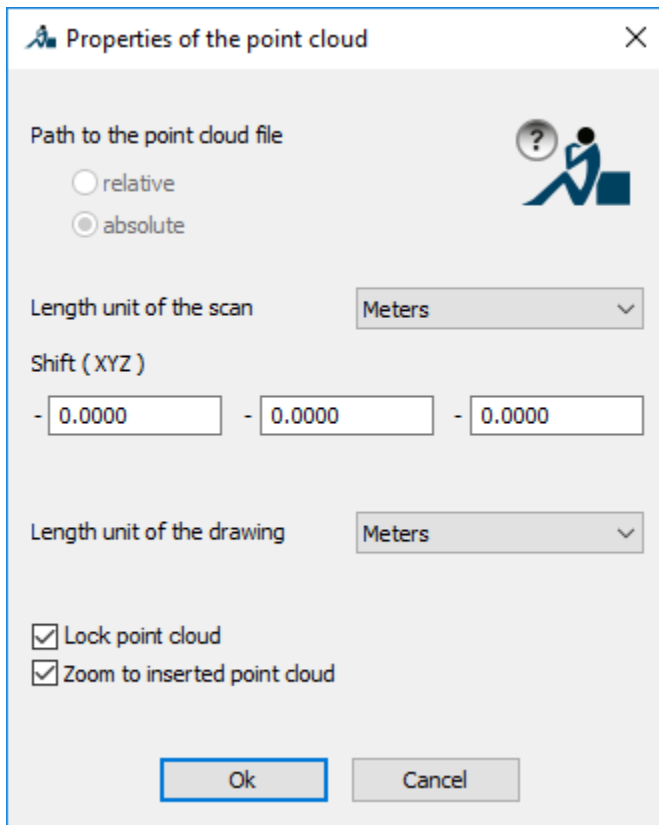
	Command Name:	PCLOAD
	Ribbon Tab:	As-Built Modeler

With this command you can insert one or several point clouds into your current AutoCAD drawing via a FARO PTC file or via an Autodesk point cloud file.

The following window appears. Select one or several (SHIFT or CTRL + left mouse button) point cloud files.



As second step following dialog opens:



Here you define if you want to reference the point cloud in the drawing by relative or by absolute path. A reference with relative path is only possible when the drawing has previously been saved and if point cloud and DWG are stored on the same drive.

When creating point cloud objects by importing from other formats the unit of the object is the same as in the original scan file. If PTC files are exported by a scanner software package one can usually select a different unit. When working with multiple point clouds and when importing into a drawing that already contains geometry one should ensure the units fit together. To ensure that the inserted point clouds are automatically scaled to the length unit of the drawing, you may state the unit of the scan and the unit of the drawing here.

Under Shift (XYZ) you may enter an offset in X, Y and Z. The values are automatically subtracted from the original coordinates. With this option you may move point clouds with large coordinates (e.g. geodetic survey coordinates) closer to the origin, because AutoCAD sometimes does have problems when drawing or modeling at large coordinates. You should write down the offset values in case you have to deliver your drawings with the original coordinates, so you can later move the geometry back that location.

The option „Lock point cloud“ prevents that you accidentally move or delete the point cloud. Unfortunately locking a point cloud also prevents selection via „Quick Select“. You may at any time lock or unlock the point cloud via the AutoCAD properties window.

If you want the drawing to be automatically zoomed to the newly inserted point cloud(s), you just enable the according option.

After loading, the point cloud will be set active and the Section Manager will be displayed as a dockable palette window.

Hint: By means of the AutoCAD command “LIST” you may get information about point clouds you have selected (e.g. number of points, coordinates of the bounding box ...).

Different behavior when inserting one or multiple PTC point clouds:

a) One point cloud object was selected

The point cloud object is inserted and loaded.

The point cloud is set to active. In the case that another active point cloud already exists in the drawing (green marks) you will be asked to set the newly loaded point cloud to active.


Following the loading, all points of the point cloud will be drawn into the current AutoCAD drawing.

b) Several point clouds were selected

Point clouds are inserted into the drawing but *not* loaded. You only see the grip and - depending on the current setting - the bounding box.

You may load a point cloud with the command →**Point clouds – PTC format - Load point cloud.**

Point clouds – Set current point cloud

	Command Name:	PCSETCUR
	Ribbon Tab:	As-Built Modeler

If more than one point cloud was imported in AutoCAD a point cloud set to active can be selected using this command. Many commands require the definition of an active point cloud.

Command: **PCSETCUR**

Select point cloud: *(Select point cloud to set active)*

You can use all of the following commands with the current point cloud. If you want to use a command with another point cloud you need to select the point cloud as the current active point cloud.

If the AutoCAD drawing has only one point cloud, this will be the active point cloud by default. If the drawing has several point clouds you need to select the point cloud you want to set active.

With this command you can set Autodesk point clouds as well as FARO point clouds to be the current (or active) one. Hence you can work with different format point clouds in one drawing. All commands always apply to the current (or active) point cloud. Though, we recommend that you decide for one point cloud format depending on the AutoCAD version you are using.


Hint for FARO point clouds (PTC)

The active point cloud (PTC) is represented by green marks at the edges of the bounding box and at the grip.

The following table shows the point cloud statuses and their corresponding color to mark the point cloud.

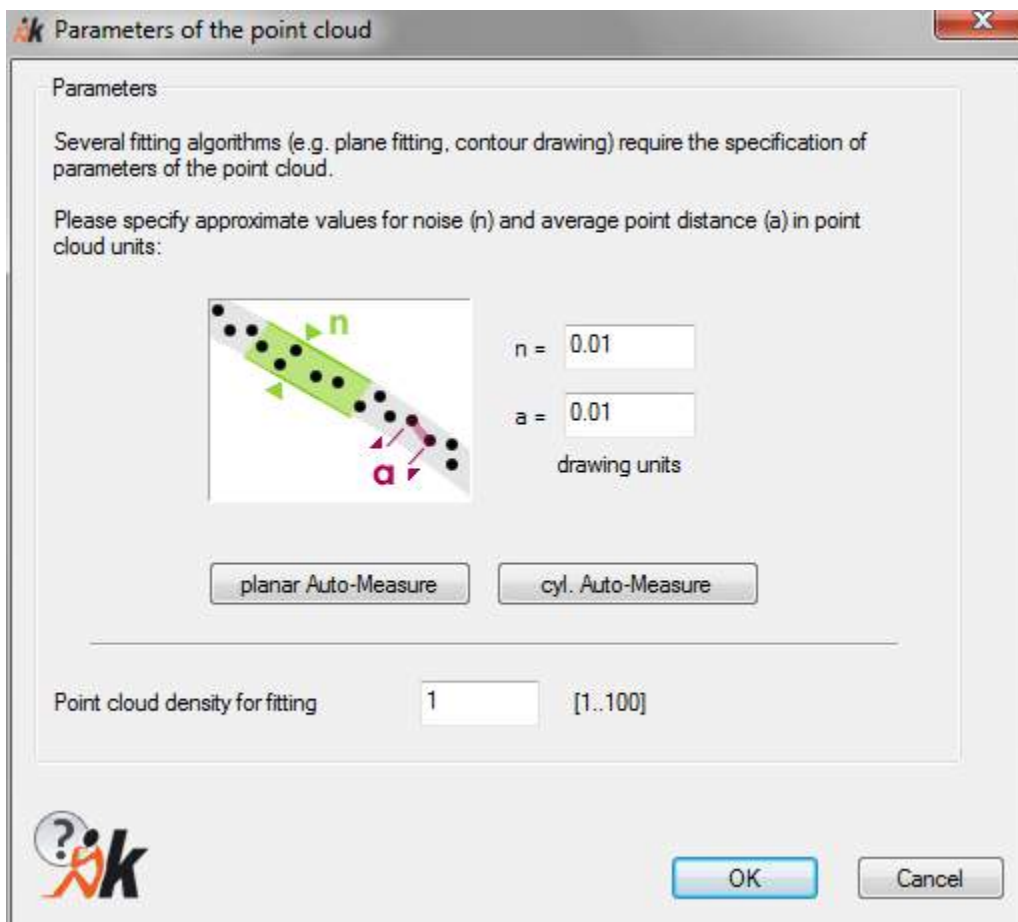
	Active	Inactive
Loaded	Green	Color of point cloud object (e.g. "byLayer")
Unloaded	(not possible)	Yellow

Point clouds – Set point cloud parameters...

	Command Name:	PCSETPARAMS
	Ribbon Tab:	As-Built Modeler

This command allows you to change the parameter for noise (n) and average distance of points (a).

The following dialog appears:



The parameters for an Autodesk point cloud are saved in the drawing, *not* in the referenced point cloud file. If you insert the point cloud in another drawing, you need to specify the parameters again.

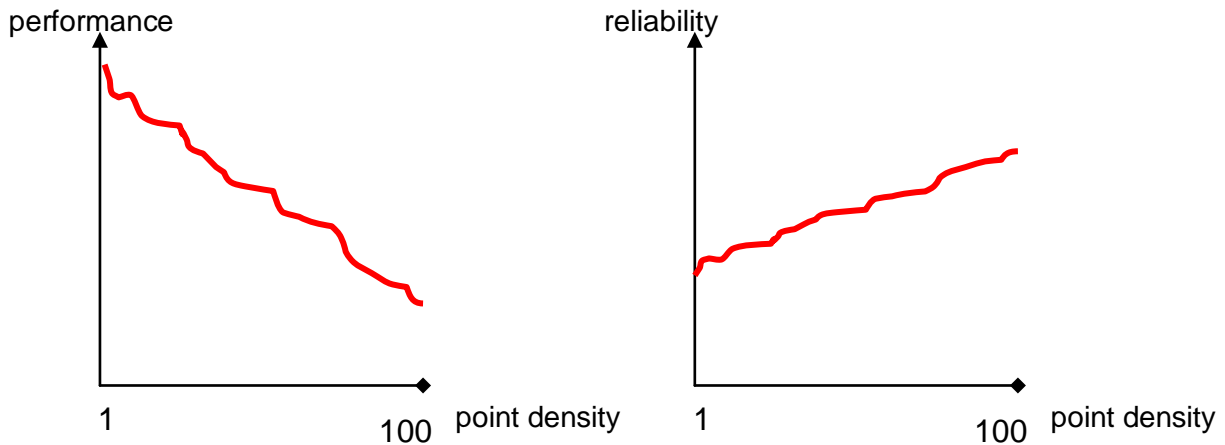
Hint: For FARO point clouds the PTC file (old format) is updated with these parameters. Depending on the size of the point cloud this process may take some time to complete.

For many modelling algorithms the correct setting of the point cloud parameters is very important. It influences the quality of the results. You may determine the values of a and n automatically by using the button **[planar Auto-measure]** or **[cyl. Auto-measure]**. Clicking on one of the Auto-measure buttons you get into the drawing and are asked to crop a small planar or cylindrical section of the point cloud and finally run the calculation of the point cloud parameters based on that section.

With the setting in the field “Point cloud density for fitting” you specify the amount of points to be used in modeling commands (e.g. →**Modeling – Fit polygon**, →**Modeling – Plane – Fit plane**).


The specified value will be saved in the configuration file.

The following chart illustrates the correlation between the point density and the performance of modeling/fitting commands (left hand side) and the correlation between point density and reliability of the result (right hand side).



The default value for the point density is 10. If you are working with large point clouds (several 100 million points) it is recommended not to increase this value.

Point clouds – Reindex AutoCAD point cloud

	Command Name	PCACREINDEX
	Ribbon Tab	As-Built Modeler


With this command you can create a new AutoCAD point cloud from one or multiple loaded AutoCAD point clouds.

Command: **PCACREINDEX**

In the following file selection dialog specify the file path and name of the point cloud file to be created.

You can use this command to **crop** and **merge** the currently loaded AutoCAD point clouds to their visible points.

Point clouds – Export LAS file

	Command Name	PCEXPORLAS
	Ribbon Tab	As-Built Modeler


With this command you can create a LAS file which contains all currently visible points of all AutoCAD point clouds of the drawing.

After running the command you are prompted for the name of the LAS file to be created then the export starts.

The colors of the sections are considered for the export,

Since LAS is a standardized format these files can be used for data exchange between various software systems. The export in our software refers to version 1.3 of the LAS standard.

Point clouds – Convert to AutoCAD 2015




	Command name	PCCONVERTDWGTO2015
	Ribbon Tab	As-Built Modeler

Use this command to convert a drawing created by an older version of AutoCAD and any embedded point clouds it contains, into ReCap format.

If you open a drawing in AutoCAD 2015 or above where there are embedded AutoCAD point clouds (PCG or ReCap RCS/RCP), As-Built operates in a kind of "Compatibility mode". The use of certain features is very limited when PCG point clouds are involved.

We therefore recommend converting the drawing and its point clouds to the new more efficient ReCap format. This automatic conversion consists of the following steps:

1. For every PCG point cloud that is referenced in the drawing:
 - a. The PCG point cloud is removed from the drawing.
 - b. The PCG point cloud is converted into the ReCap RCP format. The RCP file ("Example.rcp") is generated in the same folder as the original PCG file ("Example.pcg") together with a folder with support files ("Example Support"):


Name	Date modified	Type	Size
 Beispiel Support	10.04.2014 09:41	File folder	
 Beispiel.pcg	10.04.2014 09:40	PCG File	0 KB
 Beispiel.rcp	10.04.2014 09:40	Autodesk ReCap Project f...	0 KB

- c. The RCP point cloud is inserted into the drawing.
2. For each ReCap point cloud (RCS or RCP) from AutoCAD 2014 that has been inserted into the drawing:
 - a. The ReCap point cloud is removed from the drawing.
 - b. The ReCap point cloud is inserted into the drawing.

Important note:

- The conversion process can take a little time, depending on the type and size of the embedded point clouds.
- The conversion cannot be undone. Beforehand, please save a copy of the drawing!
- A drawing that has been converted, as described here, is not backwardly compatible to older AutoCAD versions.
- old PTC files are not affected by the conversion.

Point clouds – PTC format – Convert PTC point cloud to AutoCAD point cloud


	Command Name	PTC2PCG
	Ribbon Tab:	As-Built Modeler

With this command you can create an AutoCAD point cloud from a old PTC point cloud. The PTC file to be converted does not have to be loaded.

At first you need to select a PTC point cloud file using the file selection dialog.

Now the PTC is read and an AutoCAD point cloud is created and afterwards loaded. To work with the cloud, you need to set it current (or active) using the command → **Point clouds – Set** current point cloud.

Point clouds – PTC format - Load point cloud


	Command Name:	PCLP
	Ribbon Tab:	As-Built Modeler

With this command you can load a point cloud which has been inserted in the drawing with grip and bounding box.

The referenced PTC file is loaded and all points are displayed. In the case that another active point cloud already exists in the drawing (green marks) you will be asked to set the newly loaded point cloud to active.

Afterwards you may work with the point cloud.


Point clouds – PTC format - Unload point cloud

	Command Name:	PCULP
	Ribbon Tab:	As-Built Modeler

With this command you can unload a PTC point cloud. The memory occupied by the point cloud will no longer be used.

In the case that the point cloud is currently active, it is set non-active (yellow marks). Please note: you need to set another point cloud active before you can continue to work.

Point clouds – PTC format - Merge point clouds

	Command Name:	PCMERGE
	Ribbon Tab:	As-Built Modeler

Visible sections of several point clouds can be merged to a single new point cloud using this command.

If necessary, first create sections of point clouds you want to merge. This happens as usual with the section commands. Then enter the command PCMERGE.

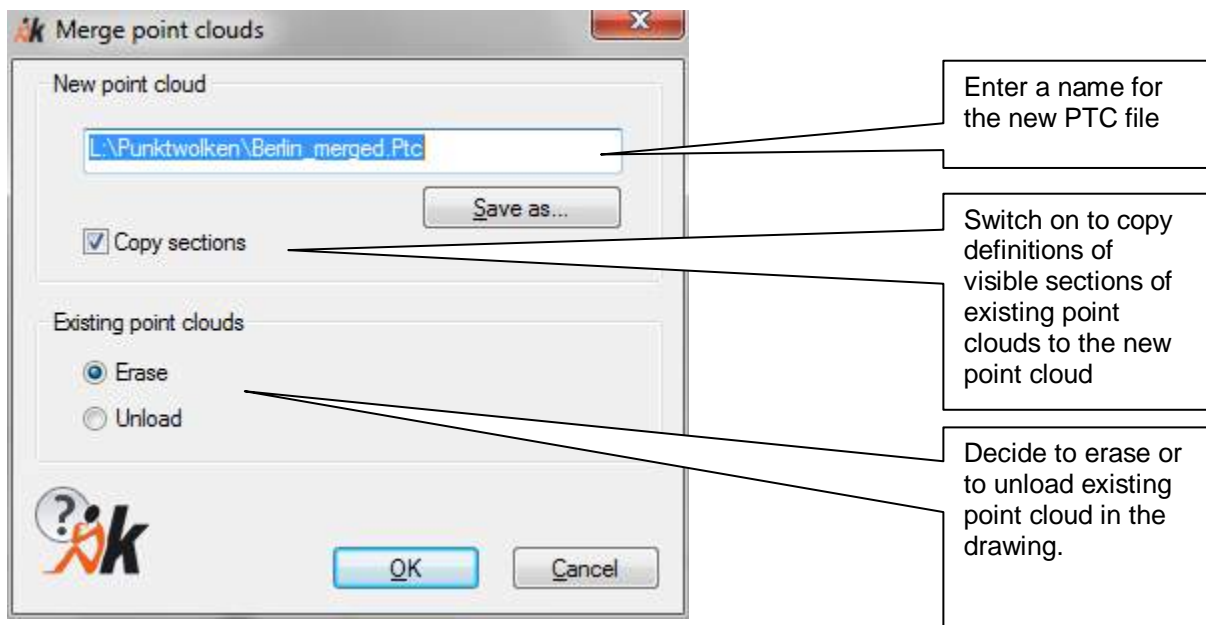
Select point clouds to merge.

Select objects:

(Select the point clouds to merge.

Point clouds must be loaded.)


Following dialog appears:



NOTE: You may overwrite one of the original point clouds. Please note this action is irreversible.

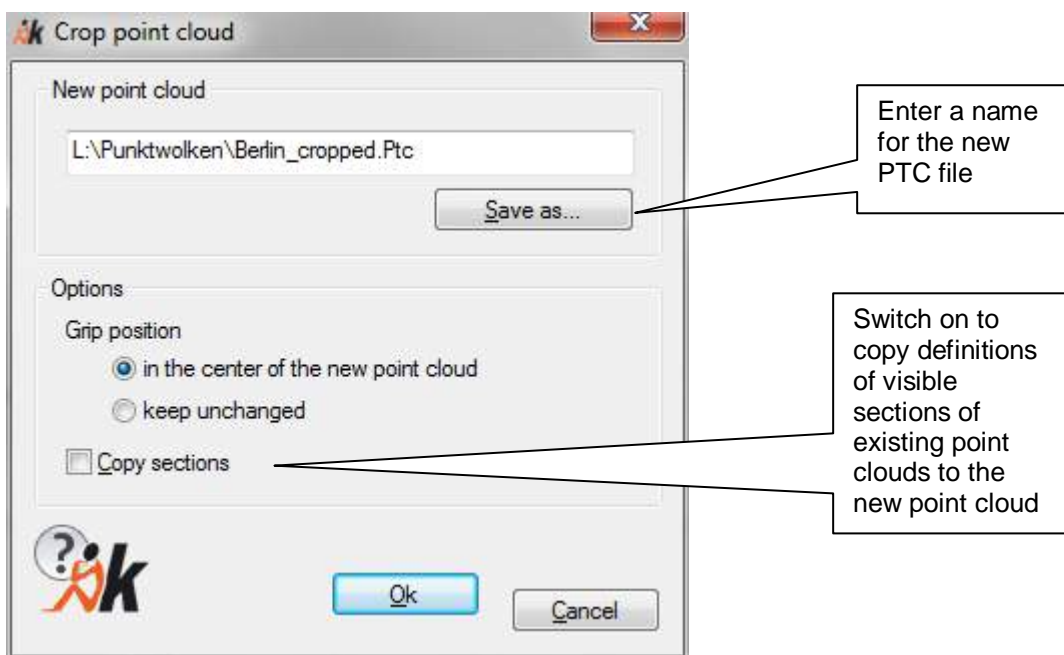
This command does not work with AutoCAD or ReCap point clouds. You can alternatively use command `_PCACREINDEX` to create a single AutoCAD point cloud from multiple AutoCAD point clouds.

Point clouds – PTC format - Crop current point cloud

	Command Name:	PCCROPCLLOUD
	Ribbon Tab:	As-Built Modeler

With this command you can reduce one point cloud to its visible section. A new point cloud (PTC file) is generated.


Following dialog appears:



NOTE: You may overwrite the original point cloud. Please note this action is irreversible.

This command does not work with AutoCAD or ReCap point clouds. You can alternatively use command `_PCACREINDEX` to create a single AutoCAD point cloud from multiple AutoCAD point clouds.

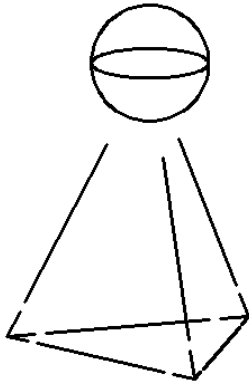
Point clouds – PTC format - Set grip position

	Command Name:	PCSETGRIP
	Ribbon Tab:	As-Built Modeler

Place the grip using this command.

Specify new grip position:


(click the new position in your drawing)



It is recommended but not necessary to place the grip within the bounding box. So it is easier to associate grip and point cloud.

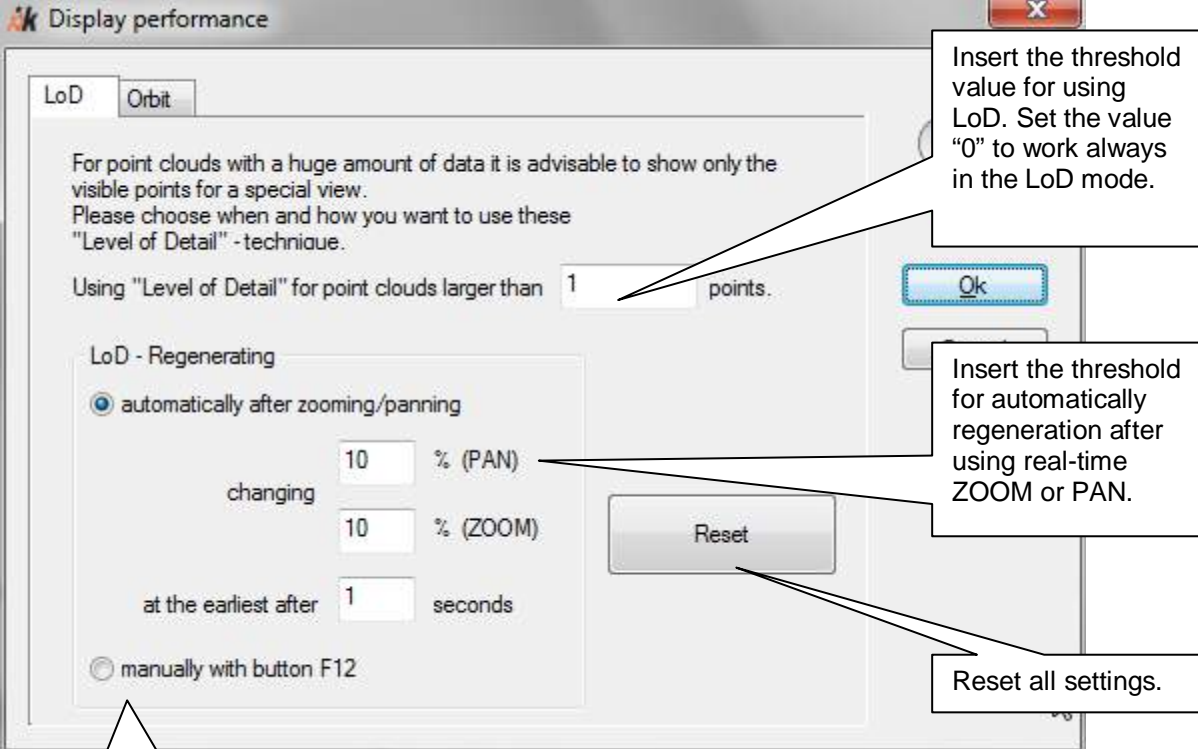
Afterwards the PTC file is updated with the new position. Depending on the size of the point cloud this process may take some time to complete.

Point clouds – PTC format - Display settings

	Command Name:	PCCFGDISPLAY
	Ribbon Tab:	As-Built Modeler

For point clouds with a huge amount of data it is advisable to show only the visible points for a given view. This technique is named „Level of Detail“ (LoD). REGENERATING of the drawing after using real-time ZOOM or PAN can be done manually or automatically.

Switching between “draw all points” and “draw only visible points” is controlled regarding the number of points. This threshold can be set in the LoD dialog. Point clouds larger than this number will be drawn applying the LoD-technique.



Display performance

LoD Orbit

For point clouds with a huge amount of data it is advisable to show only the visible points for a special view. Please choose when and how you want to use these "Level of Detail" - technique.

Using "Level of Detail" for point clouds larger than 1 points.

LoD - Regenerating

automatically after zooming/panning

changing 10 % (PAN)

10 % (ZOOM)

at the earliest after 1 seconds

manually with button F12

Reset

Ok

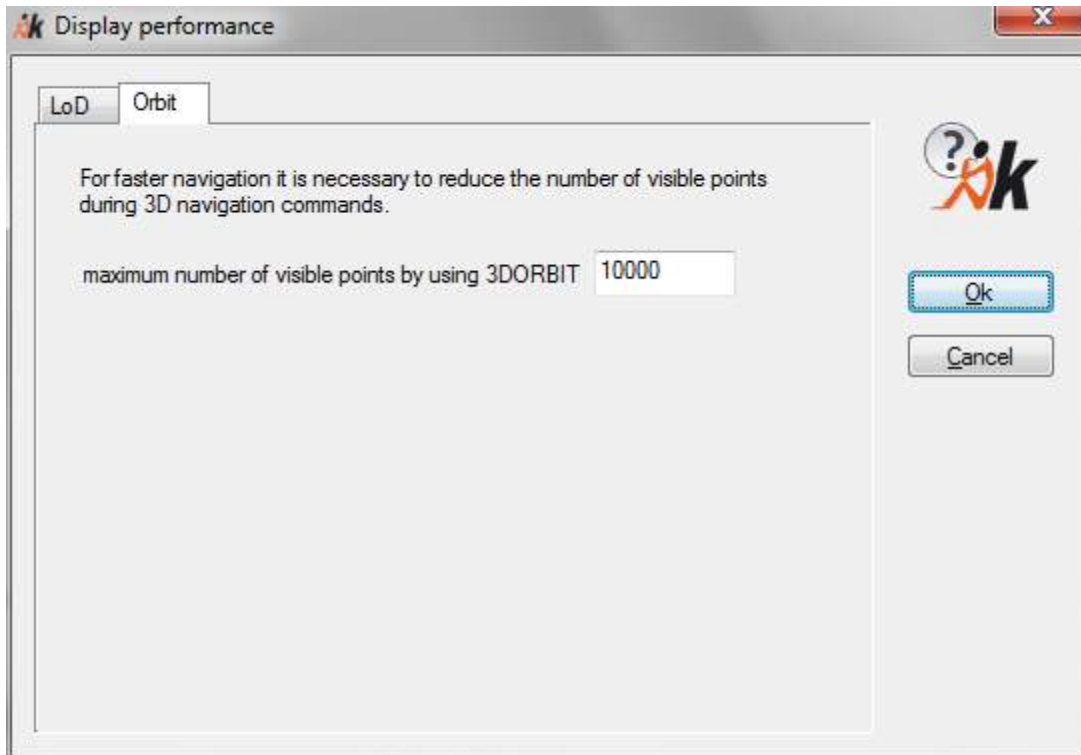
Insert the threshold value for using LoD. Set the value "0" to work always in the LoD mode.

Insert the threshold for automatically regeneration after using real-time ZOOM or PAN.

Reset all settings.


Activate this button for a manual regeneration of the drawing. To regenerate point clouds use F12.

For fast navigation in a point cloud with 3DORBIT the number of visible points should be reduced. You can enter the maximum number of points in the following dialog.




NOTE: The Display settings do only apply to PTC point clouds. For Autodesk ReCap point clouds you may set the point density using the AutoCAD command “_PointCloudDensity”.

Point clouds – PTC format - Show / hide bounding box

	Command Name:	PCBOUNDINGBOX
	Ribbon Tab:	As-Built Modeler


This command switches the bounding box of the active point cloud on or off.

Point clouds – PTC format - Show / hide grip

	Command Name:	PCSHOWGRIP
	Ribbon Tab:	As-Built Modeler

With this command you may show or hide the point cloud grip.

Point clouds – PTC format - Set point size

	Command Name:	PCPOINTSIZ
	Ribbon Tab:	As-Built Modeler

Please switch on a visual style other than *2D Wireframe*.

Now you may change the point size:

Command: PCPOINTSIZ

Input new point size (between 1 and 10) for all point clouds <1>: 2

Point size set to 2.

A point size of 2 means all points have a size of 2 x 2 pixels, always aligned to the current view.


The display of points happens dynamically, depending on ZOOM Level and view. That is why the “Level of Detail” mode (LoD) must be active. You can set this LoD mode with the command →**Point clouds – PTC format - Display** .

The adjusted point size will be saved in the configuration file and applies to all drawings and point clouds.

Example: Point cloud with point size 1 to 3 (from left to right)



Import – Create AutoCAD point cloud (ReCap)


	Command Name:	KUBT_PC_INDEXPOINTCLOUD
	Ribbon Tab:	As-Built Modeler

This command starts ReCap, the Autodesk software for importing numerous scan data formats into the ReCap point cloud format (RCP/RCS) supported by all Autodesk products.

ReCap is usually installed with every AutoCAD installation. The import functions of ReCap are free of charge.

Section **3.1.3** of this manual briefly describes how to use ReCap.

Sections – Define slice (UCS)

	Command Name:	PCS
	Ribbon Tab:	As-Built Modeler

With this command you can define a slice to select points in the active point cloud. Only the visible points in the slice will be drawn in the AutoCAD-Drawing. You can define the slice in the current user co-ordinate system (UCS).

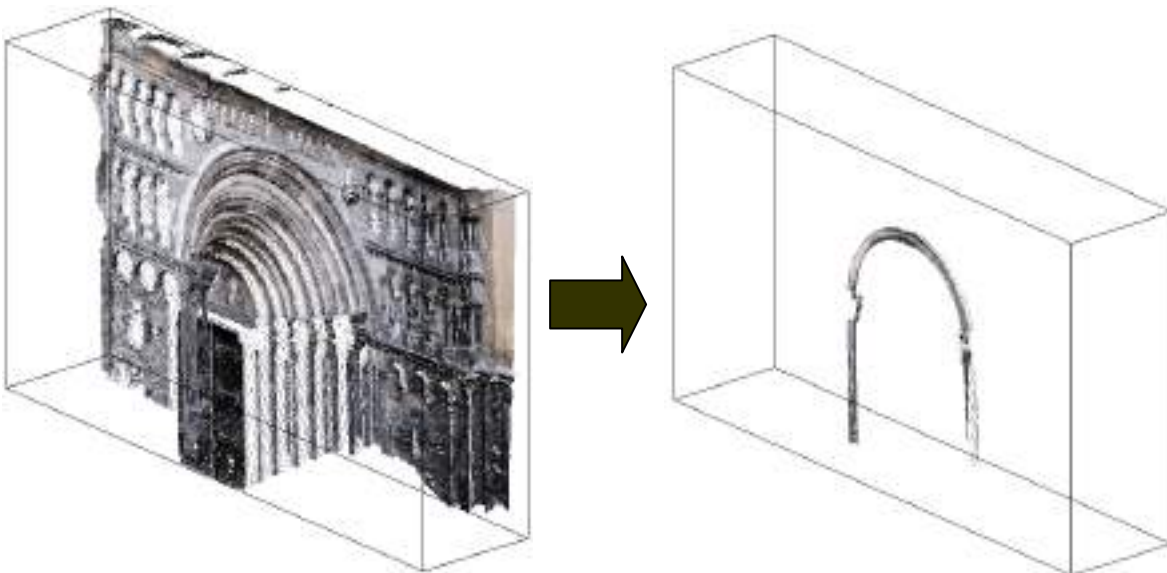
command: KUBT_PC_SLICE

Slice parallel to [XY/YZ/XZ] plane UCS <XZ>: *(Plane, to which the slice is parallel)*

First point for boundary of slice: *(Select a point for one boundary of the slice.)*


Second point for boundary of slice or [Thickness] <Thickness>:
(Select a point for the second boundary of slice or write down the thickness of slice.)

Example:



When choosing the Thickness option half of the entered measure is considered to both sides of the first clicked point. So the first point lies on the middle plane of the slice.

Sections – Define horizontale slice

	Command Name:	PCSZ
	Ribbon Tab:	As-Built Modeler

With this command you define a horizontal slice, that contains all points that are parallel to the UCS XY plane and lie at a specific Z height. The thickness of the slice can be entered.

Command: **KUBT_PC_SLICE_Z**

Slice parallel to UCS plane [XY/YZ/XZ] <XY>: **_XY**

First point for boundary of slice: .xy of (need Z): **8.4**


*(Enter the Z height, on which the slice should be created
or pick a point)*

Second point for boundary of slice or [Thickness] <Thickness>: **_T**

Specify slice thickness <0.5>: **0.1**

(Enter the thickness of the slice)

Sections – Define slice (aligned with planar AutoCAD object)

	Command Name:	PCAS
	Ribbon Tab:	As-Built Modeler

With this command you define a slice of points, which are within a given distance (thickness) from a planar AutoCAD object. The AutoCAD object will be centered in the slice.

Planar AutoCAD objects are kubit planes, circles, arcs or (2D) polylines, for instance.

Command: KUBT_**PC_ALIGNED_SLICE**


Select a planar AutoCAD object: *(Select a planar AutoCAD object)*

Specify slice thickness <0.2>: **0.1** *(Specify the slice thickness, press ENTER to apply the default value)*

If you have selected a non-planar (thus non-supported) AutoCAD object in the first step, the following message will appear in the command line:

The selected object is linear. Slice alignment plane cannot be determined.

Sections – Shift slice

	Command Name:	PCSS
	Ribbon Tab:	As-Built Modeler

With this command you can move a defined slice and select the visible points in the new position of the slice. You can move the slice downwards or upwards with changed or unchanged thickness value.


command: **KUBT_PC_SLICESHIFT**

Absolute value of shift of slice downwards [Upwards/Downwards] <2.928800>:

(Move the slice by Thickness or by a new value)

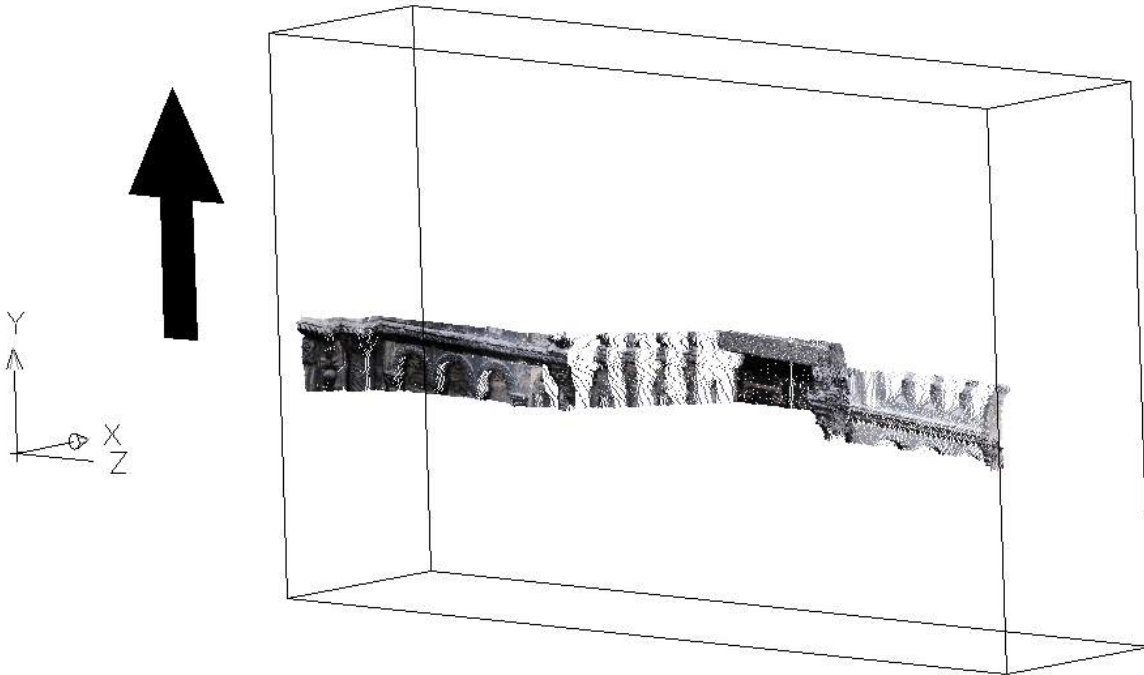
Till the operation is ended by pressing the ESC button the operation can be repeated over and over again.

Sections – Shift slice upwards


	Command Name:	PCSSUP
	Ribbon Tab:	As-Built Modeler

With this command you can move a defined slice upwards and select the visible points in the new position of a slice.

Example:

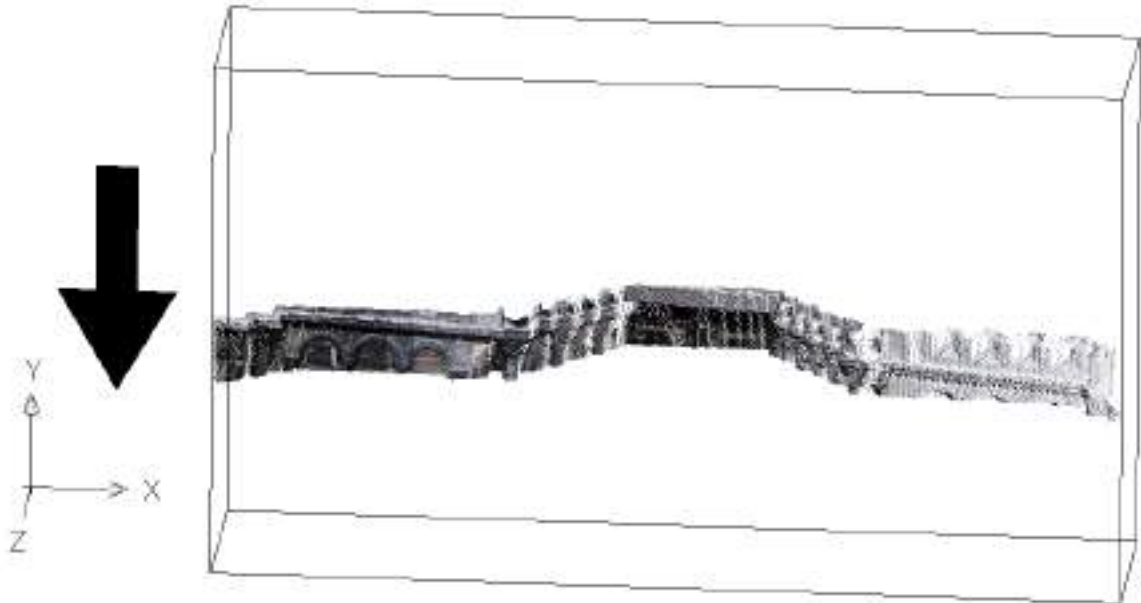


Sections – Shift slice downwards


	Command Name:	PCSSDOWN
	Ribbon Tab:	As-Built Modeler

With this command you can move a defined slice downwards and select the visible points in the new position of a slice.

Example:



Sections – Change slice thickness

	Command Name:	PCSCT
	Ribbon Tab:	As-Built Modeler

With this command you can change the thickness of the current visible slice.

Command: KUBT_PC_SLICECHANGETHICKNESS

Enter the new slice thickness <2.169000>: *(Enter the new thickness of the slice.)*


Alignment relative to current slice [Top/Bottom/Center] <Center>:

(Select the alignment for the new slice:

Top → top of the slice is unchanged, slice expands at the bottom,

Bottom → vice versa, Center → slice expands at both sides by the same amount)

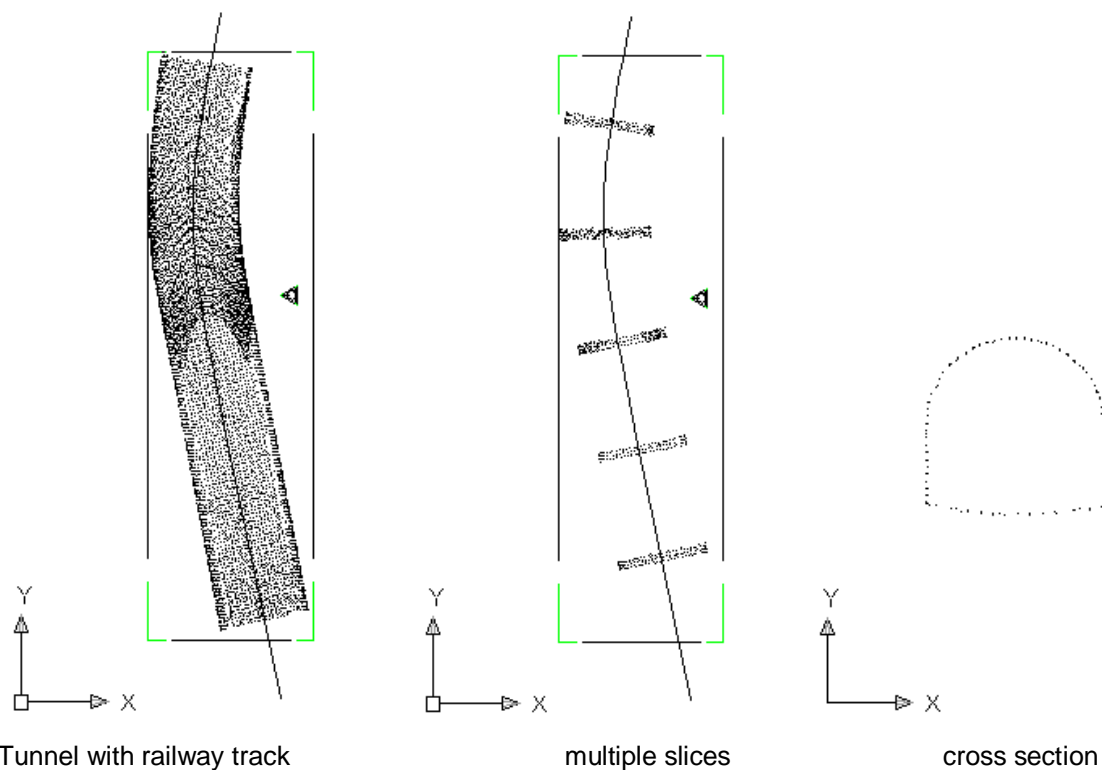
Sections – Define multiple slices

	Command Name:	PCMSC
	Ribbon Tab:	As-Built Modeler

With this command you can create multiple slices in one single step. It creates slices with a certain thickness within a user defined range parallel to a UCS - plane or perpendicular along an AutoCAD curve object.

Example: There is a scan of a tunnel with a railway track inside and you need several cross sections of the tunnel. Now you can create multiple slices perpendicular along an AutoCAD curve – the railway track. To evaluate one cross section you may use the command → **Digitizing tools – UCS from slice** and set it to “Plan view”

See the following pictures illustrating this example:

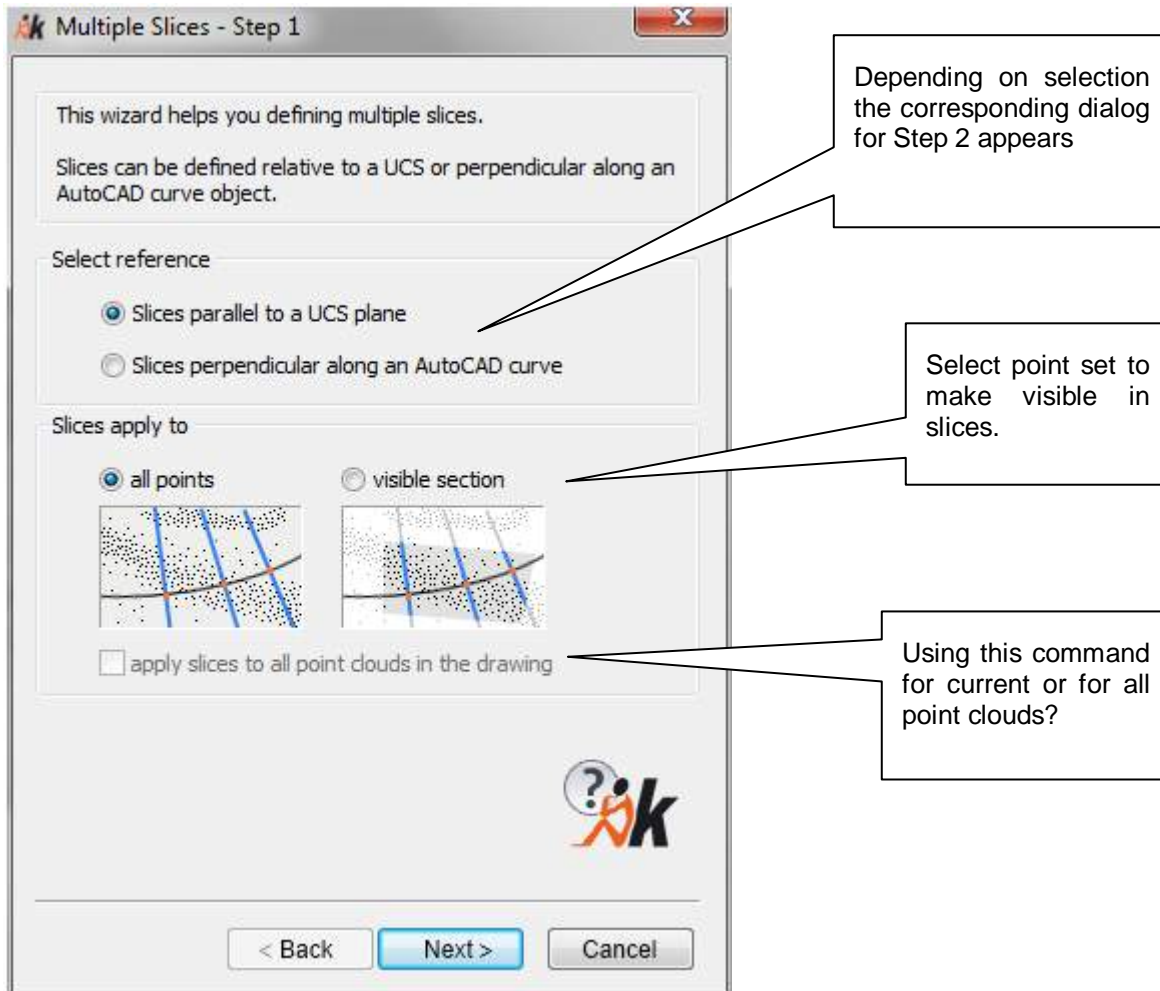


By using this command to define multiple slices, the slices can automatically be positioned in a constant distance or (additionally) by entering values or by mouse clicking in your AutoCAD drawing. Distance means the distance between the centers of the slices. The first slice centers at one of three places, with the bounding box in the selected UCS plane, with the start point of an AutoCAD curve or with a user defined start point.

After calling the command “PCMSC” the wizard starts and helps you to define multiple slices.

Multiple Slices – Wizard, Step 1:

In the dialog box, select the reference system and the section to generate slices. If the option “apply slices to all point clouds in the drawing” is active, slices will be generated in all point clouds, including unloaded point clouds (see also “**Point clouds – PTC format - Load point cloud**” and “**Point clouds – PTC format - Unload point cloud**”) When an unloaded point cloud is loaded again after finishing the multiple slice command, slices are immediately visible in the reloaded point cloud.



Multiple Slices – Wizard, Step2:

Depending on the reference system selected in step 1, the assistant offers the following options for step 2.

- **If you have set the reference system to "Slices parallel to a UCS plane" in step 1:** There is the possibility to choose between the current UCS and a named UCS. In the latter of the two all named UCS of the drawing are listed. Furthermore, you have to decide the alignment of the slices: parallel to XY-, YZ- or ZX-plane of the selected UCS.
- **If you have set the reference system to "Slices perpendicular along an AutoCAD curve" in step 1:** After clicking the button “Select curve in AutoCAD...” you are asked to select the curve to generate slices . You may choose the following kinds of curves: line, polyline, 3d-polyline, arc, circle, ellipse and spline. The coordinates of start and end point will be shown and

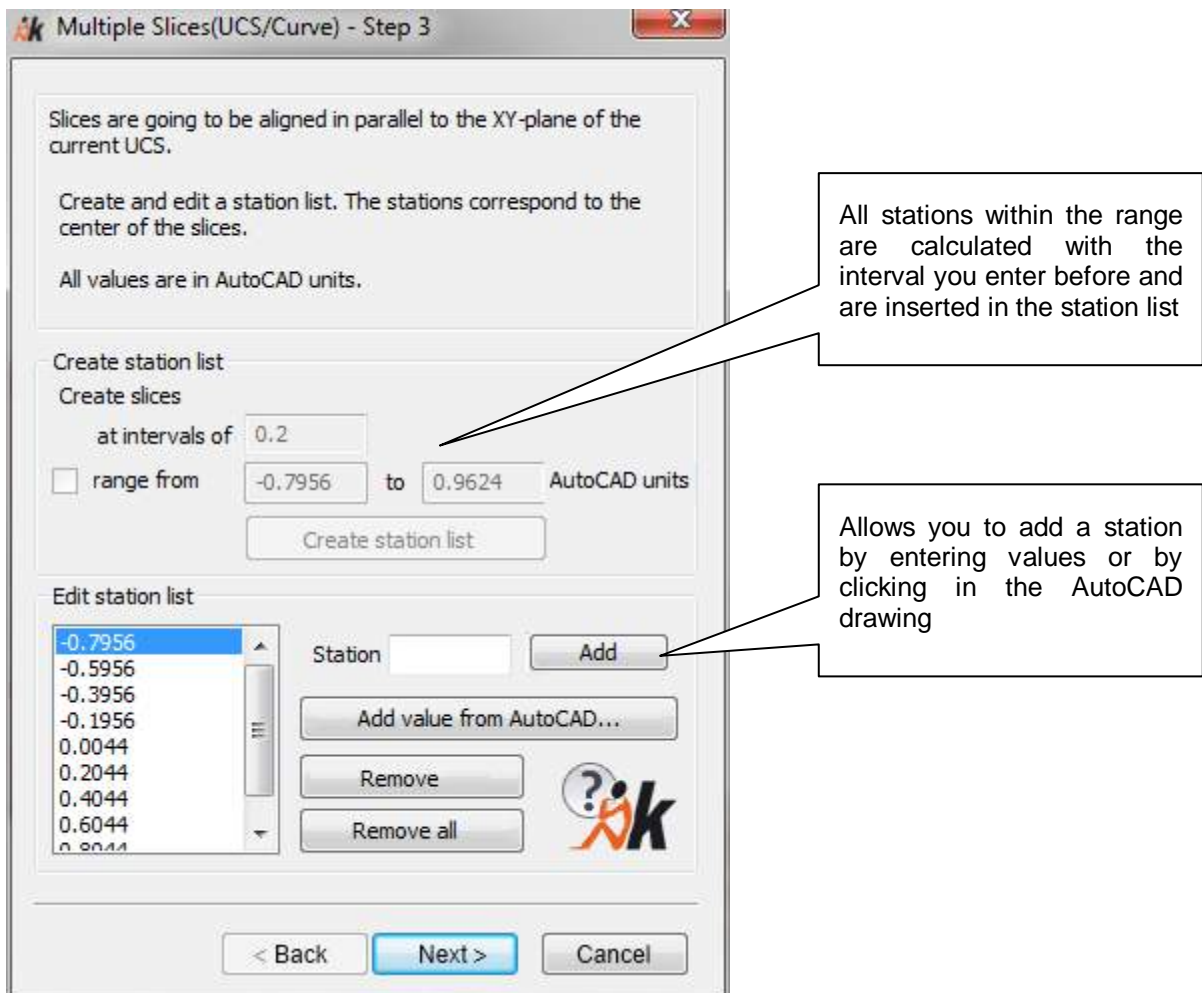
can be exchanged by setting this option. That only means the starting position of the station list is defined. Neither the curve nor the following procedure in the wizard (creating a station list in step 3) is changed. Select “Next >” to begin step 3 of the wizard.

Multiple Slice – Wizard, Step3:

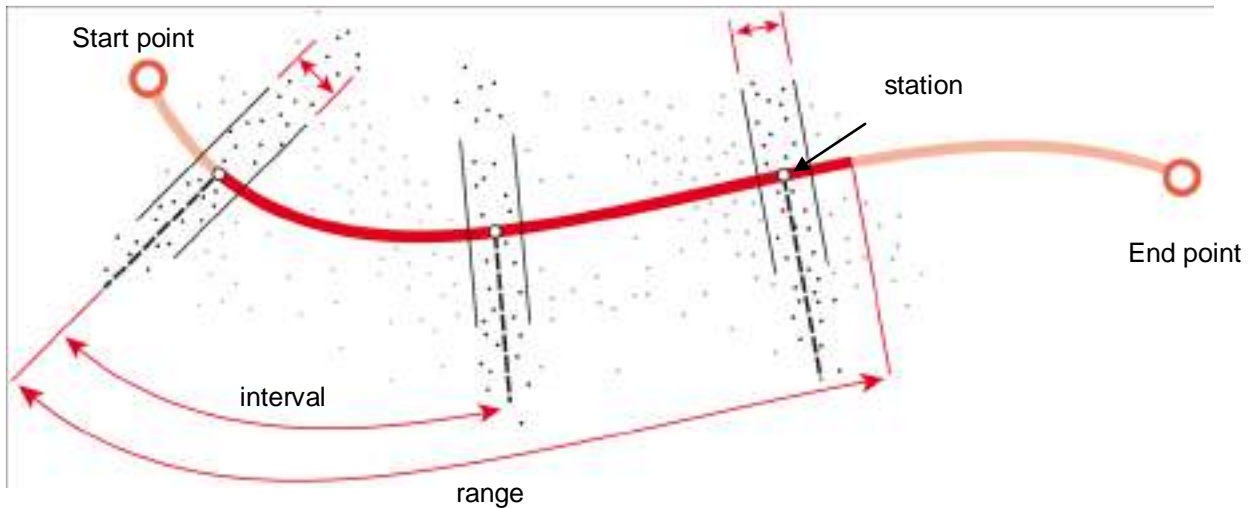
Independent of the reference system configured in step 1 and 2 the following dialog appears:

The part *Create station list* allows you to enter a constant interval and range to generate slices within.

With the button “Create station list” the corresponding positions of the slices are calculated and listed. If you do not activate the option to enter a user defined range, all slices are generated within the grey marked total range that was calculated before. If you do activate the option, the fields will automatically be filled with calculated values which depend on the selected reference system. They result either from the section of the bounding box when the reference system is a UCS or from the length of a selected AutoCAD curve.



Slices are also generated within the specified range when the option “Slices apply to → visible section” is active. Then slices **out** of the visible section contain no points and the user can delete these slices in the section manager.



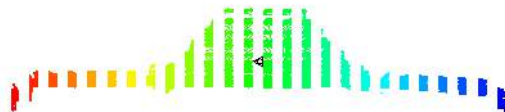
The part *Edit station list* allows you to delete or to add several stations to the list. To access it enter in the field “value” the requested station and click the button “Add”. Furthermore you can “Add value from AutoCAD...”. Afterwards you are asked to click a point in the AutoCAD drawing whereupon the value for the station will be calculated and added to the list.

To go “<Back” in the wizard one step the station list must be empty (“Remove all”). On the other hand a station list (with at least one value) must exist in order to continue “Next” in the wizard.

Multiple Slices – Wizard, Step 4:

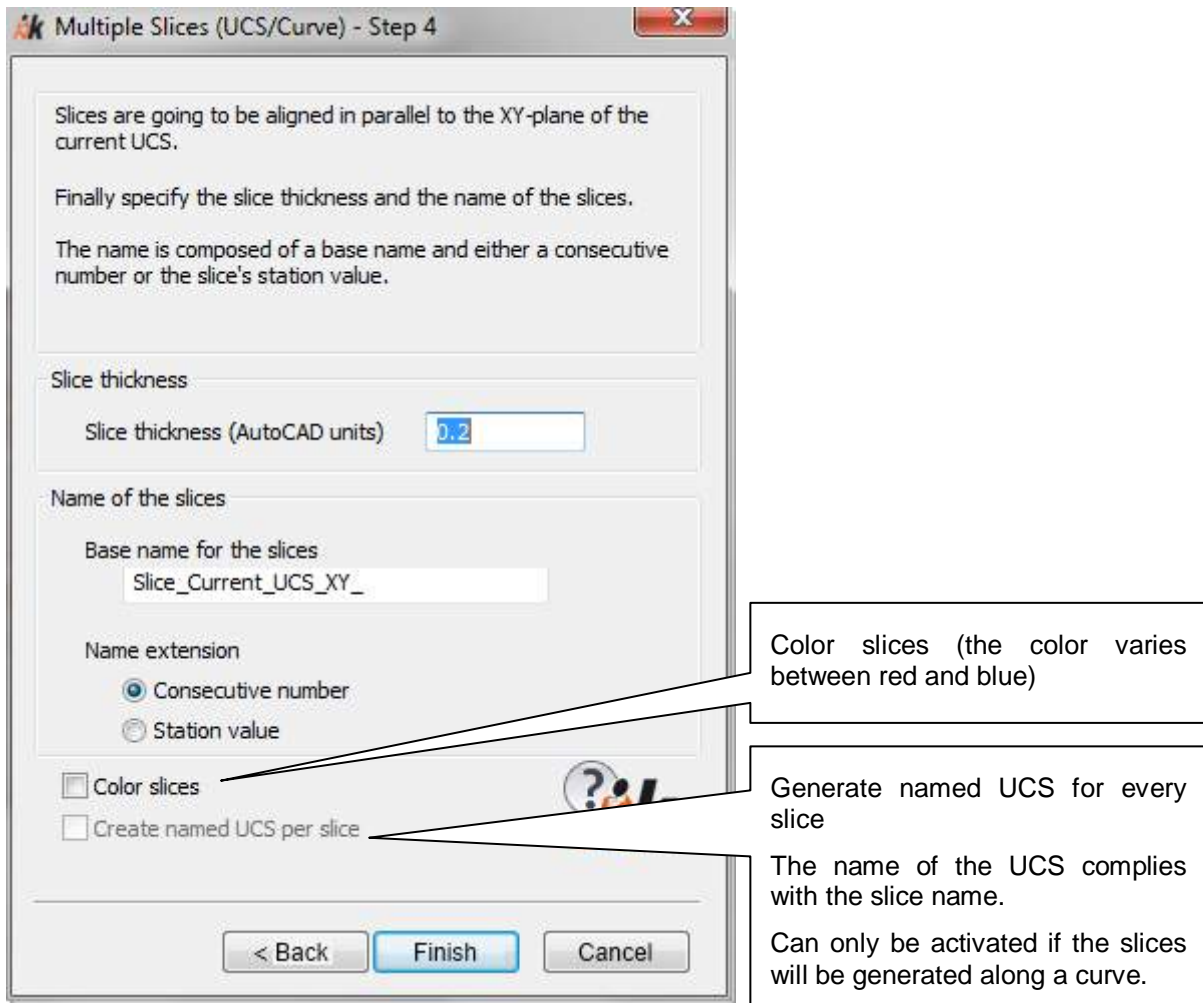
The wizard offers you a base name which may be changed by the user. Enter the thickness of the slices and the suffix of the slice name: by consecutive number or station value. Click “Finish” to generate slices.

Select if you would like to have the generated slices (layers) colored automatically. The coloring will diversify between blue and red (along the curve or the selected UCS axis)




If you generate slices along an AutoCAD curve, you may generate a named UCS for every slice. The name of the UCS complies with the slice name. Existing UCS with the same name will be overwritten.

After clicking the “Finish” button the slices will be generated. Slices which lay at the beginning or the end of the slice sequence and do not contain points will not be generated.



Note: The colouring of slices only works for PCG and PTC point clouds, but not for point clouds in the ReCap format.

Sections – Define rectangular

	Command name:	PCSR
	Ribbon Tab:	As-Built Modeler

With this command you can do a rectangular cut away on all visible points in a point cloud. The rectangle is always parallel with the view port.

Command: **PCSC**

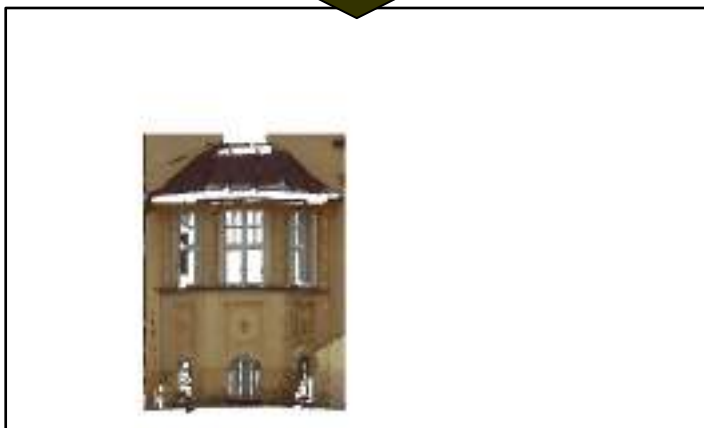
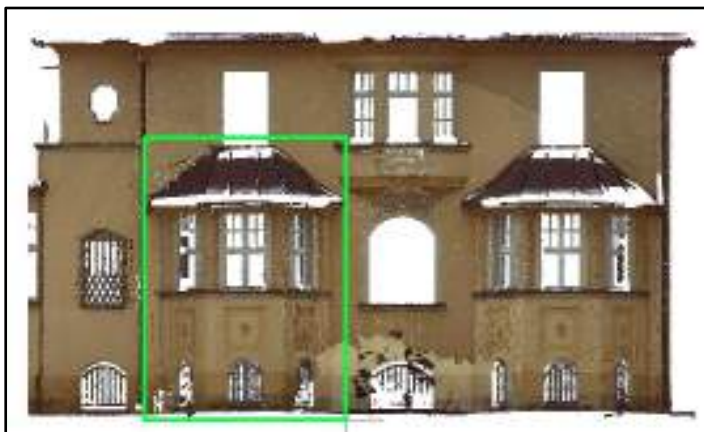
Keep points inside or outside? [Inside/Outside] <Inside>: _Inside

(The option "Inside" is always selected when called from the menu)


Specify first corner point: *(Pick point for selection)*

Specify other corner point: *(Pick point for selection)*

Example:



Sections – Exclude rectangular

	Command name:	PCSR
	Ribbon Tab:	As-Built Modeler

With this command you can do an exclude cut away on a rectangle of all visible points in a point cloud. The rectangle is always parallel with the view port.

Command: **PCSC**

Keep points inside or outside? [Inside/Outside] <Inside>: _Outside

(The option "Outside" is always selected when called from the menu)


Specify first corner point: *(Pick point for selection)*

Specify other corner point: *(Pick point for selection)*

Example:



Sections – Define clipping box

	Command Name:	PCSBB
	Ribbon Tab:	As-Built Modeler

With this command you can define a bounding box to select points. Only the visible points in the box will be drawn in the current AutoCAD-Drawing. You define the box in the current user co-ordinate system (UCS).

command: **PCSBB**

Definition of a UCS parallel box:

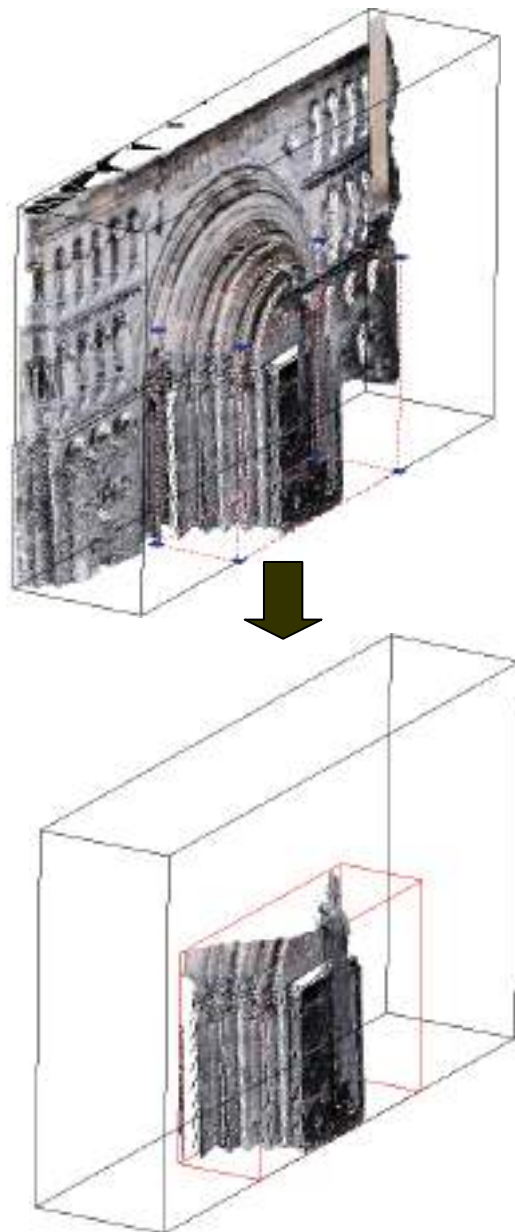
Box corner:

(Select a corner of the box.)


Opposite box corner:

(Select the opposite corner of the box.)

Example:



Sections – Define circular

	Command name:	PCSC
	Ribbon Tab:	As-Built Modeler

With this command you can do a circular cut away on all visible points in a point cloud. With this you can edit, in particular scanned pipes and tanks as well as towers, columns and other cylindrical objects. The circular base can be defined by using three points, centre point and radius or also by selecting an existing circle in the drawing. In the first two cases the surface of the circle is always parallel to the view window, whereas in the last case the resulting cylindrical region is defined along the normal to the selected circle.

Command: **PCSR**

Keep points inside, outside or shell? [Inside/Outside/Shell] <Inside>: _Inside

(The option "Inside" is always selected when called from the menu)

Specify first point on circle or [Center point and radius/Select object] <Select object>:

(Pick point for selection)

Specify second point on circle:


(Pick point for selection)

Specify third point on circle:

(Pick point for selection)



Sections – Exclude circular

	Command name:	PCSC
	Ribbon Tab:	As-Built Modeler

With this command you can do an exclude circular cut away on all visible points in a point cloud. With this command you can edit, in particular scanned pipes and tanks as well as towers, columns and other cylindrical objects. The circular base can be defined by using three points, centre point and radius or also by selecting an existing circle in the drawing. In the first two cases the surface of the circle is always parallel to the view window, whereas in the last case the resulting cylindrical region is defined along the normal to the selected circle.

Command: **PCSR**

Keep points inside, outside or shell? [Inside/Outside/Shell] <Inside>: _Outside

(The option "Outside" is always selected when called from the menu)

Specify first point on circle or [Center point and radius/Select object] <Select object>:

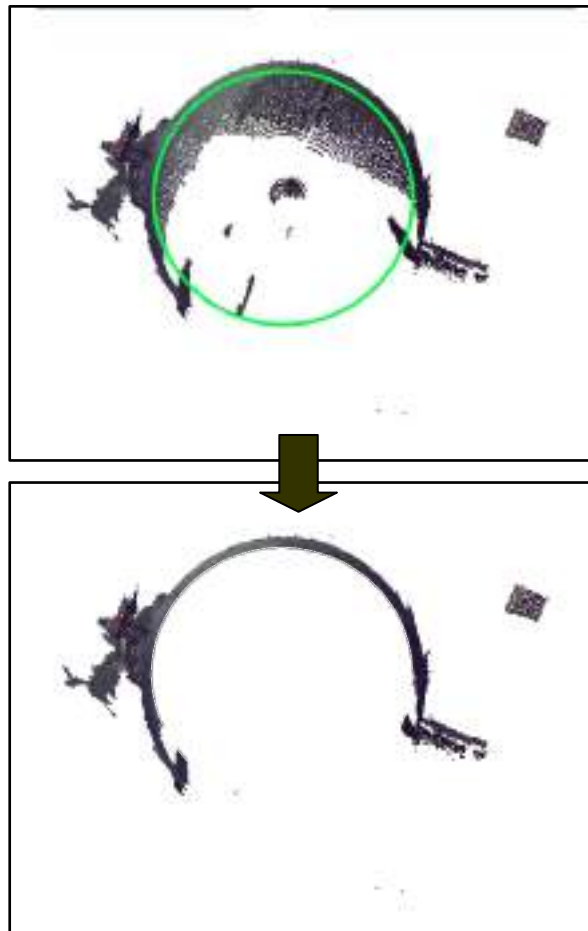
(Pick point for selection)

Specify second point on circle:


(Pick point for selection)

Specify third point on circle:

(Pick point for selection)



Sections – Define circular shell

	Command name:	PCSC
	Ribbon Tab:	As-Built Modeler

With this command you can do a ring cut away on all visible points in a point cloud. With this command you can edit, in particular scanned pipes and tanks as well as towers, columns and other cylindrical objects. The circle, on which the ring should be defined, can be defined using three points, centre point and radius as well as by selecting an existing circle in the drawing. In the first two cases the ring is always parallel to the view window, whereas in the last case the resulting cylindrical, ring region is defined along the normal to the selected circle. The thickness of the ring can be defined either by using the mouse or it can be entered by using the option [Thickness].

Command: **PCSR**

Keep points inside, outside or shell? [Inside/Outside/Shell] <Inside>: _Shell

(The option "Shell" is always selected when called from the menu)

Specify first point on circle or [Center point and radius/Select object] <Select object>:

(Pick point for selection)

Specify second point on circle:

(Pick point for selection)

Specify third point on circle:

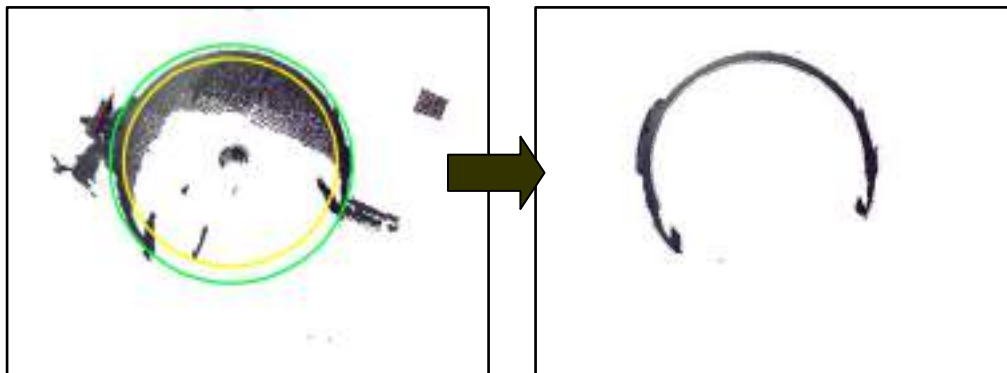
(Pick point for selection)

Specify first radius of cylindrical shell or [Thickness] <Thickness>:


(Drag the inner diameter by using the mouse or enter the thickness by choosing the option)

Specify other radius of cylindrical shell or [Finish] <Finish>:

(Drag the outer diameter by using the mouse)



Sections – Define clipping polygon

	Command Name:	PCSP
	Ribbon Tab:	As-Built Modeler

With this command you can display all visible points defined by a polygon based prism (in perspective view: truncated pyramid). The direction of the prism or truncated pyramid is determined by the viewing direction in the current view. You can determine the polygon yourself by clicking points, or by using the option "*Select object*" and selecting an already existing polygon in the drawing.

Hint:

If you prefer you can use other section commands (slice or rectangular cut away) before this command.

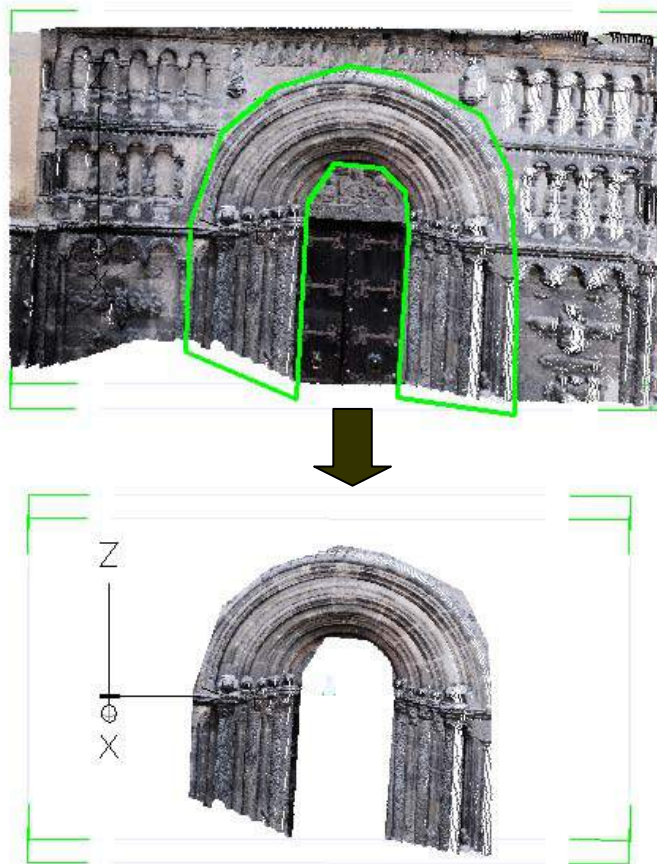
The polygonal cut away can significantly reduce the display speed of the point cloud and especially those from ReCap data.

command: **PCSP**


Draw the clipping polygon (inside is visible) or [Select object] <Select object>:

(Pick the corners of the polygon or use the option to use an already existing polygon)

Example:



Sections – Exclude polygon

	Command Name:	PCSNP
	Ribbon Tab:	As-Built Modeler

With this command you can cut away all visible points defined by a polygon based prism (in perspective view: truncated pyramid). The direction of the prism or truncated pyramid is determined by the viewing direction in the current view. You can determine the polygon yourself by clicking points, or by using the option "Select object" and selecting an already existing polygon in the drawing.

Hint:

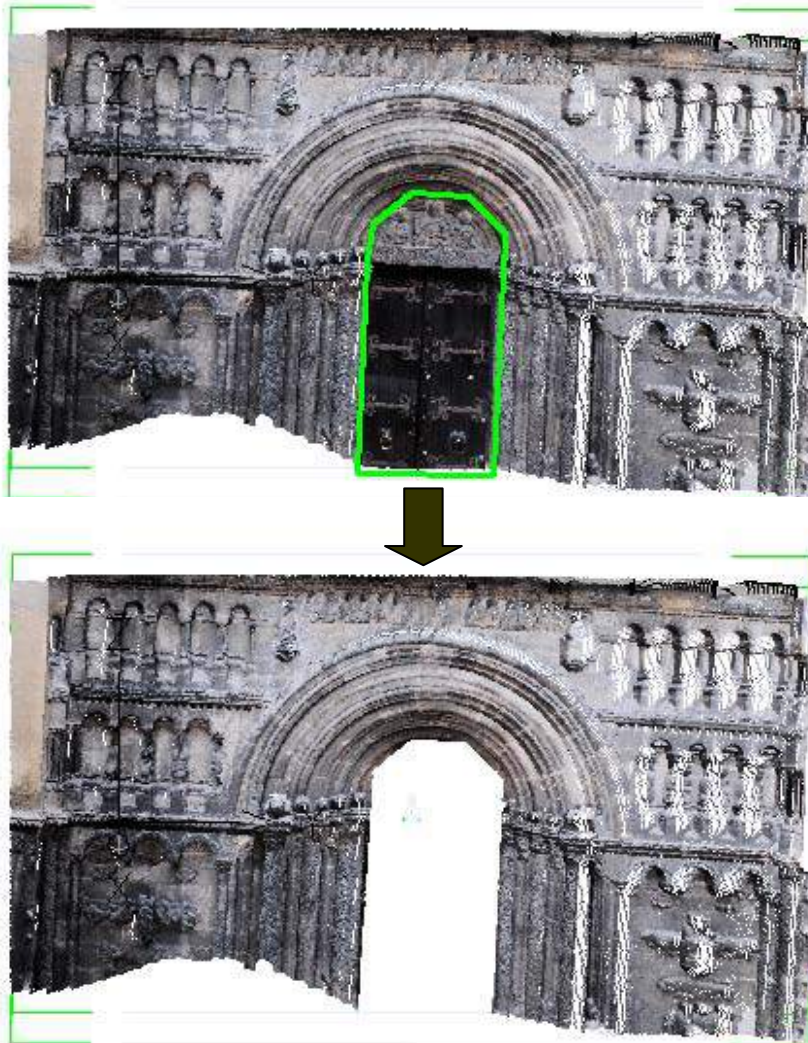
If you prefer you can use other section commands (slice or rectangular cut away) before this command. The polygonal cut away can significantly reduce the display speed of the point cloud and especially those from ReCap data.

Command: **PCSNP**


Draw the clipping polygon (outside is visible) or [Select object] <Select object>:

(Pick the corners of the polygon or use the option to use an already existing polygon)

Example:




Sections – Previous section definition

	Command Name:	PCSO
	Ribbon Tab:	As-Built Modeler

This command works analogue to the AutoCAD command "Back". It refers only to section definitions though. Other AutoCAD commands, which you have executed in the meantime, will be ignored.

Sections – Section Manager [for old PTC point clouds]

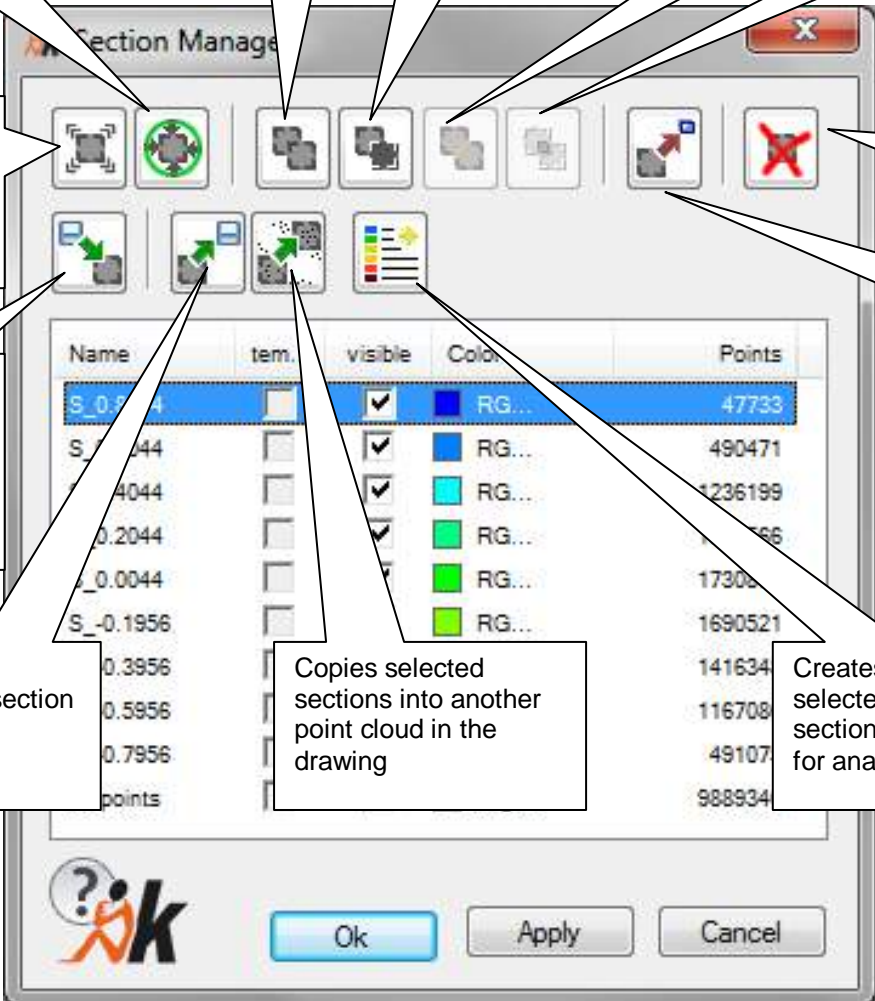
	Command Name:	PCSECMAN
	Ribbon Tab:	As-Built Modeler

With this command you start the Section Manager.

In the Section Manager you can manipulate all the sections you have previously created using several “Define sections” commands or Clash detection.

You can change a section’s color, edit its name or delete it, export sections using the Export Wizard or merge sections.

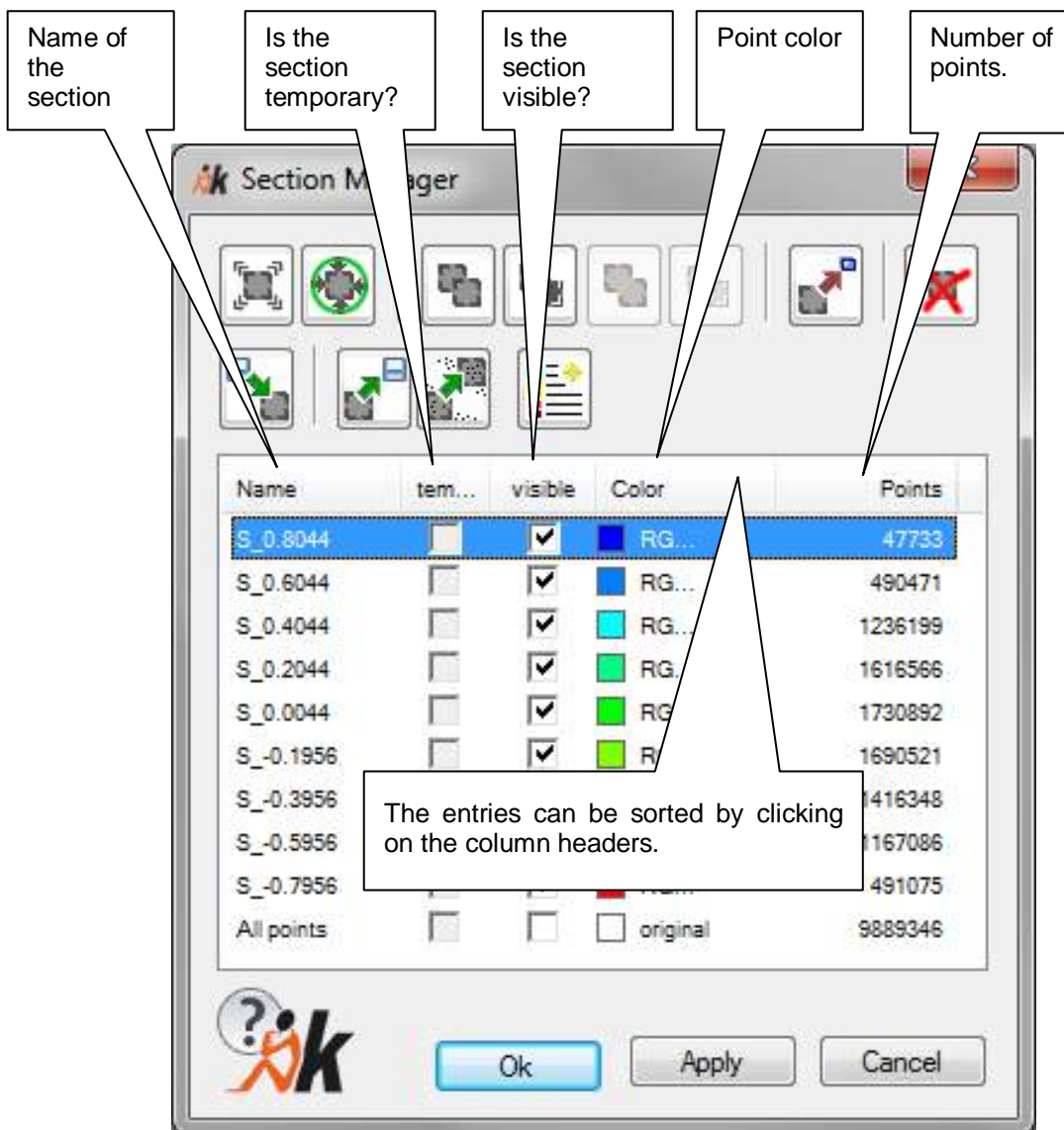
All sections and their properties (visibility, color, and name) are persistent (i.e. they are saved and loaded with the DWG) – with one exception: Temporary sections! You can recognize temporary sections by the checked box in the second column. Those sections are created by Clash Detection. For efficiency reasons they contain points by reference. If you want to **keep temporary sections** you **need to export** them!



The screenshot shows the Section Manager dialog box with various icons and a table. Callout boxes provide the following descriptions:

- Show the section marker and start AutoCAD's 3D orbit.
- Copy selected section.
- Invert the selected section.
- Disjunction of the partitions (OR).
- AND relation of the partitions.
- Delete selected sections.
- Export the selected sections' points.
- Creates legend for the selected colored sections. Useful tool for analysis.
- Copies selected sections into another point cloud in the drawing.
- Save selected sections in a section file.
- Load section definitions from a section file.
- Show selected section by scaling up its marker.

Name	tem.	visible	Color	Points
S_0.5044	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	47733
S_0.44	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	490471
S_0.4044	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	1236199
S_0.2044	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	1236199
S_0.0044	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	17300
S_-0.1956	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	1690521
S_0.3956	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	141634
S_0.5956	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	116708
S_0.7956	<input type="checkbox"/>	<input checked="" type="checkbox"/>	RG...	49107
points				988934



Description of features of the Section Manager

The sections list

If a section is selected, its marker (the bounding box) is displayed.

You can select more than one section using the SHIFT or CTRL keys and your mouse.

- **Rename a section:**

The last defined section is always named "current". If a new section is defined it will be overwritten. For that reason you should give each section a significant name. By clicking on the name of a section you can edit its name. Exception: The predefined section „All points“ cannot be renamed.

- **Make a section visible:**

By clicking on a check box in column „visible“, you can toggle the visibility of a section.

- Give a section an individual color:

By clicking on the color entry of a section, you can select an individual color which will be applied to all points of that section. If you select “original”, every point is drawn using its own color data.

Of course the selected color does not affect the physical point cloud data, it just affects the appearance.

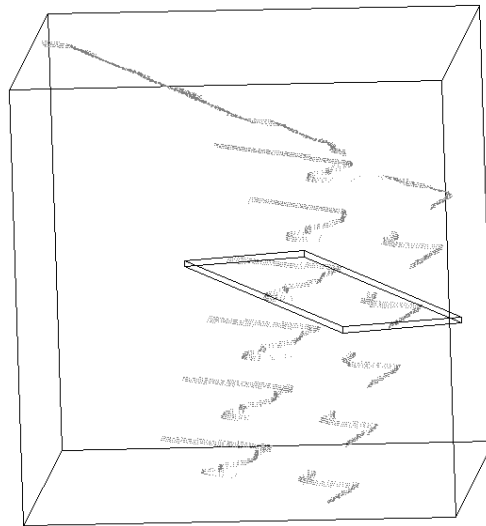
If sections overlap, the overlapping points are drawn using a merged color.


The buttons

-  Show selected section enlarged

Press this button to scale up the section marker.

This is very useful especially when the sections are small, like those eventually created by clash detection.



-  Show selected section and 3D orbit

Press this button to start AutoCAD's 3D orbit. The selected section is marked. When you exit the 3D orbit, the section manager will be restarted.

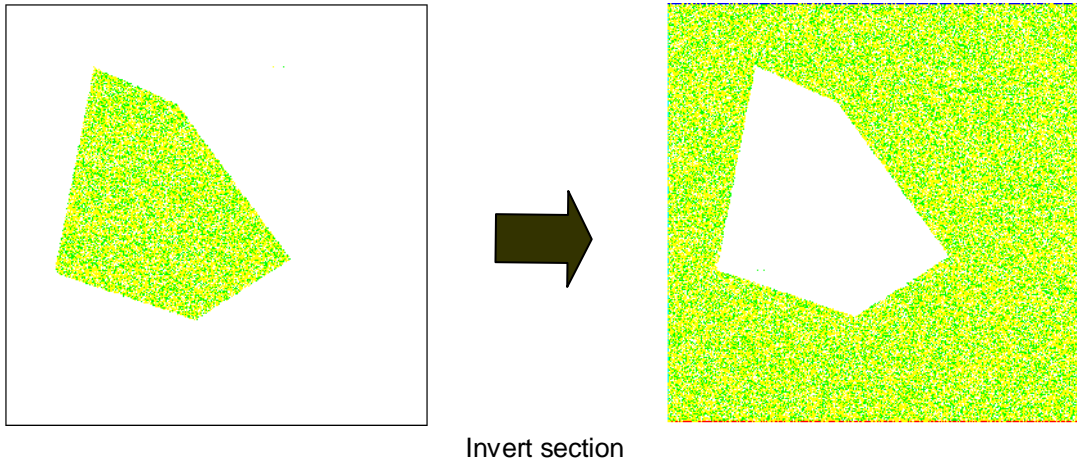
-  Create new sections – Copy:


Press this button to create copies of the selected sections.

-  Create new section – Invert:


Press this button to create a new section which is the inverse of the selected one, i.e. it contains all points which are not contained by the original section.

This button is activated only if one single section is selected.





-  Create new section – Merge (disjunction):


Press this button to merge sections into a single new section. That new section contains points which are contained by *at least one* of the sections (disjunction).


This button is activated only if at least two sections are selected.
-  Create new section – Merge (conjunction):

Press this button to merge sections into a single new section. That new section contains only those points which are contained by *all* sections (conjunction).

This button is activated only if at least two sections are selected.
-  Export section(s) as an ASCII file, PTC file or as AutoCAD points:

Press this button to export selected sections. For your convenience a wizard guides you through the export process. Available export formats are ASCII file, PTC file and AutoCAD points.
-  Delete section(s):

Press this button to delete the selected section(s). If you delete a section by mistake, cancel the Section Manager dialog to restore it.
-  Load section definitions:

Press this button to load section definitions from a file (extension PCS) and apply to active point cloud.
-  Save section definitions:

Press this button to save selected sections in a section file (extension PCS). No points are saved, only the geometric definition of sections. You may load this section file later again to apply to other point cloud(s).

-  Copy section definition:

Press this button to copy selected sections to another point cloud in the current drawing.

Select a point cloud to copy the selected sections to.

Select point cloud:

*(Select destination point cloud.
Point cloud has not to be loaded.)*

-  Create legend:

Press this button to create a legend for the selected colored sections. This is useful if you colored the point cloud sections thematically or for instance by distances or elevation. You may then create an ortho image (→Create ortho image) and the legend and hand on the result to AutoCAD users who do not have a FARO software or you may plot the plan with legend.

In the command line you are prompted for the insertion point of the legend and you have an option for setting the text height. The legend will be filled with the colors from the section manager and the section names as text. The resulting elements can be edited with native AutoCAD tools. The legend will not be updated automatically.

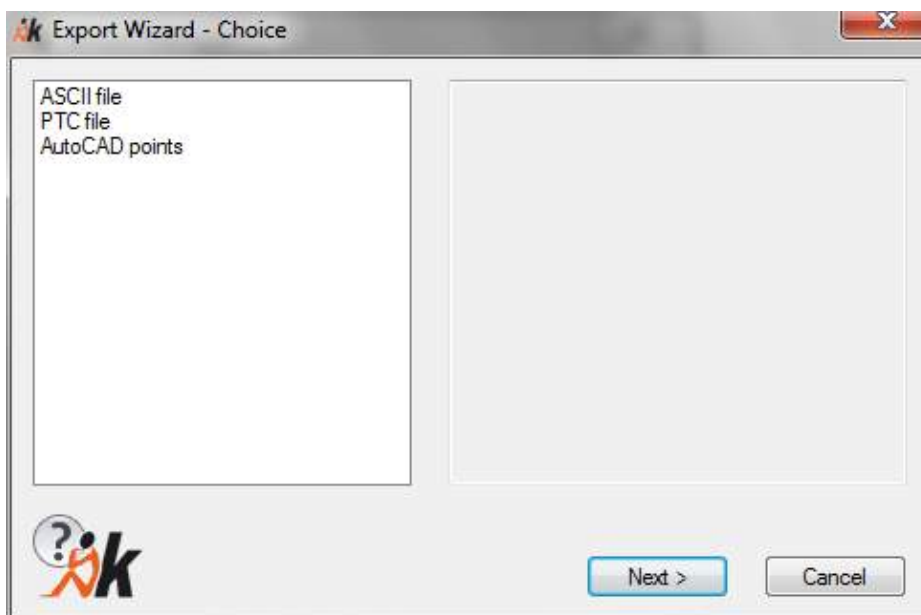
Export Wizard:

With the help of this wizard you can export the points of selected sections.

The point coordinates are exported in AutoCAD's world coordinate system so that currently or previously defined UCS's do not affect the export.

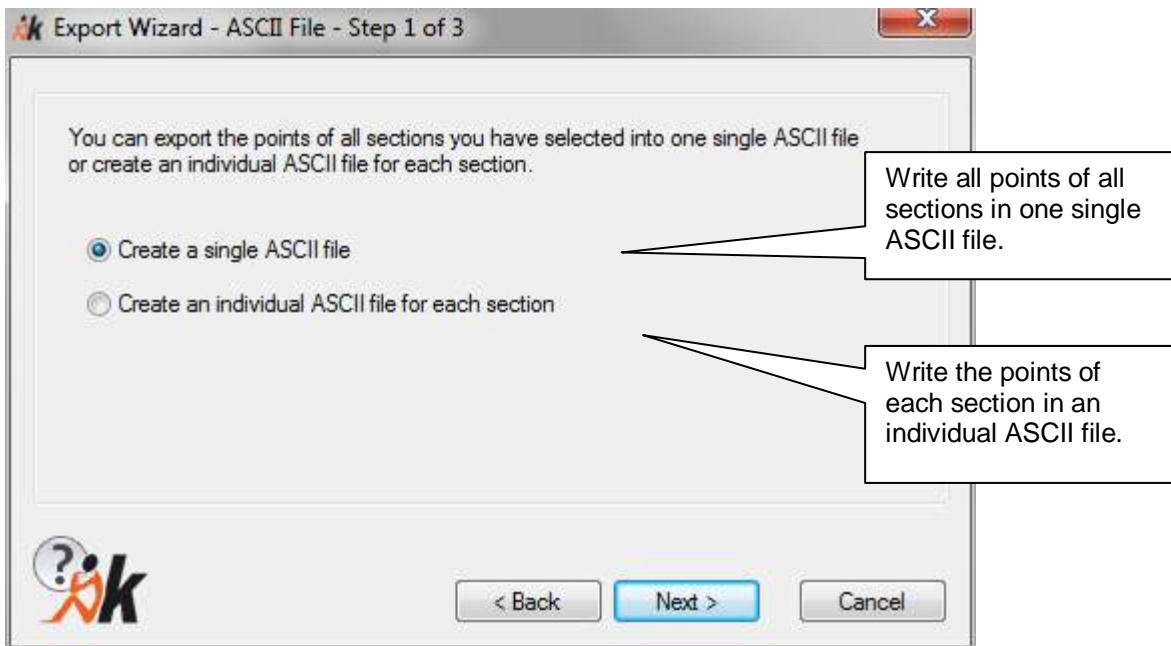
You can write the points in one or more ASCII files. You can create one or more new PTC files which can be loaded into the drawing. And you can export the points as native AutoCAD points. In the latter case you should keep in mind that AutoCAD is not able to manage millions of points. The limit is – dependent on your system configuration – about 100,000 points.

Choose an export format:

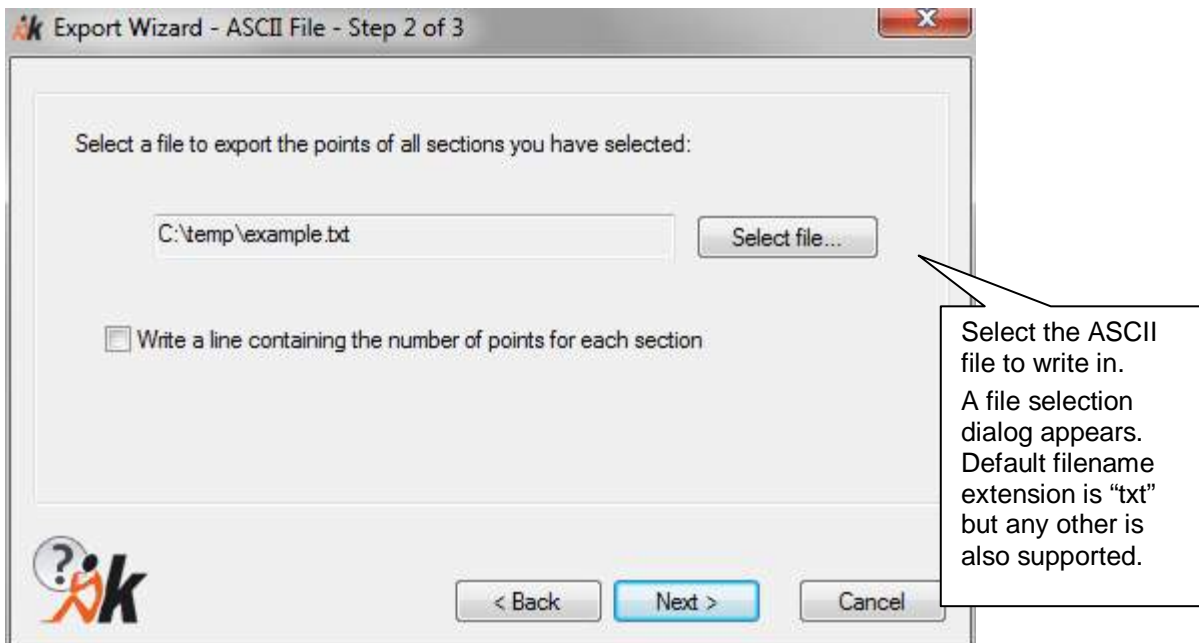


Step 1: ASCII file

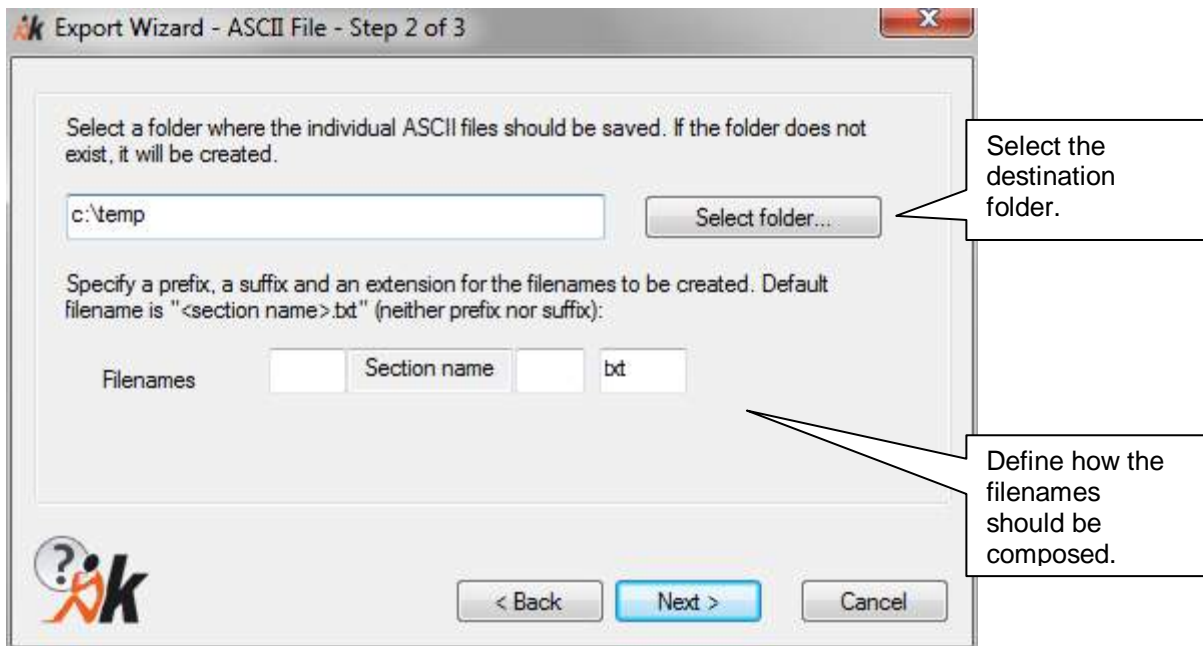
If you have selected more than one section to export, the following window will appear next:



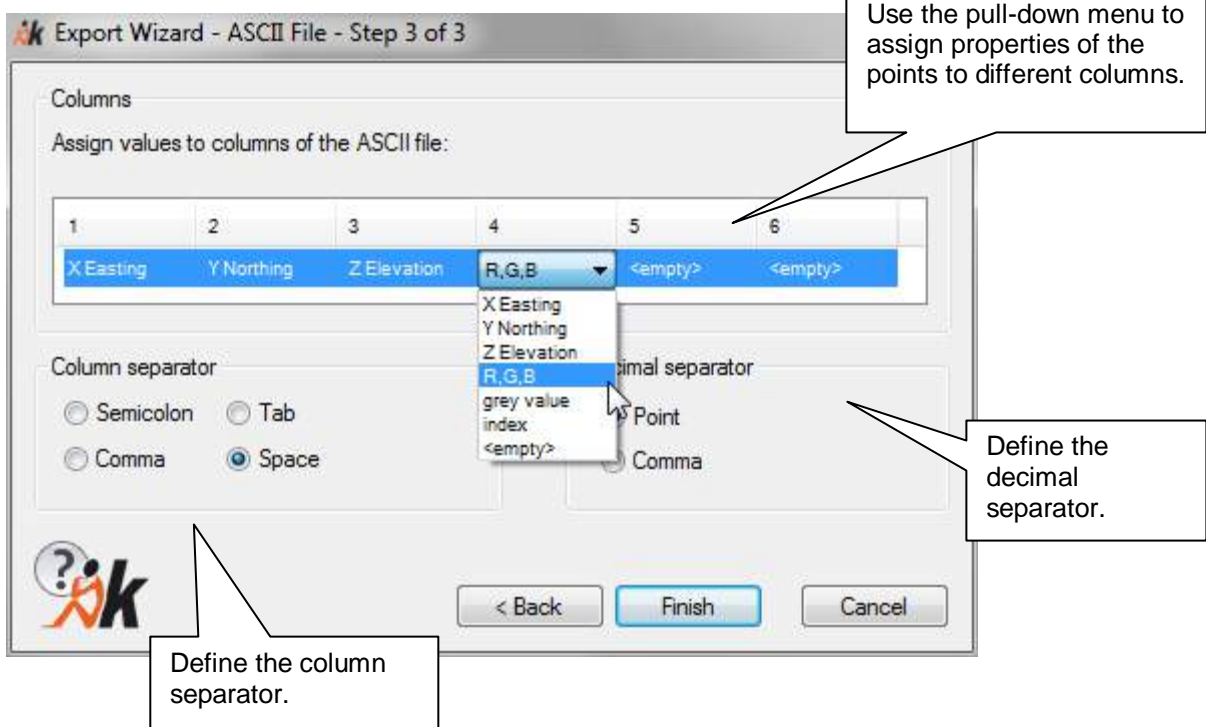
Step 2: Create a single ASCII file



Step 2: Create an individual ASCII file for each section

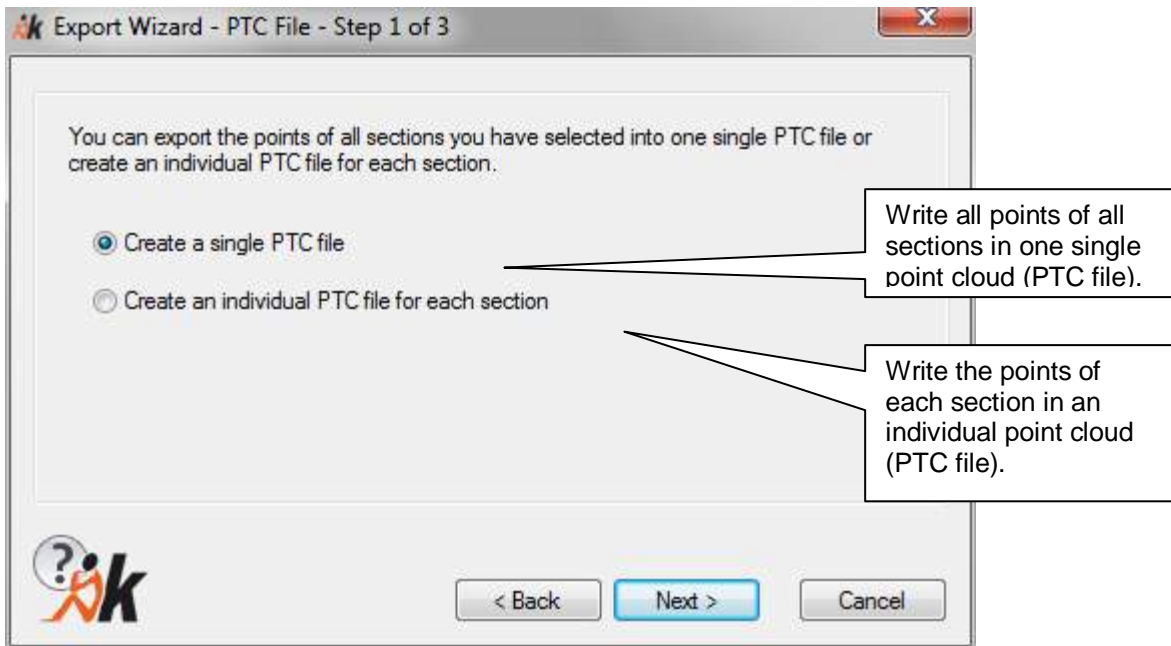


Step 3: Define ASCII file format

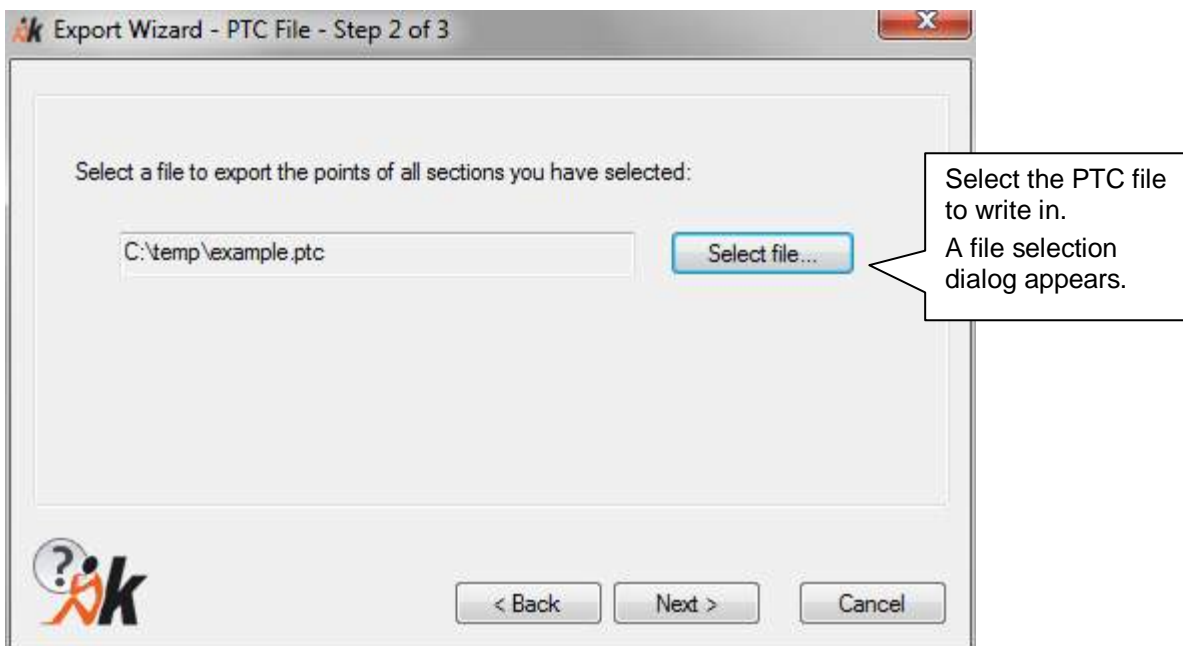


Step 1: PTC file

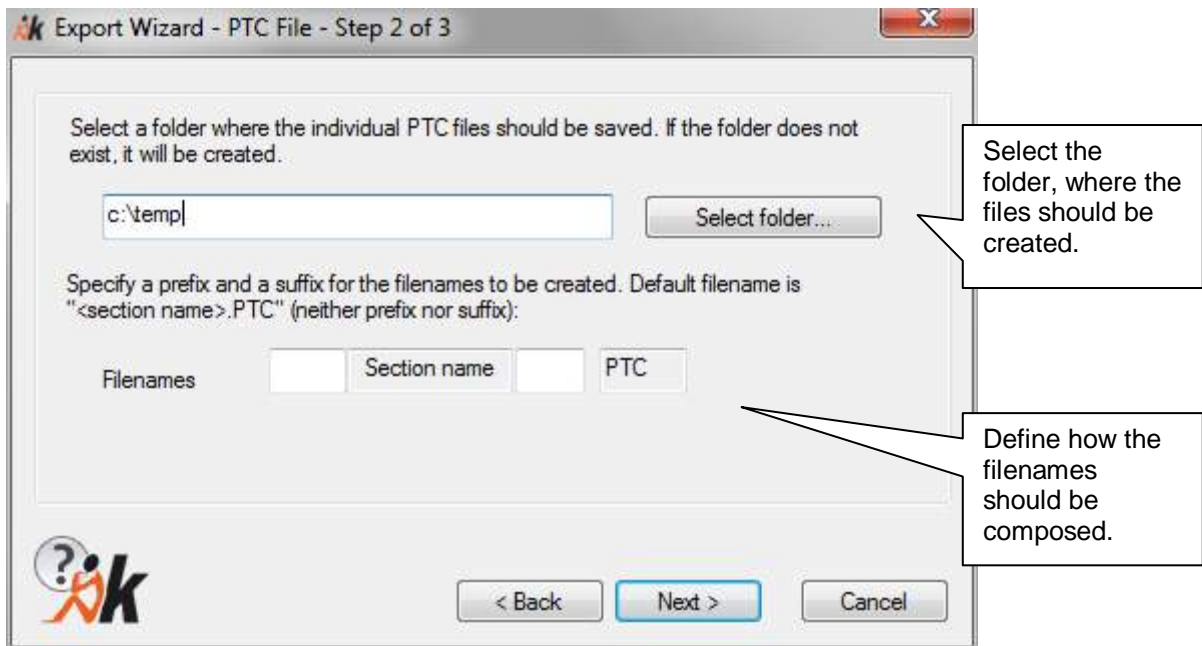
If you have selected more than one section to export, the following dialog appears:



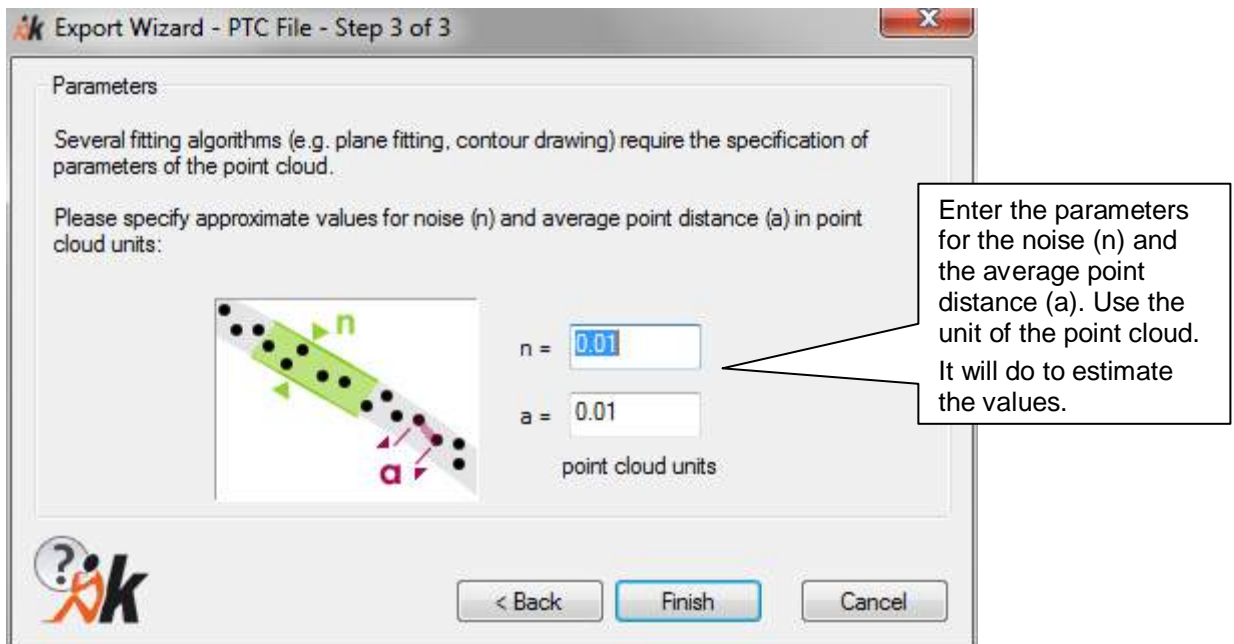
Step 2: Create a single PTC file



Step 2: Create an individual PTC file for each section



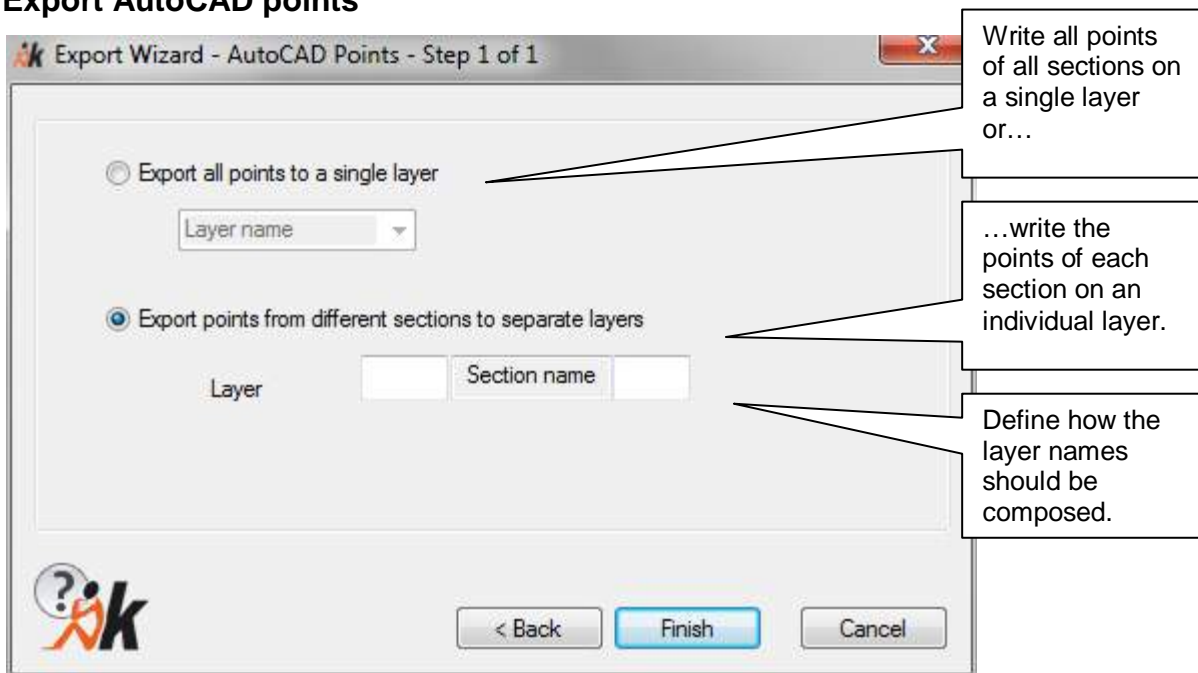
Step 3: Set parameters of the point cloud:



Parameters are saved in the PTC file. You always have the option to change them using this command → **Point clouds – Set point cloud parameters...**

The grip of the new PTC file is placed by default in the centre of the bounding box. You may change this position using the command → **Point clouds – PTC format - Set grip position.**


Export AutoCAD points



The following table gives an overview, how the color property of the sections affects the color of the created AutoCAD points and layers:

	one layer	several layers
Section has a color except from "original" (section's color)	Points → section's color	Points → color "by Layer"
		Layer → section's color
Section has "original" color	Points → original color	Points → original color

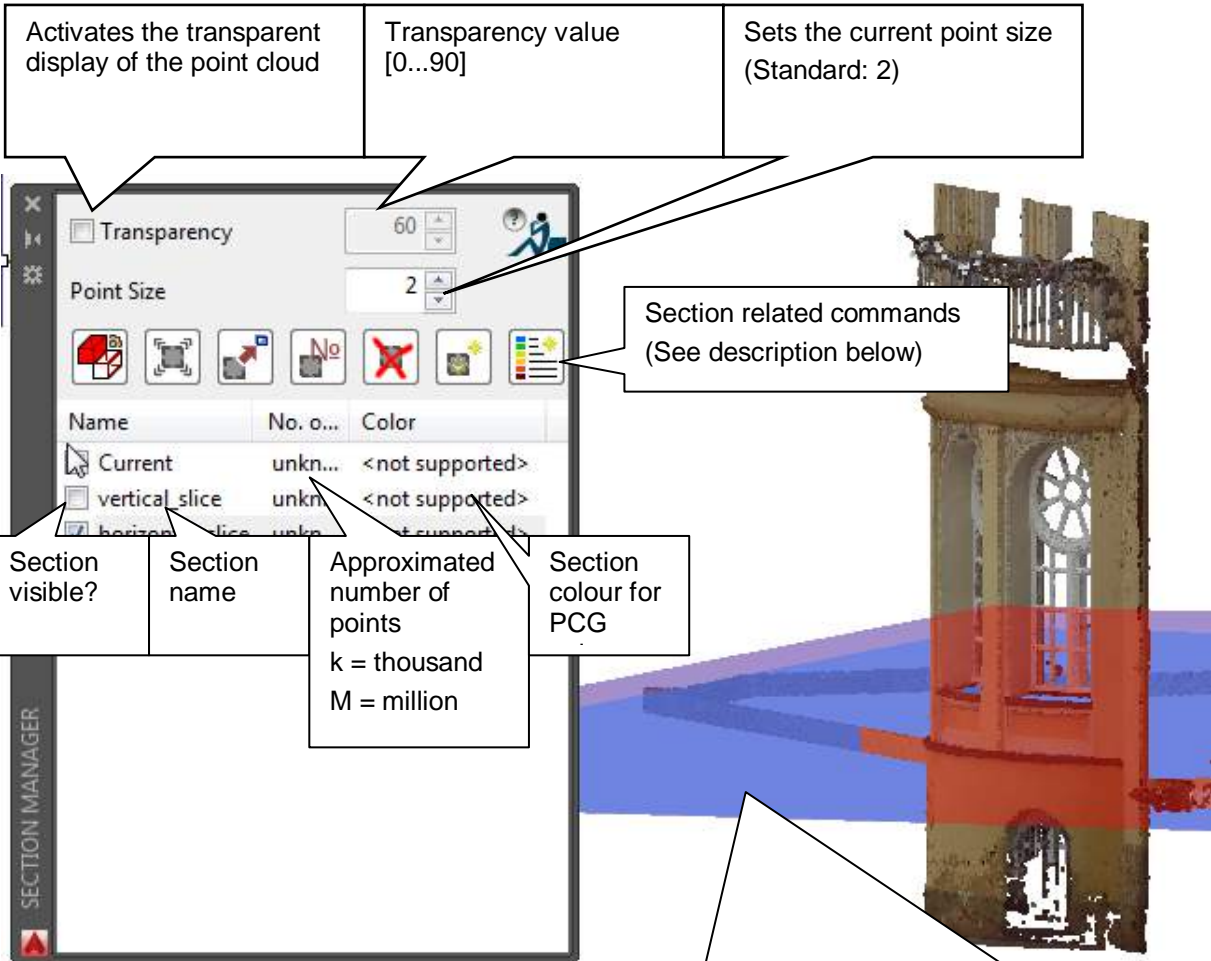
Sections – Section Manager [for Autodesk/ReCap point clouds]

	Command Name:	PCSECMAN
	Ribbon Tab:	As-Built Modeler

With this command you can show the Section Manager for AutoCAD point clouds, provided an AutoCAD point cloud is the current one.

For Autodesk point clouds we have developed a new section manager. It appears as a dockable AutoCAD palette window (similar to AutoCAD's property window or layer manager).

The buttons provide section related commands.



Activates the transparent display of the point cloud

Transparency value [0...90]

Sets the current point size (Standard: 2)

Section related commands (See description below)

Section visible?	Section name	Approximated number of points k = thousand M = million	Section colour for PCG
<input checked="" type="checkbox"/>	Current	unkn...	<not supported>
<input type="checkbox"/>	vertical_slice	unkn...	<not supported>
<input type="checkbox"/>	horizontal_slice	unkn...	<not supported>

Section Marker: The selected section will by default be highlighted with an axis-aligned half-transparent red box. To remove the box, click in the empty space of the section list.
For performance reasons the size of the box will be approximated until the number of points of the section has been calculated.

Transparency settings

The point cloud often covers, because of its density and its photo realistic display the geometry that has already been modelled. The point cloud will become transparent when the transparency tick box has been ticked. You can change the transparency value with the small arrow keys or by entering a value in the input box. The value must lie between 0 (no transparency) and 90 (maximum transparency) and is saved in the drawing.

The default value for new drawings can be set by using the configuration command (**PCCONFIG**).

Point size settings

The input box for the point size reflects the value of the AutoCAD system variable **POINTCLOUDPOINTSIZE**. You can easily adjust this value with the arrow keys or direct entry. The value can lie between 1 and 10.



Display options for the section marker. You may change the look of the section marker. By default the sections are highlighted by a half-transparent red box.

Command: Select a style for the section marking [None/Wireframe/Transparent] <Transparent>:
(Select one of the offered styles)

The options mean:

None	No section marker is displayed.
Wireframe	Only the edges of the axis-aligned box are displayed.
Transparent	The transparent lateral faces of the axis-aligned box are displayed.

If you select „Transparent“, you are prompted to enter a value for the transparency of the box.

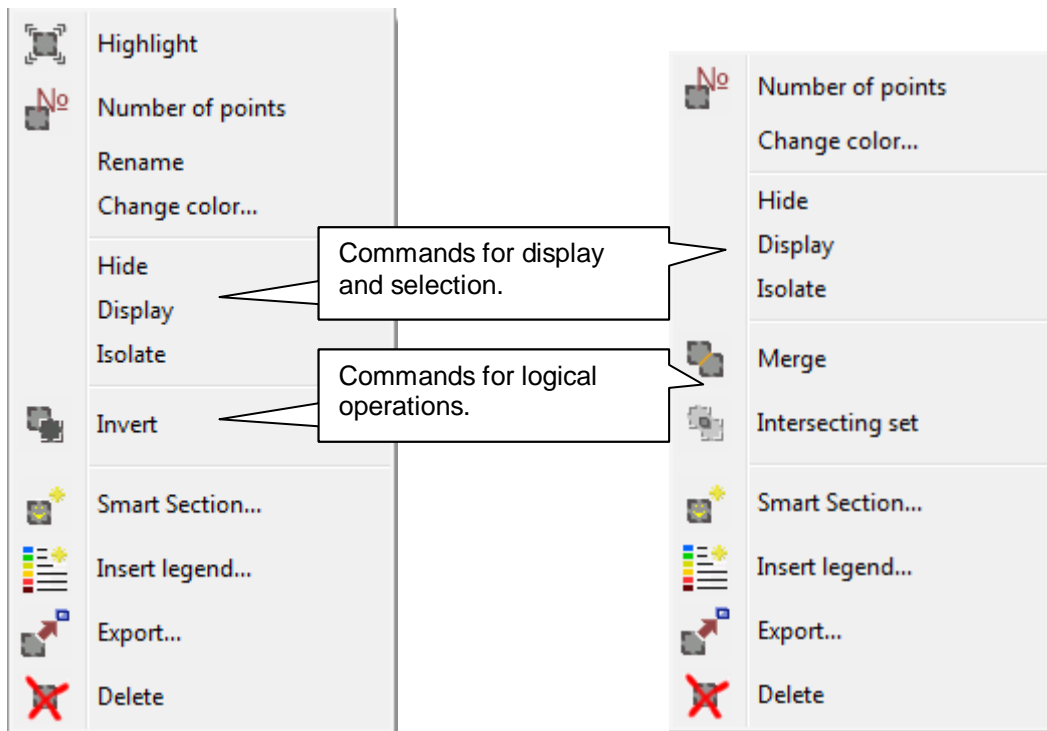
Set transparency in percent <90>:
(Select a value between 0 and 100,
A value of 100 means completely transparent)



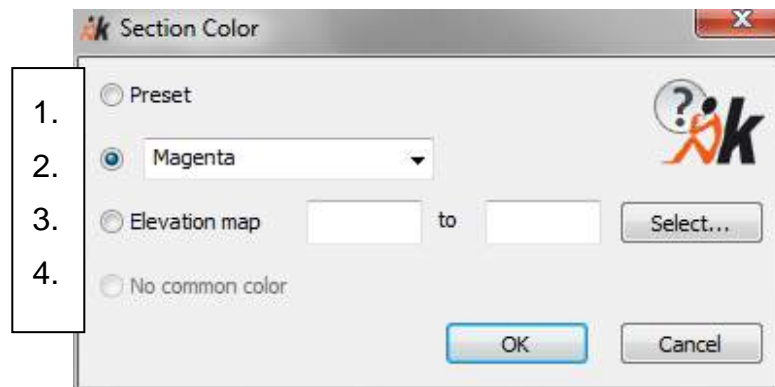
Export section(s) as an ASCII file, PTC file or as AutoCAD points:

Press this button to export selected sections. For your convenience a wizard guides you through the export process. Available export formats are ASCII file, PTC file and AutoCAD points. You find a detailed description of the export command on **page 97** and following.

By clicking with the right mouse button on one or several selected sections you open the context menu which provides additional commands depending on the section selection:

one section selected**several sections selected****Change color...**

With this command you may change the color of one or multiple sections.



1. Preset: Default color of the points (e.g. True Color, intensity)
2. Here you may select an AutoCAD color
3. Elevation map: The points of the selected sections will be colored with a red blue spectrum dependent on their Z elevation (referring to the current UCS). Points below the selected minimum elevation are displayed in red. Points above the selected maximum elevation are displayed in blue. Via the [Select...] button you may pick the values by snapping two points in the drawing.
4. No common color: If you want to change the color of multiple sections, which so far are colored differently, at first the radio button is active. You may now either define a common color for all selected sections (1.-3.) or close the dialog without changes ([OK] or [Cancel]).



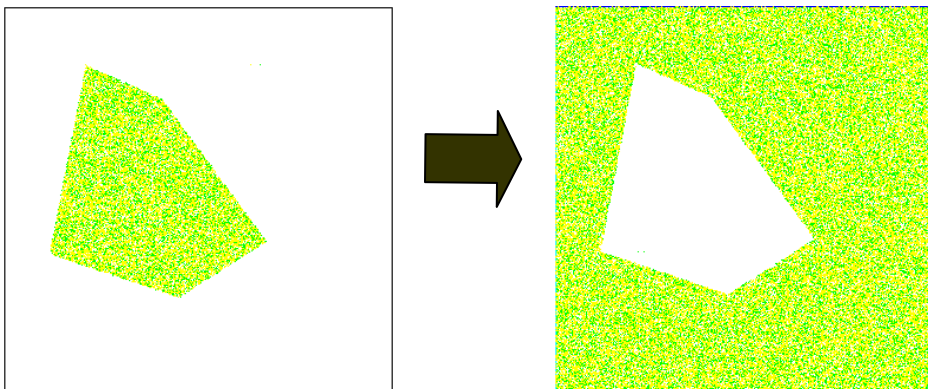
Note: Due to limitations in the new API of AutoCAD colouring of ReCap point cloud sections does not work.


For elevation colouring of a point cloud (RCP/RCS) (WCS only), please use the point cloud context menu under “Visualization”→“Scan Colours”→“Elevation“. Read the AutoCAD help file for details.

Commands for creating new sections from existing ones

-  Create new section – **Invert**:

Use this command to create a new section which is the inverse of the selected one, i.e. it contains all points which are not contained by the original section. This button is activated only if one single section is selected.



-  Create new section – Intersecting set (disjunction):

Use this command to merge sections into a single new section. That new section contains points which are contained by *at least one* of the sections (disjunction). This button is activated only if at least two sections are selected.

-  Create new section – Merge (conjunction):

Use this command to merge sections into a single new section. That new section contains only those points which are contained by *all* sections (conjunction). This button is activated only if at least two sections are selected.

Creating a Smart Section

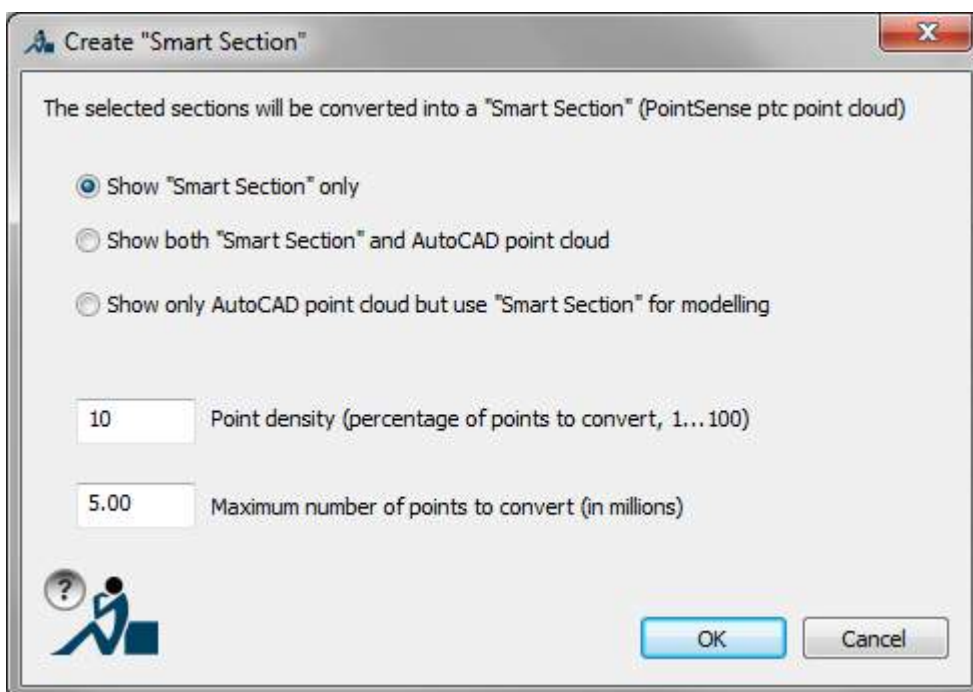


With this command you can create a so called **Smart Section**. A Smart Section is a section of an AutoCAD point cloud which is temporarily converted into a point cloud in PTC format for a better display. This point cloud is not saved in the drawing. For a comparison of both formats and their advantages and disadvantages please refer to chapter 3.2 of the manual.

Besides the better display, above all other advantage the **proper object snap** is what makes the conversion of a section into a Smart Section sensible. A proper object snap which only snaps points which lie in front in the current view is essential for an accurate evaluation of details. Furthermore the conversion of sections into the PTC format enables the utilization of features like clash detection and the 3D drawing which so far are not working with AutoCAD point clouds.



Click on the button and following dialog box opens:



The Option “*Show ‘Smart Section’ only*” is the default option. We recommend using this because only this display method allows for proper object snap.

The Option “*Show both ‘Smart Section’ and AutoCAD point cloud*” allows a higher-detail section of a an point cloud while still being able to display all other points in the native format. This way you keep the overview of the whole project.

The Option “*Show only AutoCAD point cloud but use ‘Smart Section’ for modelling*” is beneficial during modelling when performance is essential and the object snap is not used.

Besides the described options you may define the number of points to be converted into the temporary PTC file by their percentage. And you may enter a maximum number of points. These settings are necessary since you cannot load as many points in PTC format as you can in native AutoCAD format. (see chapter 3.2) The

maximum number of points for PTC depends on the performance and RAM of your machine.


Note: The PTC point cloud created for the Smart Section is not saved in the drawing. If a Smart Section exists in the drawing the section manager is disabled. You may enable it by deleting the Smart Section. But you can also return to the original point cloud data at any time by selecting the command **→Set current point cloud** and thus enable the section manager again.

Create a legend



A click on this button starts the **→Create Legend** command (PCDISTCOLLEGEND). With this you can create a legend for RCP/RCS point clouds. The legend is either based on a customized As-Built color scheme (**→PCDISTCOLEEDIT**) or on the colors of the current AutoCAD visualization according to elevation.

ScanNav – Create ScanLabel

	Command Name:	PCSCANLABELCREATE
	Ribbon Tab:	As-Built Modeler

With this command you create the markers for the scanner positions (*ScanLabel*), which are meant for controlling the single scan navigation (*ScanNav*).

Command: **PCSCANLABELCREATE**

Create ScanLabels for ReCap point clouds in the drawing, import from VirtuSurv scan positions file or specify points [Recap/Virtusurv/Point] <Recap>:

(Select one of the options)

ReCap :

Please note that this option is only available for ReCap point clouds!

It will create ScanLabels for all ReCap point clouds (RCP or RCS) in the drawing, if they do contain information about the single scans. The ScanLabels are then linked with the ReCap scans.

VirtuSurv :

A dialog opens where you may select a scan position file, which you previously exported from VirtuSurv (see figure below). ScanLabels will be created for all scan positions and they will be linked with the corresponding VirtuSurv scans.



When exporting scan positions from VirtuSurv please ensure the „Measure Unit“ in VirtuSurv matches your drawing unit As-Built.

Point :

Specify points for the scan positions (by clicking, measuring or entering coordinates).

Specify scan position: *(specify the position of the first scan)*

Specify scan position: *(specify the position of the second scan)*

...

(finish by ESC or ENTER)

Finally the ScanLabels are created and you get a brief message in the command line:

```
12 ScanLabel(s) created. 0 updated. 0 failed.
```

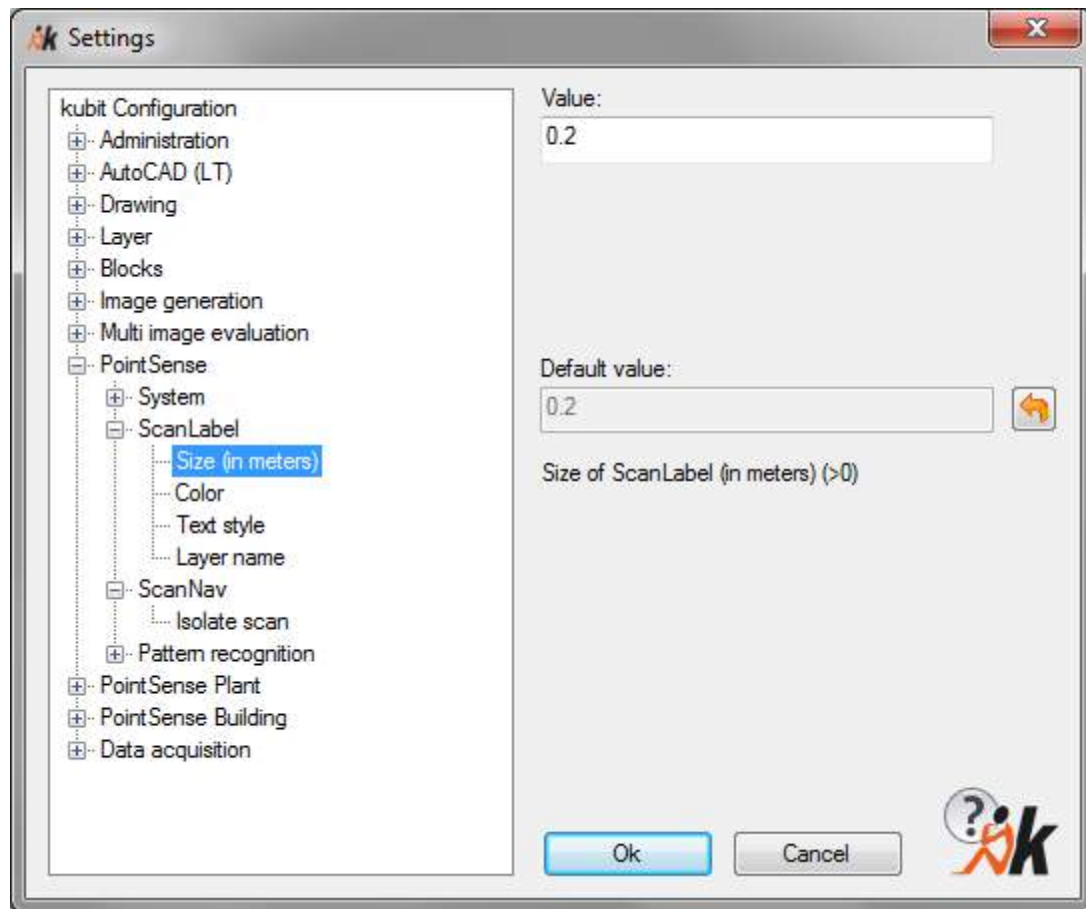
If there already exists a ScanLabel at that particular scan position it will be updated. Thus, you may for instance create the ScanLabels for the ReCap scans that are inserted in the drawing and then import a corresponding VirtuSurv scan position file. The ScanLabel is then linked with the ReCap scan as well as with the VirtuSurv scan.

ScanLabels are located on a separate layer (Default: KUBITSCANLABEL).




Fig. 1 ScanLabel with ReCap and VirtuSurv link

The layer for the ScanLabels as well as further parameters for their visualization and the behavior of the navigation commands can be configured in the configuration dialog (see figure below., command: **PCCONFIG**).



ScanNav – Scan navigation on

	Command Name:	PCSCANNAV
	Ribbon Tab:	As-Built Modeler

Select the ScanLabel for which you want to enable the scan view and navigation in the current view port.

The previous view will be saved and restored after turning the scan navigation off (command: **ScanNav – Scan navigation off**).

Command: **PCSCANNAV**

Select ScanLabel:

(Select the ScanLabel you want to enable the scan navigation for)

To be able to more easily select, or to make it even possible to select, the ScanLabel the whole point cloud is made “transparent”. The more distant ScanLabels will be otherwise hidden by the point cloud.

After selecting the ScanLabel the respective scan will be isolated, as long as it is associated with a ReCap scan. Displaying one single (isolated) scans in a scan view is much clearer than showing multiple scans. This is because there is no occlusion and the colors of all displayed points are consistent.

The settings to isolate and to temporarily activate the transparency can be adapted in the configuration dialog (**PCCONFIG** command, As-Built → ScanNav → “Isolate scan” or “Temporary point cloud transparency”).



Fig. 1 Comparison between scan view with isolated scan (left) and multiple visible scans (right)

A perspective view will be created for the current view port with the camera position at the position of the ScanLabel. The default direction of view is the Y axis of the WCS.

Scan navigation will be enabled.

Scan navigation

In scan navigations mode you may (almost) navigate as your are used to with your mouse or the camera navigation commands:


	Mouse	Command
Zoom	Scroll mouse wheel	CamNav Zoom
Pan	Hold down mouse wheel and move the mouse	CamNav Pan
Twist direction of view	CTRL + holding down the mouse wheel and moving the mouse up and down	CamNav Twist

When scan navigation is active for a view port all other AutoCAD navigation commands (like ZOOM, 3DORBIT, PAN,...) as well as commands that change the view are disabled. Editing of all AutoCAD system variables that may change the view in an inadmissible way is also prevented.

Scan view and scan navigation mode are stored in the drawing and are active when opening the drawing the next time.

Hint: You may also start the scan navigation by double clicking on a ScanLabel. This way you may also quickly go to the next scan position while you are still in scan navigation mode. You can do the same via the context menu (right mouse button click) of the ScanLabel.

ScanNav – Scan navigation off

	Command Name:	PCSCANNAVOFF
	Ribbon Tab:	As-Built Modeler


This command turns off the scan navigation for the current viewport and restores its last view before scan navigation.

command: **PCSCANNAVOFF**

The current parameters of the scan view are saved and will be applied when activating ScanNav next time.

If the according scan has been isolated when activating the scan navigation, this change will now be set back that means scans that have been visible before scan navigation will now be visible again.

ScanNav – Open linked VirtuSurv scan

	Command Name:	PCSCANNAVVS
	Ribbon Tab:	As-Built Modeler


This command opens the linked scan in the external application VirtuSurv.

commands: **PCSCANNAVVS**

*(Select a ScanLabel that is linked with a
VirtuSurv scan.)*

VirtuSurv opens and the scan linked with the selected ScanLabel is displayed.

ScanNav – Specify direction of view for ScanNav

	Command Name:	PCSCANNAVTARGET
	Ribbon Tab:	As-Built Modeler

You may define the direction of sight for the scan view of one or more ScanLabels (scan positions).

Command: **PCSCANNAVTARGET**

Specify target point for ScanNav view. Select ScanLabels


Select objects: *(Select the ScanLabels for which you want to define the direction of sight. Confirm the selection with ENTER)*

Specify target point: *(Pick the desired target point for the scan view.)*

The target point has been set for 2 ScanLabel(s). 0 failed.

When activating the scan view for these ScanLabels the next time, the direction of sight will be the direction to the specified target point.

Digitizing tools – SmartSnap

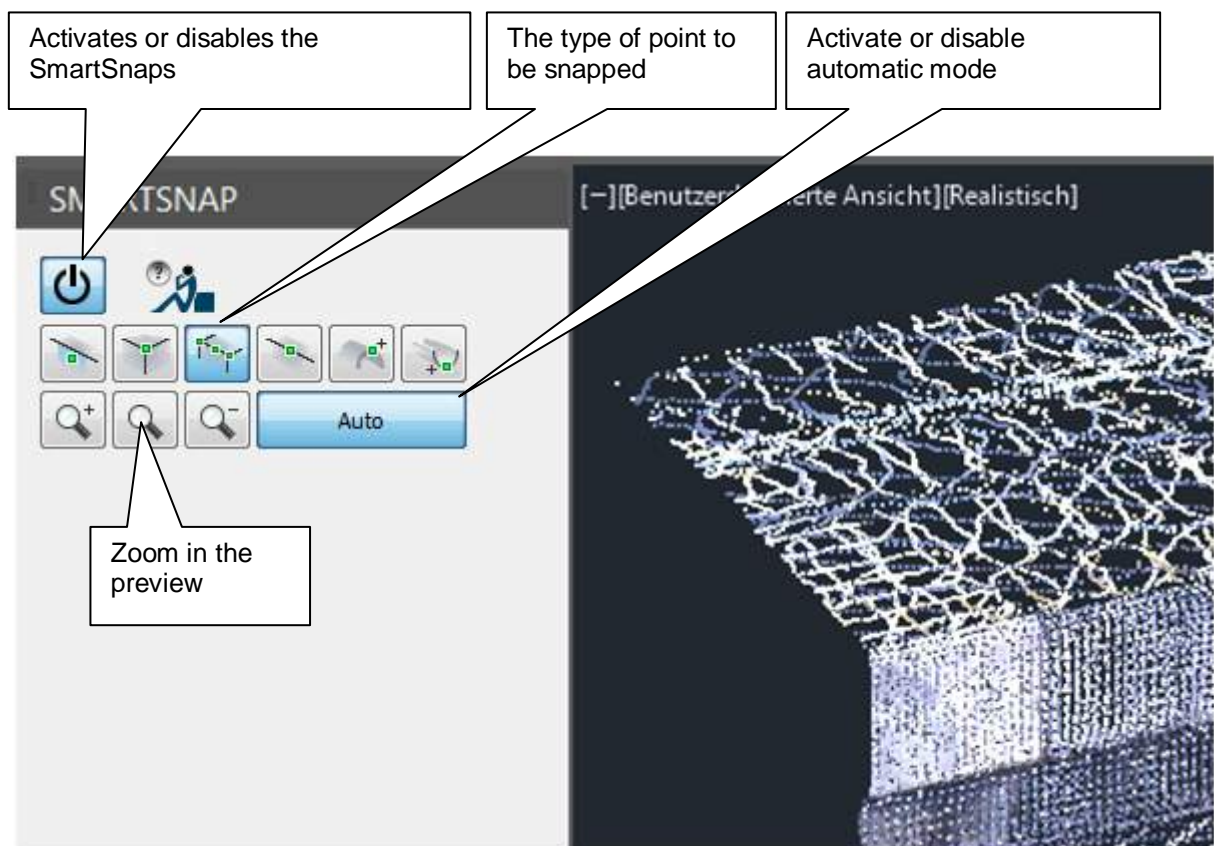
	Command Name:	PCSM
	Ribbon Tab:	As-Built Modeler

In AutoCAD objects such as lines or solids can be precisely placed by using OSNAP, which then allows you to accurately create new construction objects. The SmartSnap concept has been developed to allow, in the same way, the snapping to corners, edges or planes in point clouds.

This command displays the SmartSnap palette.

The palette can always be kept open (similarly to the properties window or layer manager in AutoCAD).

The buttons are organised in three rows.



The button for the activation or disabling of the SmartSnaps is to be found in the first row.

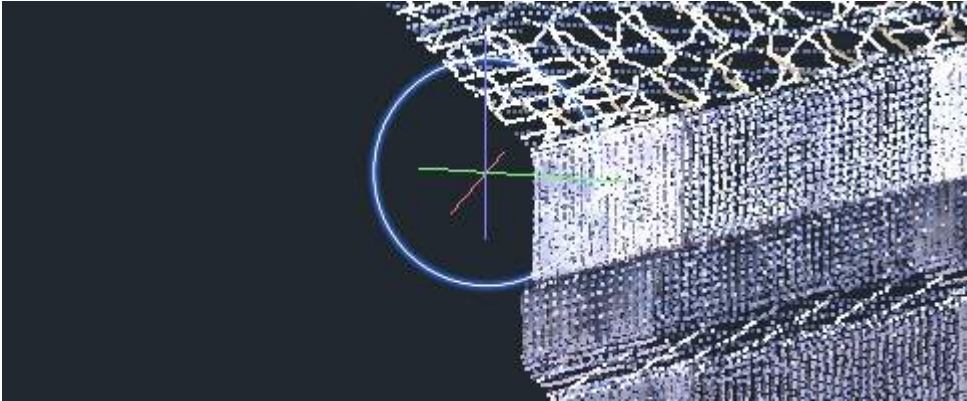
The second row contains the options you can choose from to define the type of point to be used:



- Point on a plane
- Corner point
- Corner point or a point on a straight line or an edge

- Point on a straight line or edge
- Highest point
- Lowest point

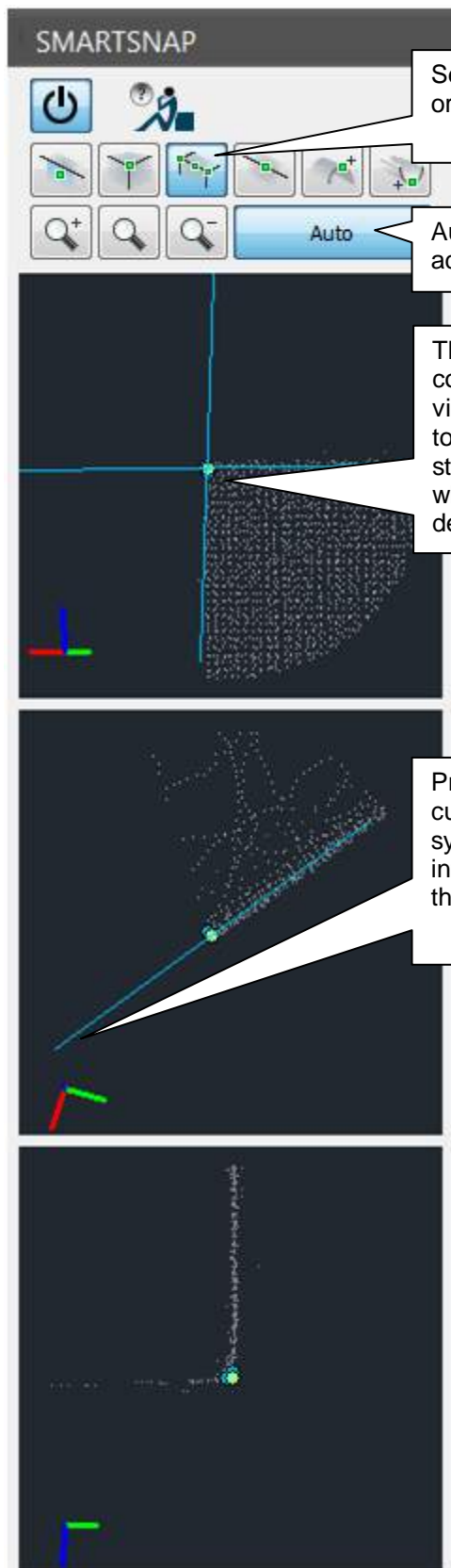
Should a command be used that expects a coordinated point (e.g. LINE) then a circle appears around the cursor. These freely selectable drawing commands will be hereafter referred to as a “Context command”.



The circle visualises the area in the point cloud that will be analysed. The size of the selection area can be changed by using **Shift + Scroll wheel**. The drawing command can be cancelled by pressing the <Esc> key. The selection circle then disappears.

SmartSnap works in two modes. In automatic mode the snapped point is immediately passed to the context command. Otherwise the snapped point must be confirmed.

SmartSnap: Automatic mode activated



Search for corner or edge points

Automatic mode is active

The computed corner point is visualised together with the straights, from which it was defined

Projection in the current coordinate system for a better interpretation of the images

A left click triggers the SmartSnaps for the selection area. Thereby the previously defined selection area will be analysed and searched for points of the selected type. Should a point of the selected type be found, then it will be automatically passed to the context command (e.g. the LINE command) without any user interaction.

In the SmartSnap window images will be shown, for a visual control, that correspond to the sections of planes used or the returned point found.

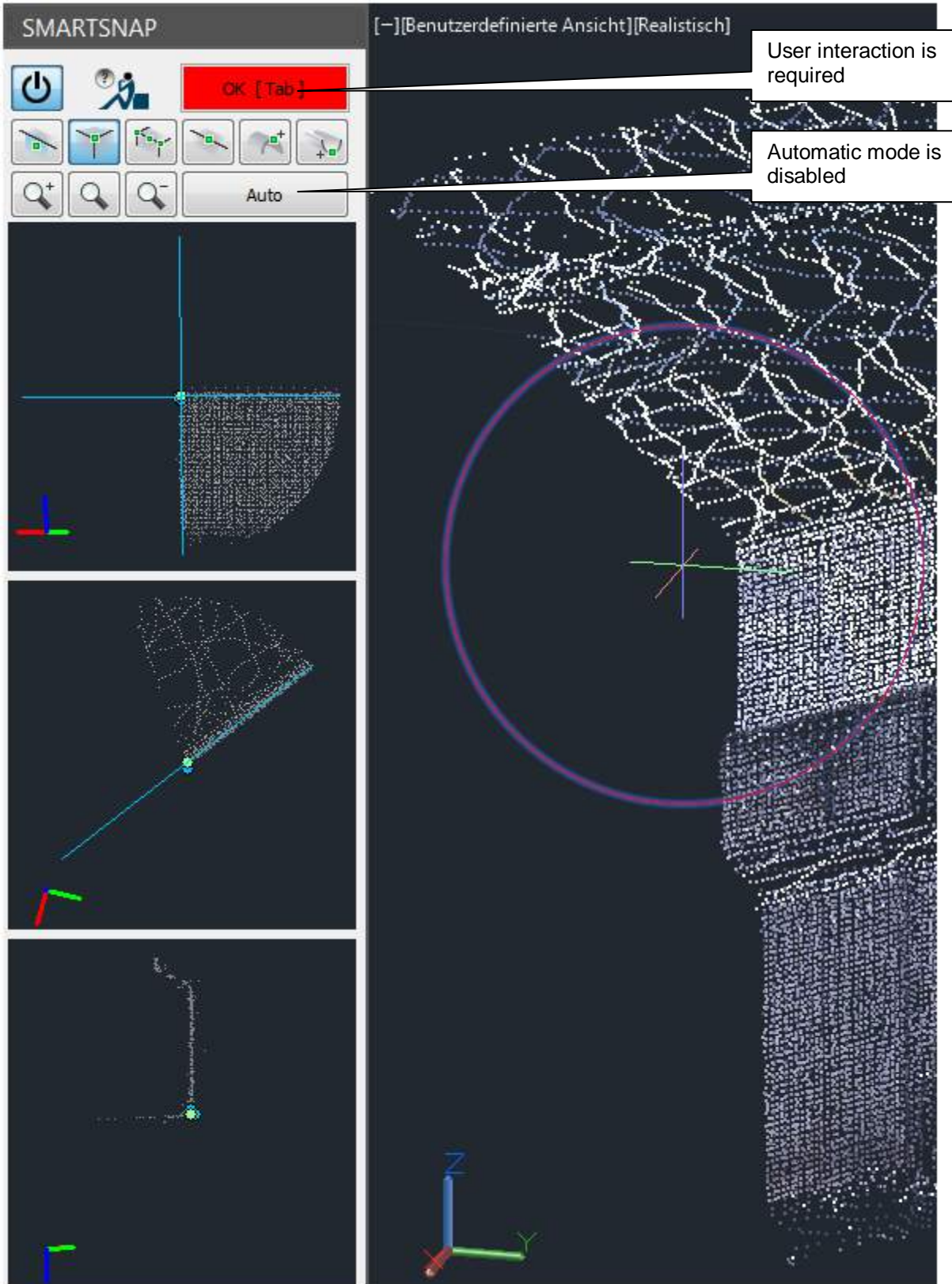
If the point is incorrect or the wrong point type was inadvertently selected, the “Undo” of the context command can be used to reverse the action of the last click.

The images serve as a visual control. The “Zoom” button can also be used.

In automatic mode you don't have the opportunity of selecting another computed point. To achieve this the automatic mode must be disabled. Without the automatic mode more user interaction is required, that means each point must be confirmed and also confirmed after each correction.

SmartSnap: Automatic mode disabled

If the automatic mode is disabled the [OK] button is displayed for confirming the computed point. User interaction is awaited after each click. This is indicated by the **change in colour** of the [OK] button and the circle.




If the search for selected point has not been performed, the point type can be changed or selected again. Furthermore the user can select another point in the images, whereby he clicks on one of the blue coloured alternative points offered (the selected point is green), and then confirms it with [OK].

Should no point be found, that corresponds to the selection criteria, then the user can also not select anything. In this case the user is required to click again.

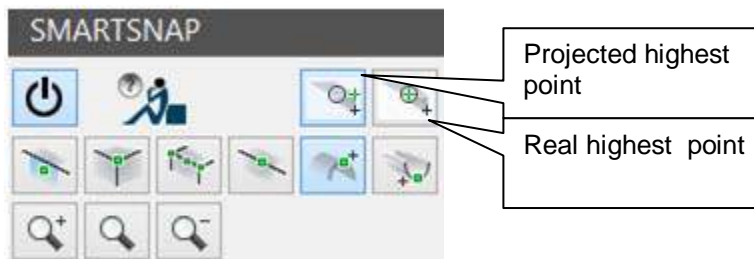
The previously selected point can also be confirmed by pressing the tab key.

SmartSnap: Active/Disabled

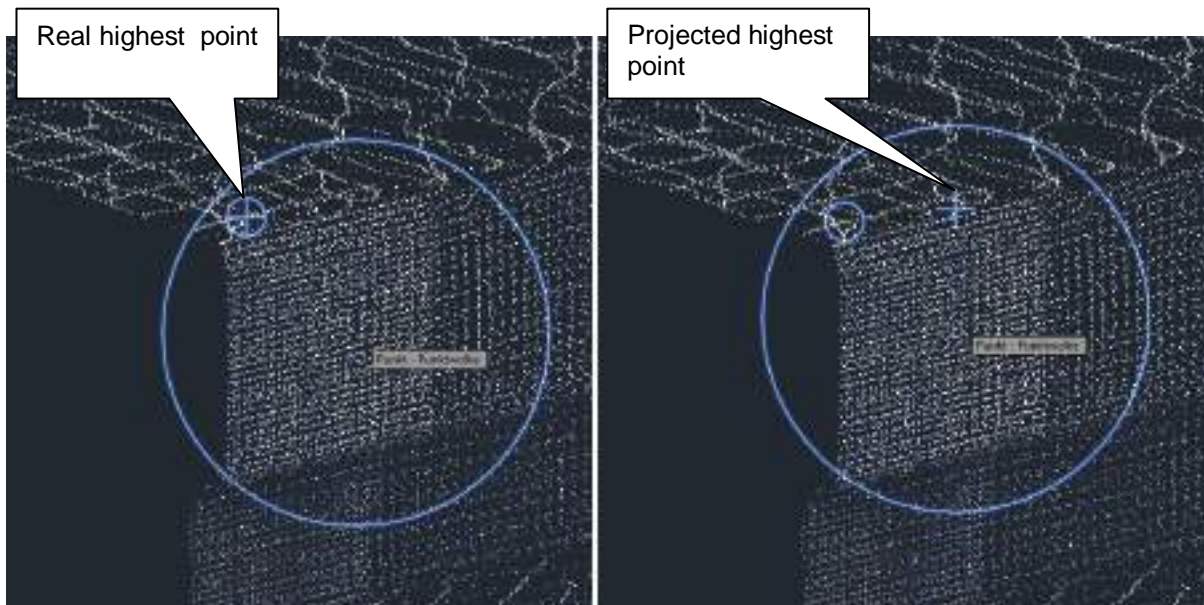
By clicking this button  the SmartSnaps can be generally disabled. The SmartSnaps can be activated or disabled at any point in time before, during or after a command. For example, if the start point of a line is to be selected using AutoCAD point snaps and all further points are to be defined using SmartSnaps, then before selecting the first point switch off SmartSnaps and afterwards turn them back on. For faster activation and deactivation of the SmartSnap mode the shortcut [Caps Lock] may be used.

Highest or lowest point


If SmartSnap is used together with the point option highest or lowest point then no preview image will be displayed. The resulting point will be displayed directly in the selection area, when the cursor is moved over the point cloud. One click confirms the point. The highest or lowest point is determined using the Z values of the current user coordinate system. Two options to determine the highest or lowest point are provided within the SmartSnap user interface.



The first option offers the possibility to project the Z value of the real highest or lowest point onto the mouse position. This results in a point containing the X and Y coordinates of the mouse position and Z coordinate of the real highest or lowest point. With the second mode the real highest or lowest point is evaluated.

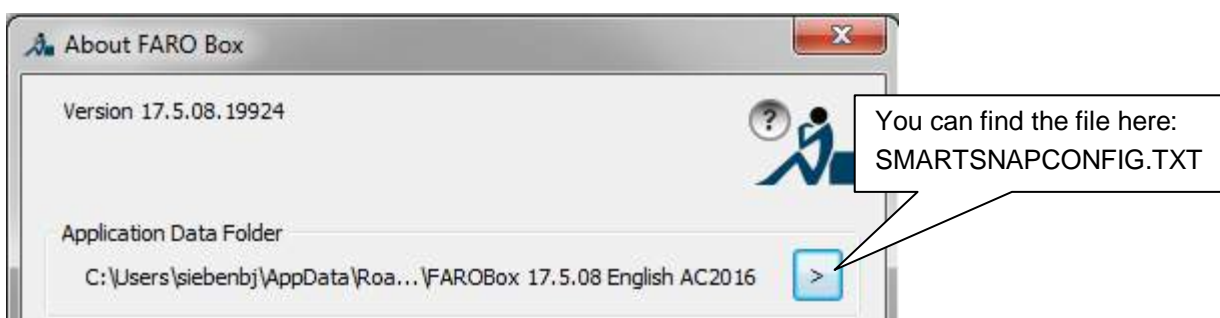


Quitting the SmartSnap mode

To just temporarily disable SmartSnaps the  button can be clicked or [Caps Lock] can be used as shortcut. To completely finish using the SmartSnaps, close the SmartSnap window.


To exclude specific commands from using SmartSnaps

If SmartSnaps are activated, then each AutoCAD command that requires a point will trigger a SmartSnap. Exceptions are all of the commands that are defined in the SMARTSNAPCONFIG.TXT file.



It can happen, when you are working with SmartSnaps, that snaps are activated for some commands that you would wish to have differently. You can add such commands to the SMARTSNAPCONFIG.TXT file to generally disable SmartSnaps for the command.

Digitizing tools – UCS from slice

	Command Name:	PCUCSSLICE
	Ribbon Tab:	As-Built Modeler


With this command you can define an AutoCAD user coordinate system, which is aligned with the currently visible slice.

The UCS for slices, that are not parallel to the XY plane of the WCS, is defined as follows:

- The Z axis points in the direction of the normal of the slice's center plane
- The Y axis points as far as possible to the Z axis of the WCS (into the direction that forms the smallest angle with the Z axis of the WCS)
- The X axis points preferably to the right (referring to the current AutoCAD viewing window).
- The UCS origin is the point closest to the WCS origin on the slice's center plane and the XY plane of the WCS.

With slices that are parallel to the WCS XY plane only the origin of the UCS will be moved into the center plane of the slice.

Digitizing tools – UCS from plane

	Command Name:	KPUCS
	Ribbon Tab:	As-Built Modeler

With this command you can define an AutoCAD user coordinate system which is aligned with a kubit plane.


Select a kubit plane: *(Select a plane for the UCS definition)*

The UCS for planes, that are not parallel to the XY plane of the WCS, are defined as follows:

- The Z axis points to the direction of the plane normal
- The Y axis points as far as possible to the Z axis of the WCS (into the direction that forms the smallest angle with the Z axis of the WCS)
- The X axis points to the right (referring to the current AutoCAD viewport) as well as possible.
- The UCS origin is the point closest to the WCS origin on the plane.

With planes that are parallel to the WCS XY plane only the origin of the UCS is moved into the supporting plane.

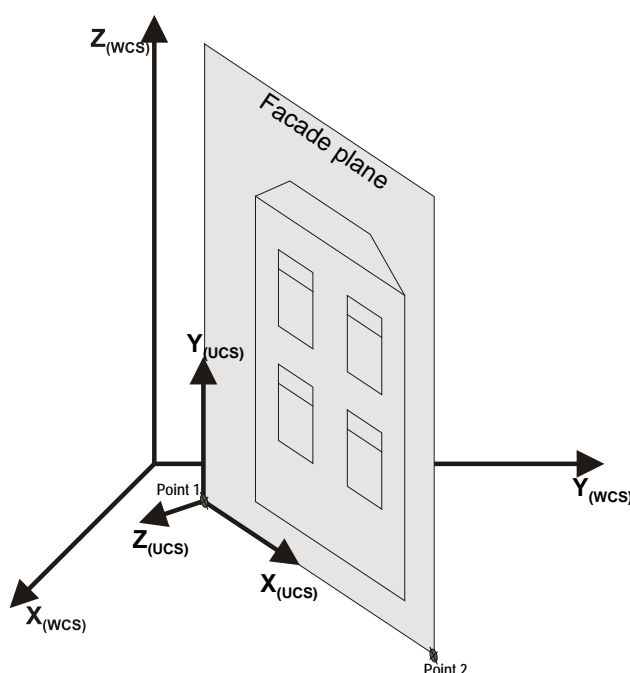
Digitizing tools – Vertical UCS

	Command Name:	PSBVUCS
	Ribbon Tab:	As-Built Modeler

With this command you can define a vertical User Coordinate System (UCS) with two points. Vertical means that the XY plane of this UCS runs parallel to Z axis of the World Coordinate System (WCS), as needed to construct vertical sections or views.

Two points are needed for the alignment of the UCS. The direction from the first to the second point defines the X axis of the UCS, which always runs through the XY plane of the WCS. Thus, it is also determined from you side you 'look' at the plane since the X axis always points to the right.

The coordinate origin is set to the first point of the new UCS.



Command: PSBVUCS

Vertical view (UCS) through 2 points

The direction from first to second point defines the X-axis.

Select first point for X-axis:

(Click or enter 1st point)

Select second point for X-axis:

(Click or enter 2nd point)


UCS name: **View East facade**

(Enter name)

After the command has been ended, TachyCAD automatically changes to the top view of the new UCS.

With the help of the default name, you can easily switch back to this UCS later. Use the command 'Named UCS' from the AutoCAD menu "Tools" (or type UCSMAN in the command line).

Digitizing tools – UCS from image


	Command Name:	PCSETUCSFROMIMAGE
	Ribbon Tab:	As-Built Modeler

With this command it is possible to establish a UCS, with the XY plane matching the image plane exactly. The coordinate origin is in the bottom left corner of the image.

Optionally it is possible, to switch to the plan view of this image, if you answer "yes" to the following question in the command line:

Zoom to image? [Yes/No] <Yes>:

Digitizing tools – 3D orbit (with center of rotation)

	Command Name:	3DORBITCTR
	Ribbon Tab:	As-Built Modeler


Via this button you start the native AutoCAD command 3DORBITCTR. It is a very useful tool when working with point clouds. You can use it transparently.

In the AutoCAD help file it is described as follows:

3DORBITCTR starts 3D Orbit view and uses a center of rotation that you specify with your pointing device. If you specify a point outside the current view, 3DORBITCTR ignores the specified point and uses the default center of rotation.

3DORBITCTR overrides the AutoTarget option in the 3DORBIT command.

Digitizing tools – Set elevation (from point)

	Command Name:	PCELEVPOINT
	Ribbon Tab:	As-Built Modeler

With this command you can easily set the AutoCAD system variable ELEVATION to a value you snap from a point of the point cloud. That allows you to digitize on a plane parallel to the XY plane of the current UCS at the depth you selected. This helps to simplify e.g. digitizing of spatial structured facades.

Select a point. Z value of the selected point defines AutoCAD's elevation property.


Point for AutoCAD elevation property or [Reset] elevation to zero:

(Select any point or "R" for reset. OSNAP is being activated)

ELEVATION was set to <VALUE>.

After having set the ELEVATION all lines and other AutoCAD objects are drawn with the corresponding z value. Usually the command is used to draw at a certain elevation without OSNAP "Point".

Digitizing tools – Align cursor

	Command Name:	PSBALIGNCURSOR
	Ribbon Tab:	As-Built Modeler

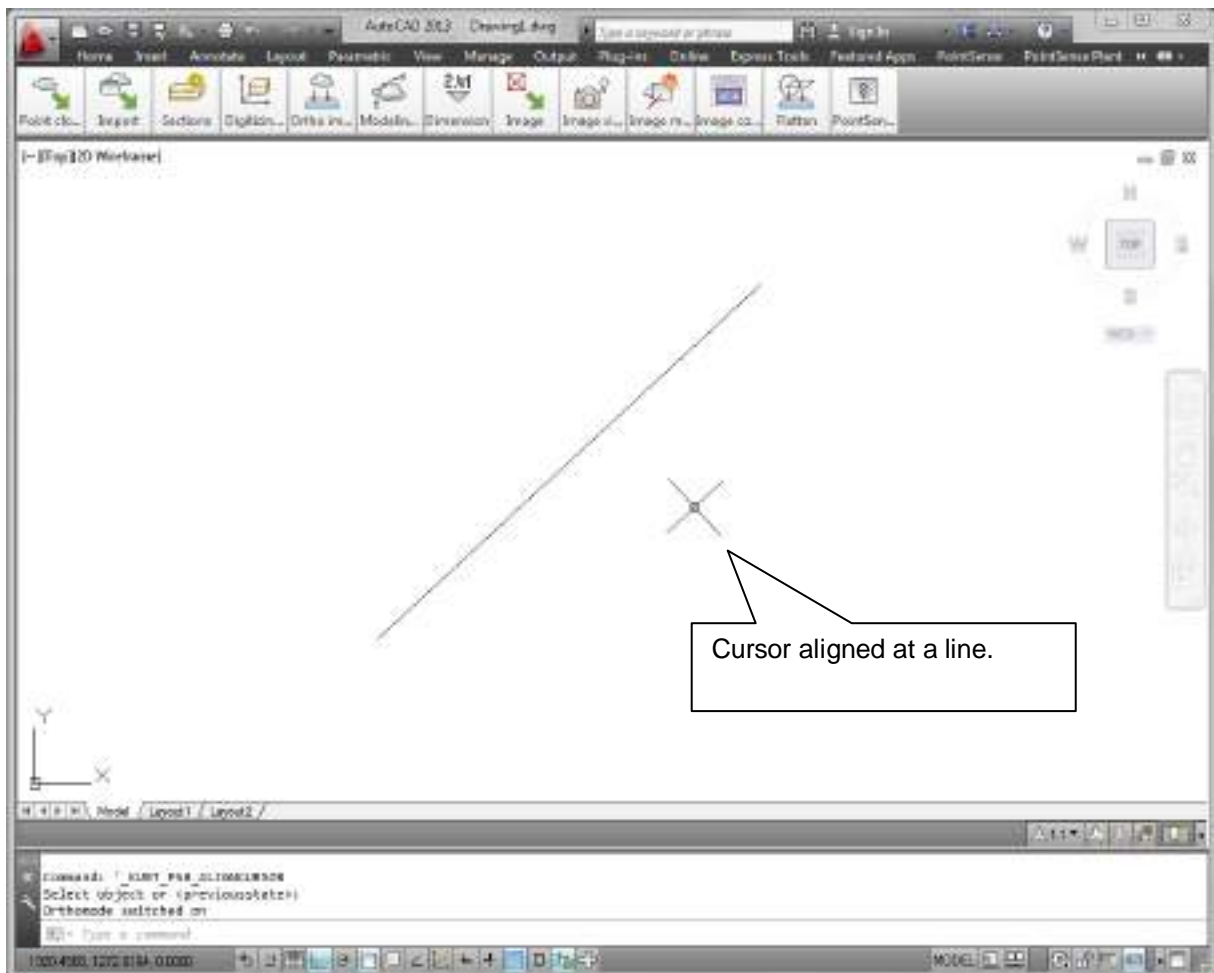
This command is used to give the cursor axes the same inclination as a certain line or a polyline on the screen. This is necessary if you want to draw perpendicular to a line or polyline using the option ORTHO in connection with AutoCAD. **Thus, manually measured additions (a small room, a recess) can easily be inserted.**

When you call up the command, following prompt appears:

Command: **psbaligncursor**


Select object or <Return> for previous cursor state:

Now select an object at which the cursor is to be aligned.



To reset the cursor to the normal orthogonal position, just press Return.

Dimension – 3D distance dimension

	Command Name:	PC3DDIMENSIONCREATE
	Ribbon Tab:	As-Built Modeler

With this command you can determine and dimension the spatial distance between two points. The 3D dimension will always be aligned with the current view and thus will be readable.

First point for 3D dimension:

Second point for 3D dimension:

The 3D dimension is meant for quick determination of distances and the presentation of spatial situations.




It is always displayed in the current text style and with the current As-Built dimension size. The latter depends on the settings for the drawing unit and the block scaling factor under As-Built → Settings → section “Drawing”.

The 3D dimension is by default put on the layer “3DDIMENSIONS”. But you may change the layer under As-Built → Settings → section “Layer”.

The number of decimals depends on the setting for the AutoCAD unit.

Measure – Measure horizontal distance

	Command Name:	'PSBHZDIST
	Ribbon Tab:	As-Built Modeler

The command is used to measure the horizontal distance between two points. This means that the measured points are projected into the XY plane of the current UCS, and the distance is measured there. The command is transparent i.e. **you can use it within other commands** in case you have to enter such a measurement.

Befehl: 'PSBHZDIST


First point for horizontal distance: *(measure or click the first point)*

Second point for horizontal distance: *(measure or click the second point)*

4.731 *(this is the calculated distance between the two points)*

You can use this command to determine the horizontal distance between two points.

Measure – Measure height difference

	Command Name:	'PSBHTDIFF
	Ribbon Tab:	As-Built Modeler

This command allows to measure the height difference of two points. It is a transparent command, i.e. you can call it up within another command if you have to take a vertical measurement.

Command: 'PSBHTDIFF


First point for height difference: *(measure or click the first point)*

Second point for height difference: *(measure or click the second point)*

height difference = 2.5 *(this is the calculated height difference)*

The height difference always refers to the difference of the Z coordinates of the two points in the **World Coordinate System (WCS)**.

Measure – Measure Z difference

	Command Name:	'PSBZDIFF
	Ribbon Tab:	As-Built Modeler

This command is used to determine the Z coordinate difference between two points. It is a transparent command, i.e. you can call it up within another command when you have to enter such a measurement value.

Command: 'PSBZDIFF


First point for Z difference: *(Measure or click first point)*

Second point for Z difference: *(Measure or click second point)*

Z difference = 2.5 *(this is the calculated Z difference between both points)*

The height difference always only refers to the difference of the Z coordinate of both points in the **User Coordinate System (UCS)**.

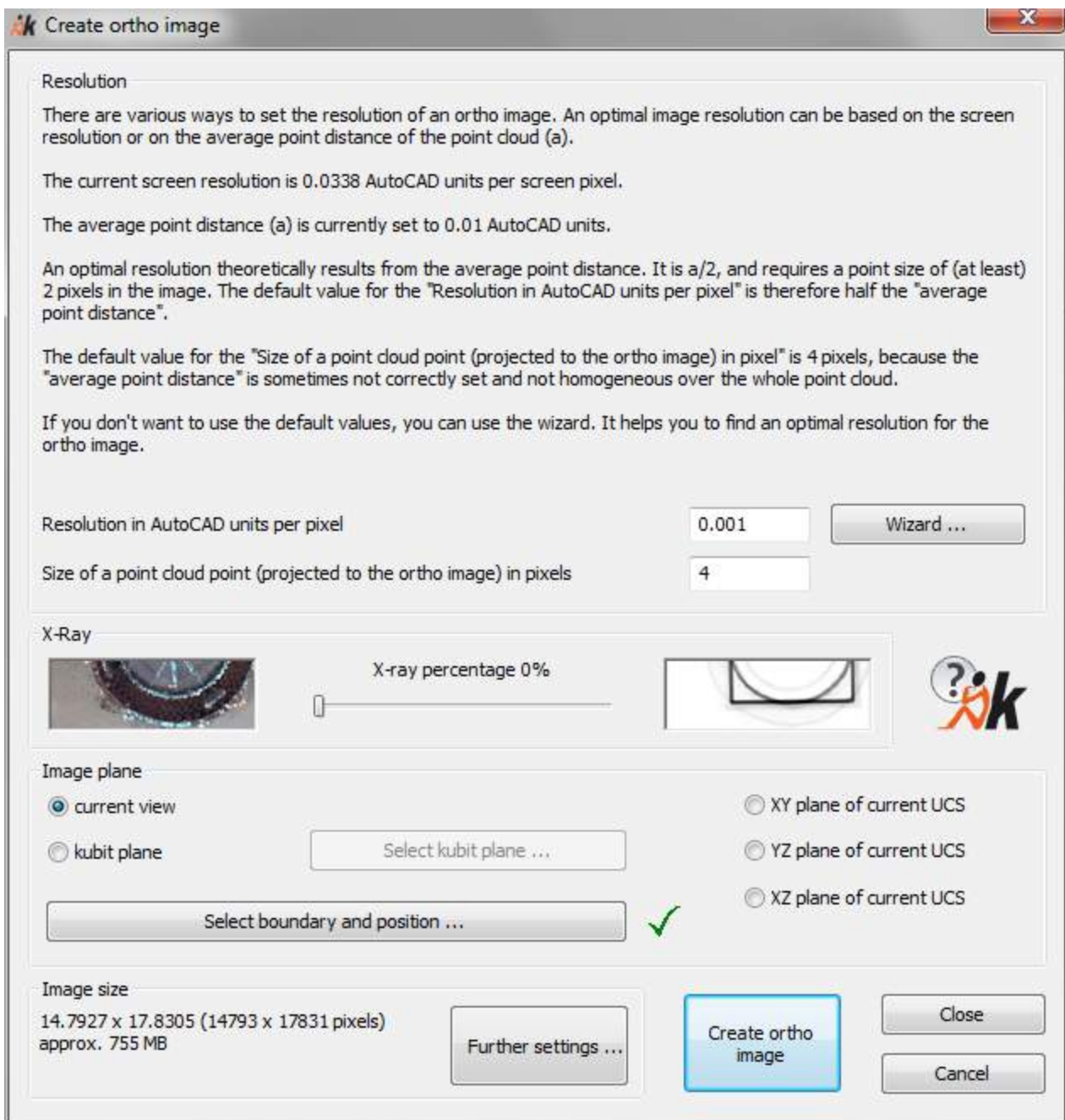
Create ortho image

	Command Name:	PCCREATEIMG
	Ribbon Tab:	As-Built Modeler

This command creates an ortho image from a point cloud. Ortho images are parallel projections into a plane. They differ from orthophotos. Ortho images do not use photos as source of the colour they use the colour of the point cloud.

An Ortho image can be a useful tool for digitizing. You may use it to display scan data inside CAD software which does not support point clouds yet.

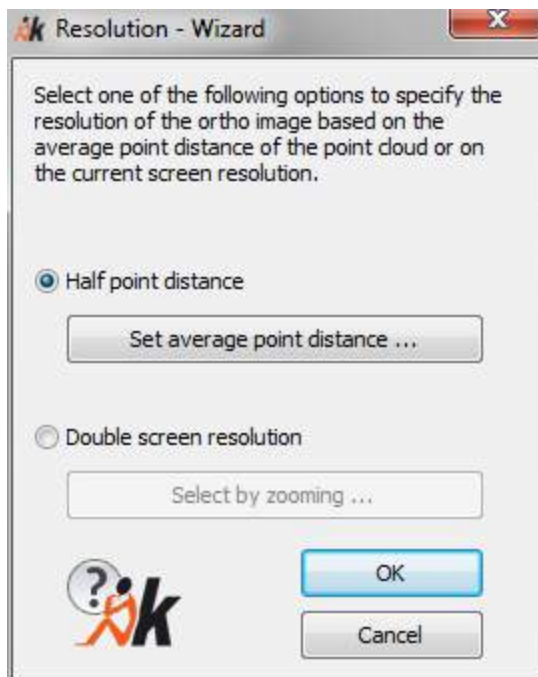
When running the command following dialogue opens:



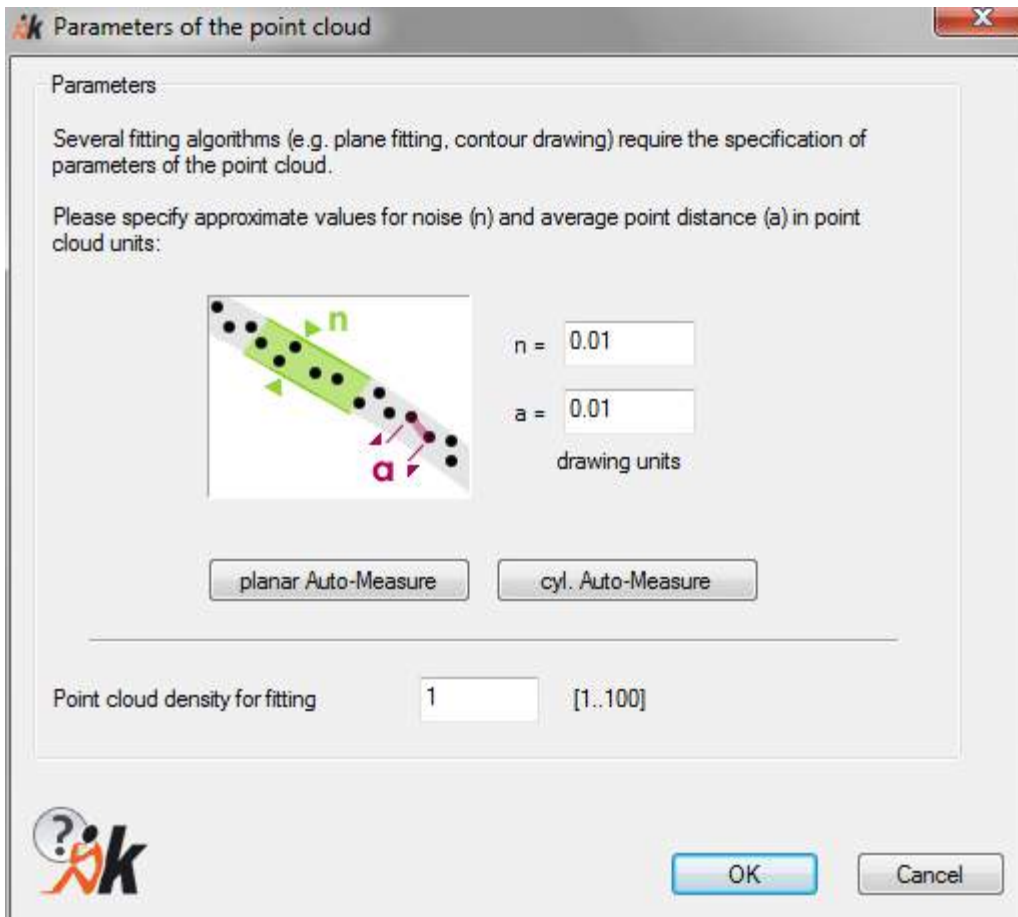
The creation of an ortho image always refers to the *current* point cloud. You have to define the resolution of the ortho image, its position and size, the point size as well as the x-ray percentage.

Resolution

The dialogue should largely be self-explaining. If needed you may use the wizard to assist you with the resolution settings.



The **option “Half point distance”** is perfect, if the point cloud parameter **a** (average point distance) is set adequate. When clicking on “Set average point distance...” you may directly check or edit and apply the parameters.



The option **“Double screen resolution”** is perfect, you orient on the screen resolution. When clicking on **“Select by zooming”** all dialogues disappear temporarily and the command line prompts you to zoom the view according to your wish and confirm with ENTER.

Select zoom level. [Enter] to finish...

Afterwards you are back in the main dialogue with the new resolution setting.

We recommend to experiment with the resolution settings for getting the perfect resulting image for your project and the certain point cloud.

X-Ray



a) Colour



b) 80% X-ray



c) 100% X-ray

If you leave the x-ray percentage at 0%, you will get a true colour image of the point cloud surface that faces to the image plane.

If the x-ray percentage is set to 100%, image areas where many points lie behind each other are displayed dark while the others are light. This way you get a kind of x-ray image of the object.

Image plane

Here you define the image plane of the ortho image. It can lie exactly in one of the three planes of the UCS, in a previously defined kubit plane or simply in the plane of the current view. When selecting the “current view” option the image plane is automatically placed in front of all selected surfaces.

Note: We recommend to use the “current view” option. First clip the point cloud in a way that only the parts you like to have in the ortho image are visible. Then draw or define an object for the alignment of the UCS (e.g. a kubit plane). Now align the UCS with this object (e.g. with the kubit command “UCS from plane”). Finally switch to the plan view of the new UCS and create your ortho image.

Button „Select boundary and position...“

When clicking the button the dialogue is left. The view of the drawing changes to the selected plane with a zoom to the current point cloud. Now you draw a rectangular window to define the frame and position of the new ortho image.

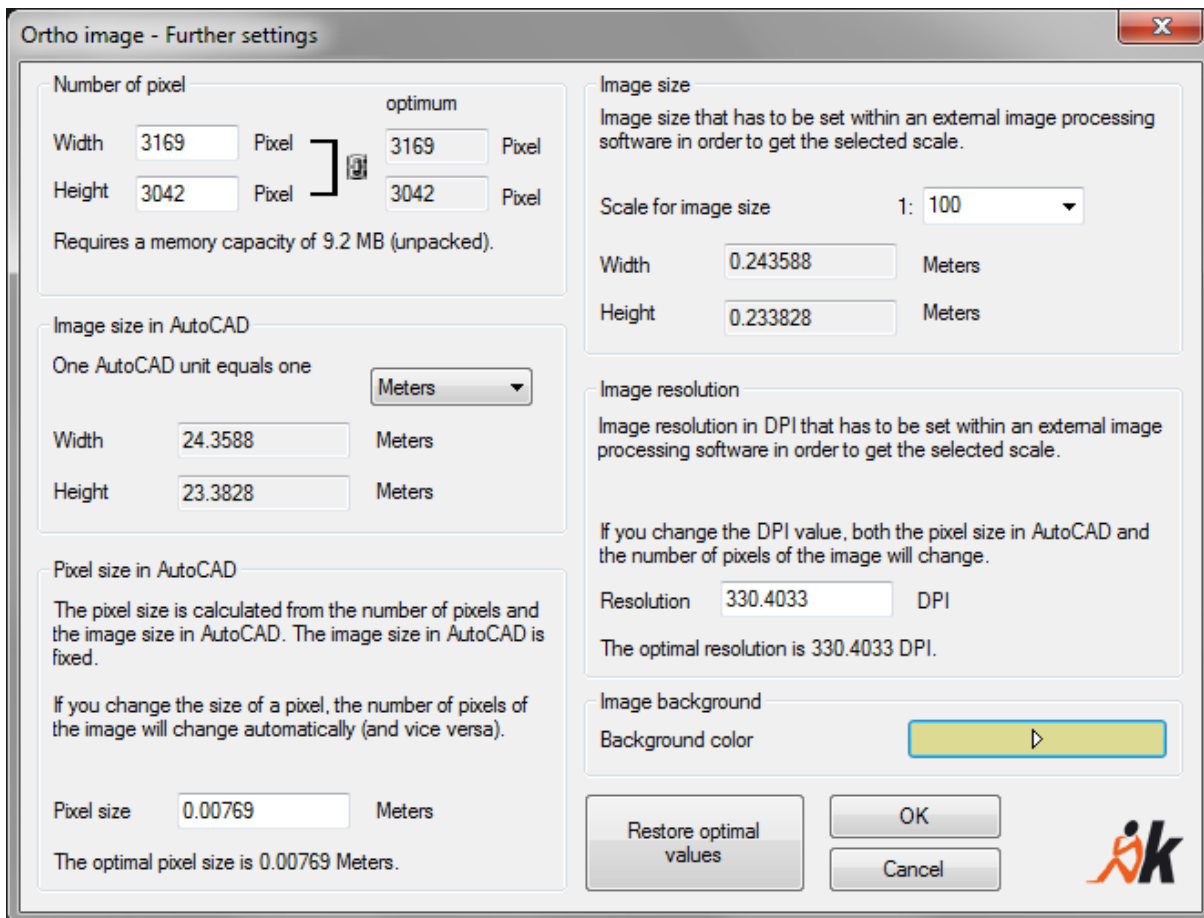
Image boundary:

(first point for boundary rectangle)

Second point:

(second point for boundary rectangle)

Further settings

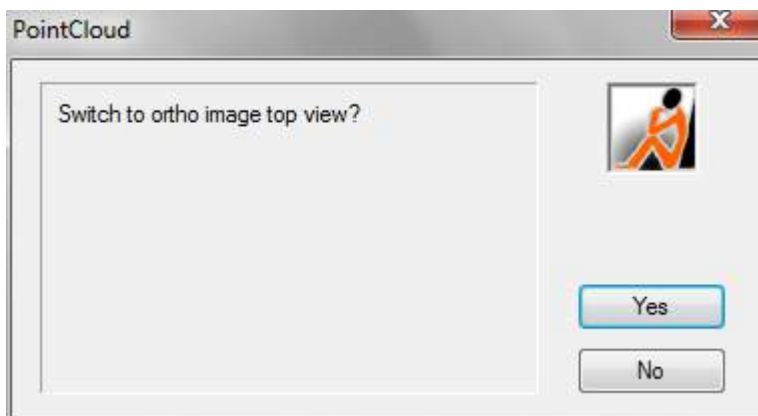


Here you may do further settings for the image size, resolution, background colour and image scale. The settings will help you processing the image with external image processing software.


Button “Create ortho image“

A click on this button starts the image computing. You are prompted for a path and name for the ortho image. You can chose between BMP, TIF, JPG and PNG for the image format.

After the computation following message opens.



Flatten drawing...

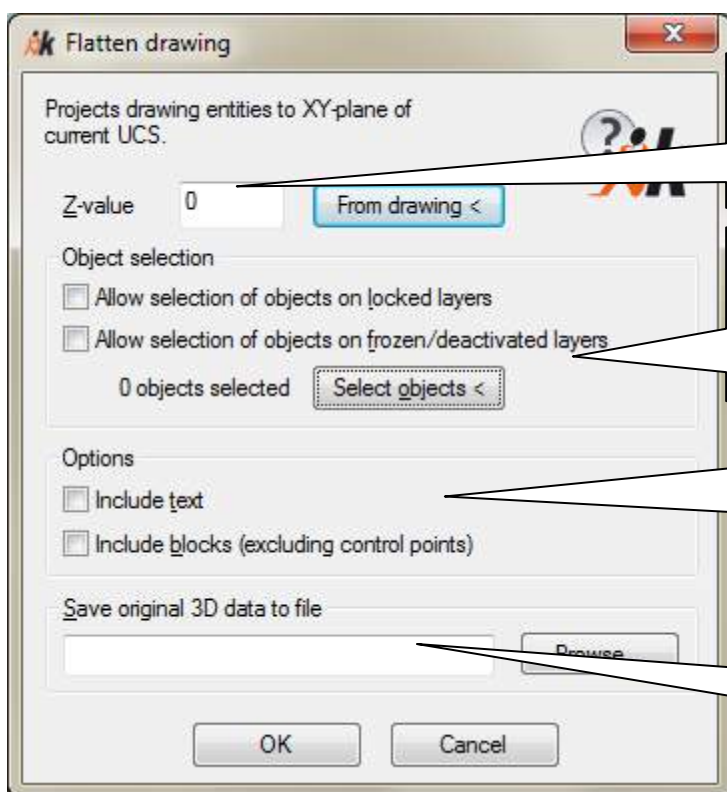
	Command Name:	KFFLATTEN
	Ribbon Tab:	As-Built Modeler, Building Plans, Photo, TS Control

The command is used to orthogonal project line-like 3D drawing elements into a plane. Thus, elements that have been measured in the 3D mode can be processed to a 2D plan in AutoCAD.

The projection plane is defined parallel to the XY plane of the current UCS with the Z value that has been entered in the field 'Z-value'.

Following types of elements are flattened:

Points, circles, lines, all poly lines, ellipses, arcs, beams, straight lines and dimensions.



All elements are projected to this Z value of the UCS. Via the button you can pick a value from the drawing.

Via the button you can select the objects to be flattened in the drawing. Furthermore you can enable the selection of objects for locked or frozen layers.

Tick these options if you like to flatten text and blocks as well. Note that control points and instrument stations cannot be flattened.


If necessary, all elements can be saved in an external file before the flattening process.

If the options 'include text' and 'include blocks' are activated, texts and blocks too will be shifted to the defined Z value. However, the inner geometry of a block is kept.

Dimensions are only flattened if measurement lines and texts are parallel to the UCS. It makes sense to dimension the plan after having flattened the drawing.

The blocks for control points are not flattened. Flattening does not make sense here, since the 3D positions are absolutely valuable for image rectification and image orientation.

Modeling – Fit outline plan

	Command Name:	PCWALLFIT
	Ribbon Tab:	As-Built Modeler and Building Plans

This command serves to extract straight contours from the point cloud by using an automatic fitting algorithm.

The result is strings of 2D lines (optionally lines or polylines), that run along a point cloud slice or the current UCS plane. The command is particularly suitable for efficiently and precisely drawing wall polygons in building floor plans and sections.

The command is flexibly designed and supports various workflows:

- Working with point cloud slices: Precise wall forms by automatically fitting the lines onto the point cloud track.
- Working in the planar scan view or scan navigation: Intuitive clicking in the image without line fitting
- Constrained angularity (perpendicularity) and parallelism for architectural surveys.

Command sequence

The command works on the following **basic principle**:

Standard Mode “Faces”: Two points are picked on each of the line sections to be drawn. The resulting line sections are automatically trimmed, so that the corners are indirectly created. The command also always alternately prompts for the first and second point of the line section.

Command: **PCWALLFIT**

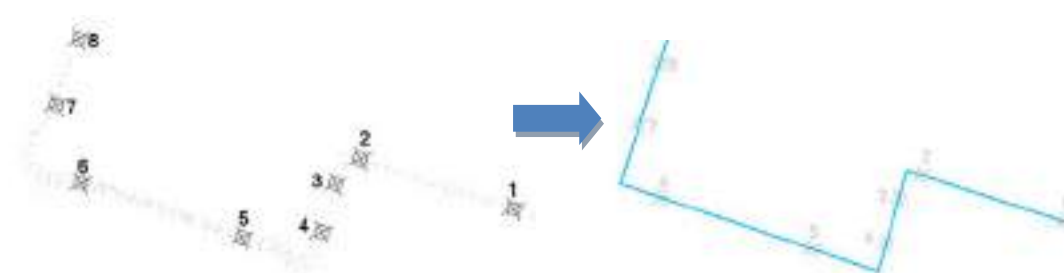
Current settings: Fitting on, Fixed angles off

First point on wall or [CORner/Fitting off/UNDO/CONFig] <CORner>:

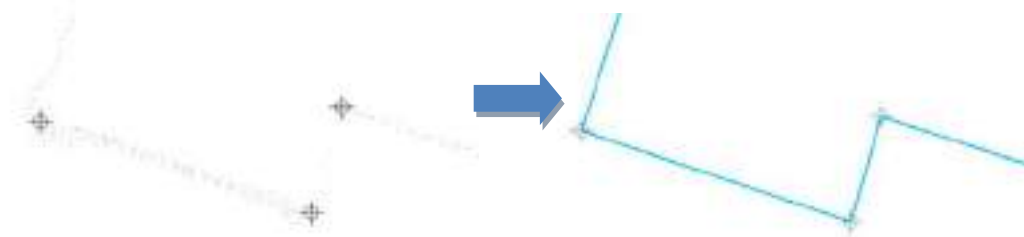
Last point on wall or [Fitting off/UNDO/CONFig]:

First point on wall or [CORner/ORtho/CONTInue/Fitting off/UNDO/CONFig] <CONTInue>:

...



Alternatively, you can switch at any time into the “Corner” mode. Here the corner points are directly picked and the command joins them with a line section.



Command: **PCWALLFIT**

Current settings: Fitting on, Fixed angles off

First point on wall or [CORner/Fitting off/UNDO/CONFig] <CORner>: **COR**

(Switch to "Corner" mode)

Point in corner or [FAce/ORtho/CLose/Fitting off/UNDO/CONFig] <FAce>:

Point in corner or [FAce/ORtho/CLose/Fitting off/UNDO/CONFig] <FAce>:

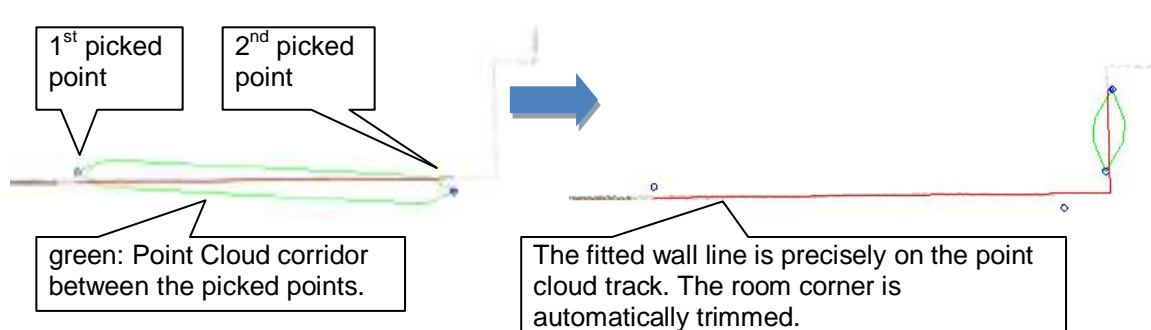
...

Command line options:

You can switch between the two modes "Faces" and "Corners".

With "Fitting on/off" you can activate/disable the automatic fitting of the line onto the point cloud slice. If the drawing contains a point cloud and there is defined slice in it, then the command automatically starts with "Fitting" activated.

The scan points, that lay within a corridor between the first and second picked points of a line section, are analysed and an average straight line is computed. You don't need to precisely pick in the scan point track, nearby will suffice:

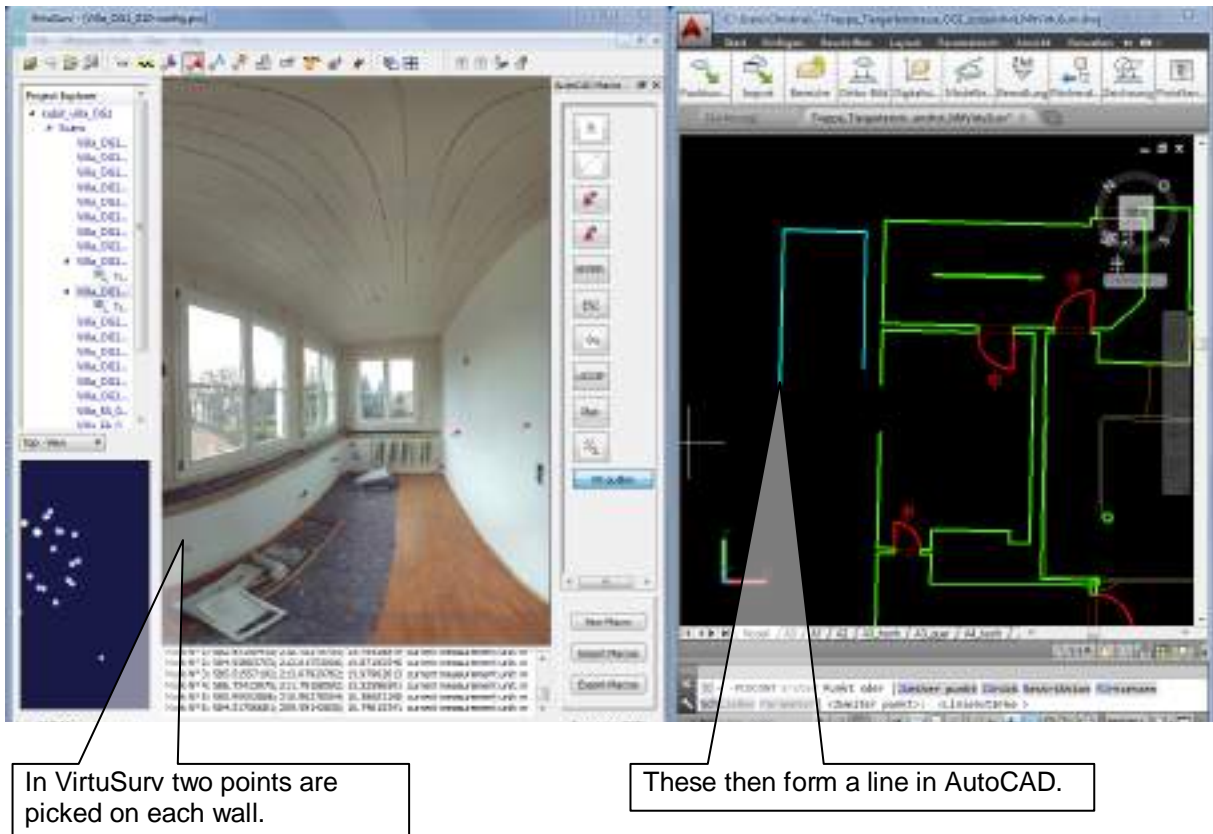


The corridor is displayed green, when it contains sufficient scan points and a line can be fitted. It is red when there are insufficient points to allow line fitting.

When fitting is switched off the line is drawn precisely between the picked points. Fitting can be re-activated at any time.

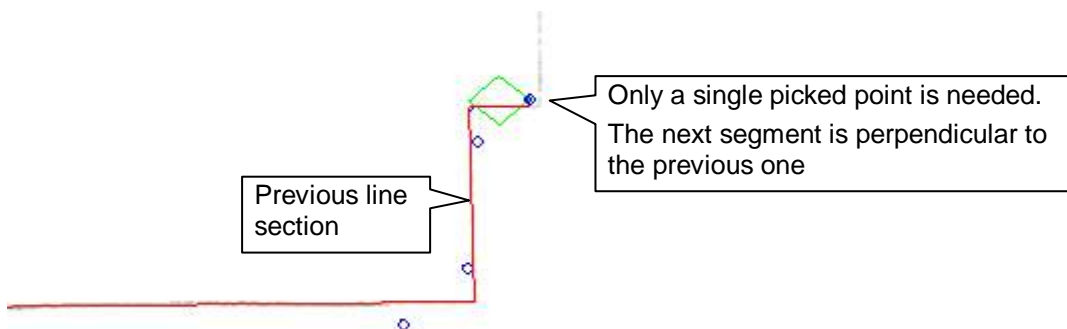
Switching off fitting is meaningful

- When working without any point cloud slices, e.g. when the point cloud is being used in the planar scan view outside of AutoCAD or when you are using ScanNavigation.



- In areas that contain just a few or no points and thereby would lead to an inaccurate fitting.

Use Ortho when the next line section must be perpendicularly constrained to the previous section, e.g. in small alcoves or projections.

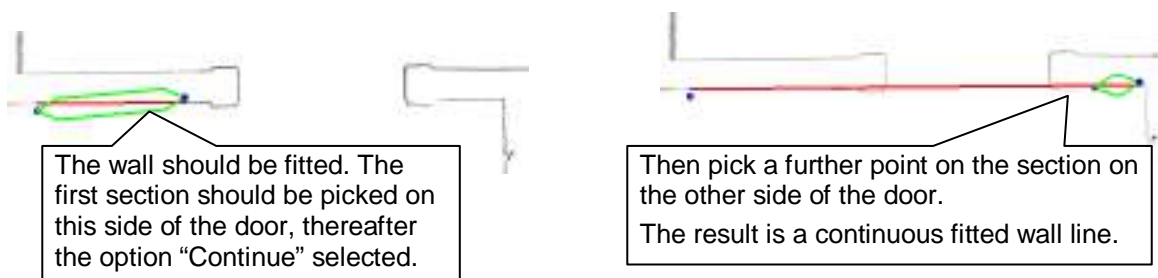


With “Ortho” only a single point must be picked on the line section. The resulting line will be kept perpendicular and will be the best fit possible.

First point on wall or [CORner/ORtho/CONTInue/CLose/Fitting off/UNDO/CONFig]
 <CONTInue>: **OR** (select ortho option)

Last point on wall or [Fitting off/UNDO/CONFig]: (Pick the end point of the line)

With “Continue” you can skip areas of poor scan quality in the point cloud track and exclude them from the adjustment calculation (e.g. furniture or windows). This option is only available when “Fitting” is activated.

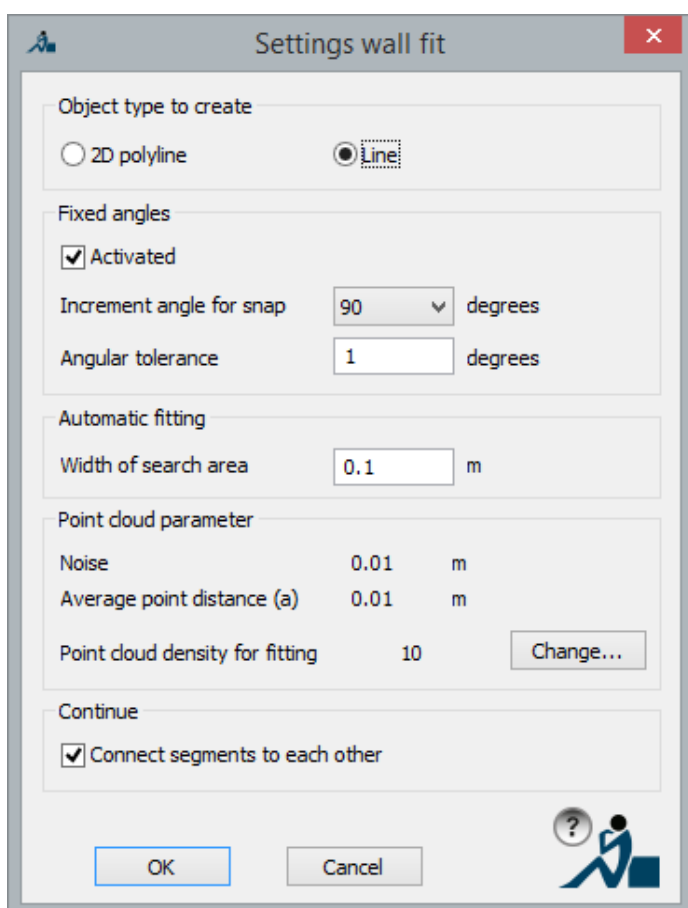


You can close the line chain at the end of the command with “Close”. It will be attempted to intersect the first and last line with each other.

With “Configure” you open the settings dialogue box for the command, which is described further below.

The command is ended by pressing the ESC key or by using the option “Close”.

Configuration and options



Object type to be created: You decide whether the result should be single lines or a connected 2D polyline.

Fixed angles: Automatic alignment of the line segments in accordance with a fixed angular grid. This option will be described in more detail below.

Automatic fitting, width of the search area: Width of the point cloud data corridor used for the line fitting. A higher value allows a less precise picking, a too high a value introduces problems with parallel lying contours within a small distance of each other (e.g. walls).

Point Cloud Parameter: The quality of the fitting results can be influenced by the settings of the point cloud parameters.

Continue – Connect segments to each other: Here you define, when using *Continue*, whether the fitted line segments should be connected to each other or if single line segments should be created with gaps between them.

Fixed angles

This option is used to draw perpendicular wall lines (or other predefined wall line angles), as are often required in architectural plans. The lines/polylines will be automatically aligned to a fixed angular grid, when their direction almost corresponds with the grid (within a tolerance).

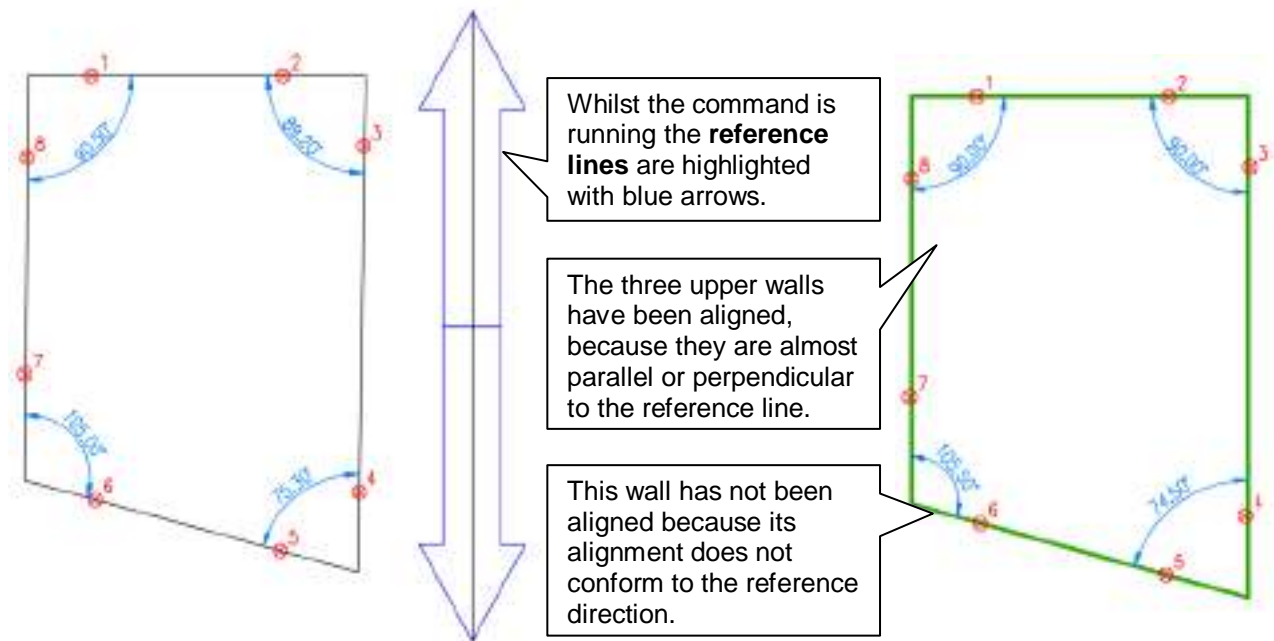
Incremental angle for snapping: Defines the angular grid for the automatic alignment of line segments. Angular steps of 90°, 45°, 30° and 22.5° can be set.

Angular tolerance: Gives the maximum angular variance for the alignment. Should the direction of the surveyed line vary more than this tolerance from one of the reference lines, then it will not be aligned, but retains its surveyed direction.

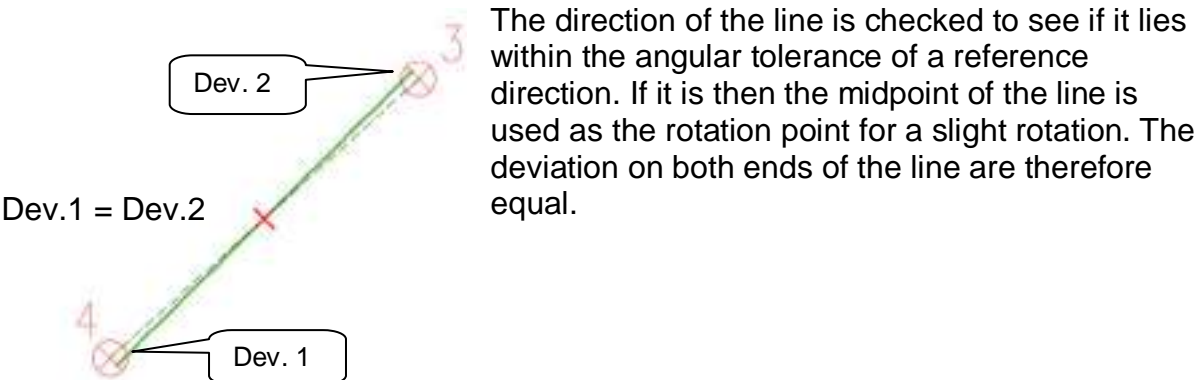
If the option “Fixed angles” is activated then the following prompt comes after starting the command:

Select reference line(s) or [UCs/CONFig] <UCs>: (Select one or more reference lines)


As reference lines, select structural elements that have been accurately surveyed and are very important (e.g. external walls of the building, load bearing walls, etc.). It is imperative that they should be longer than the wall lines that are to be created. Several reference lines can be selected.



What happened in detail?



Modeling – Fit polygon

	Command Name:	PCFITCONT
	Ribbon Tab:	As-Built Modeler and Building Plans

With this command you can fit polygons as good as possible to a point cloud section. The software uses a highly sophisticated algorithm to do so.

For preparation define a slice by using the command → **Sections – Define slice (UCS)**. Thus the contours to be modelled emerge, e.g. the profile of a facade or a layout. If previously no slice has been defined or if there are several slices visible (→ **Define sections – Section Manager**) following message pops up when running the command.



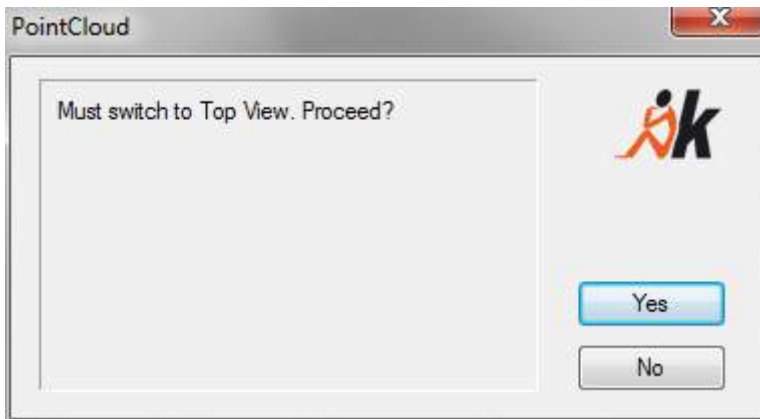
An approximation has to be entered for every polygon to be fitted. This approximation polygon can be drawn roughly along the point cloud slice. It is **important** that polygon do not overlap each other. If they do an error message will open.

The approximations can be defined either during the command run by clicking single points or by selecting an existing polyline. After starting the fitting process the approximations are aligned with the point cloud slice within seconds. Please **note** that the number of vertices of the resulting polygon always complies with the one of the approximation polygon.

The command run is as follows.

Command: KUBT_PC_FIT_CONTOUR

First the software checks if the current user coordinate system is aligned with this slice (→ **Digitizing tools – UCS from slice**) and if the plan view is enabled. If that is not the case, it is done automatically after a brief inquiry call.



Next, you are asked to enter the points for an approximation polygon.

- Specify first point of approximation or [Select objects/paraMeters]: *(click a point)*
- Next point or [Undo/paraMeters]: *(click another point)*
- Next point or [Finish/Undo/Another polyline/paraMeters] <Finish>: *(click another point or
undo or start another polyline or start the fitting)*
- Next point or [Finish/Undo/Another polyline//paraMetersClose] <Finish>: *(click another
point or undo or start another polyline or close or start the fitting)*

After the input of the first polygon segment you may already start the fitting (default option <Finish>). To revise the point selection use the option [Undo] which is available during drawing. If you like to fit several polygons you may select the option [Another polyline] to start the next polyline. After you entered at least two segments you also have the option to [Close] the polyline. After having closed a polygon or after having chosen the option [Another polyline] you are asked to define the first point of the new polygon.

Specify first point for new polygon or [Finish/Undo/paraMeters] <Finish>:

Of course also here the option <Finish> for the final automatic fitting or the ESC button for canceling the command are available.

Optional ([Select objects]) it possible to use existing polylines from the drawing as approximation.

Specify first point of approximation or [Select objects/paraMeters]: **S**

Selected objects: Copy

Select approximated polylines or [Toggle copy replace/Draw approximation]: *(Selection of
polyline(s) by clicking)*

1 found *(With ENTER you confirm your selection and run the process)*

By the option [Draw approximation] you get back to the direct point wise definition of polylines. Using the option [Toggle copy replace] you may specify if the approximated polylines shall be copied or replaced. The current state is always shown before of the input prompt, e.g.:

Selected objects: Copy


You may use 2D or 3D polylines as approximation information. During the fitting process 3D polylines are converted to 2D polylines which are placed on the XY plane of the UCS of the slice.

Hint:

If you for instance wish to model a tunnel from a point cloud or generate multiple cross section profiles, you may use the →**Sections – Define multiple slices** command in combination with the →**Modeling – Fit polygon in multiple slices** command for fast automated processing.

Note: The quality of the results of fitting can be influenced by how the point cloud parameters are set. Use the “paraMeters” option to do so. For detailed information on the dialog that opens please refer to →**Point clouds – Set point cloud parameters**....

Modeling – Fit a polygon with variable number of vertices

	Command name:	PCFITCONTVAR
	Ribbon Tab:	As-Built Modeler and Building Plans

With this command polygons can be best fitted to a point cloud slice, whereby in contrast to the command →"Polygon fit" new corner points are automatically added, to nestle the best possible shape onto the slice.

The use of the command is analogue to that for the previously described command →"Polygon fit".


Tip for the approximated polygon:

Where you want to create a polyline that is as true as possible to the shape of the point cloud you should enter a few more corner points and where it is definitely a straight edge then as few as possible.

Note: The quality of the results of fitting can be influenced by how the point cloud parameters are set. Use the "paraMeters" option to do so. For detailed information on the dialog that opens please refer to →**Point clouds – Set** point cloud parameters....

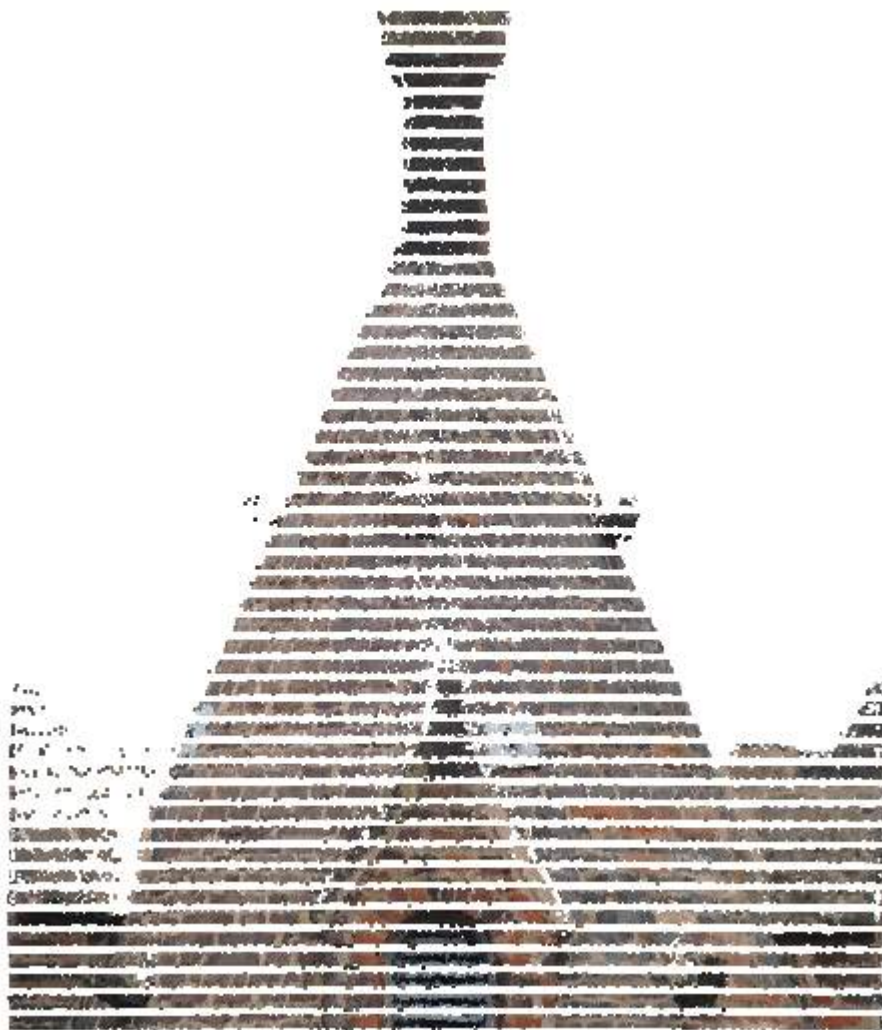
Please click the start point and the end point of the approximate polygon as close to their true position as possible since these two points are only moved slightly (maximum by the value you have set for the point cloud noise (n)).

Modeling – Fit polygon in multiple slices

	Command name	MFIT
	Ribbon Tab:	As-Built Modeler and Building Plans

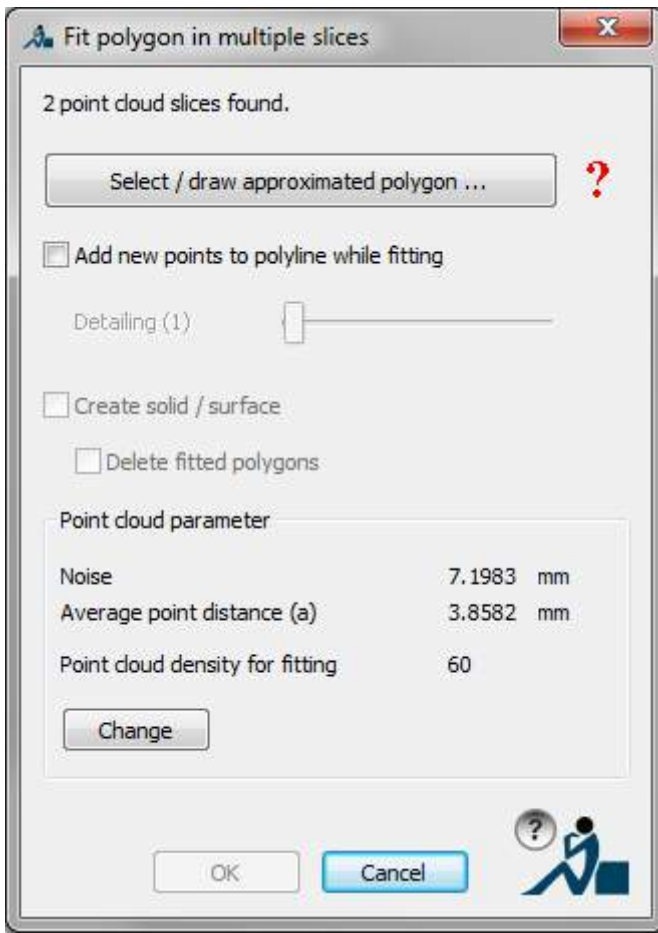
With this command polygons can be best fitted to several point cloud slices and – optional – 3D surfaces or solids can be derived from them. It lends itself, for example, to creating many cross sections or contour lines.

First cut the point cloud of your object into the desired number of slices. For this you can best use the command → **Sections – Define multiple slices**.



Ensure that in the section manager all slices that are to have a polygon fitted to them are switched to visible.

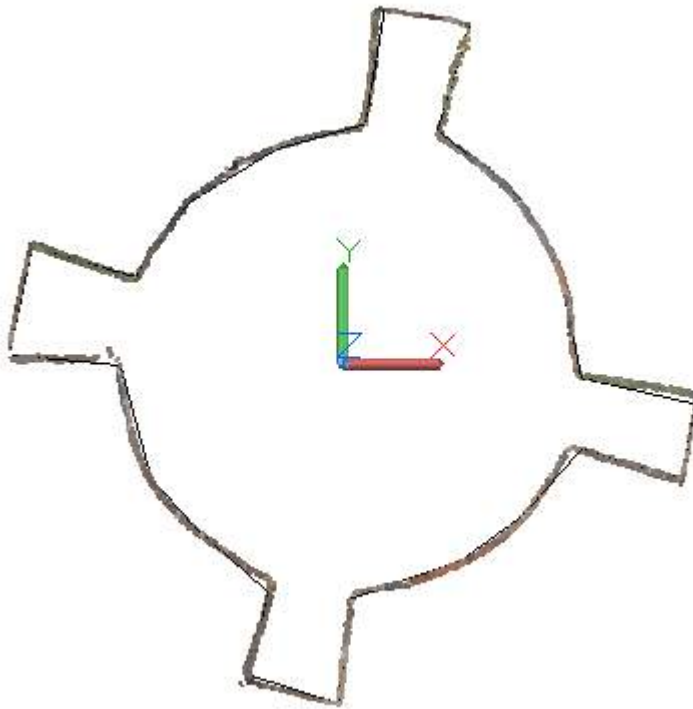
Run the command MFIT.



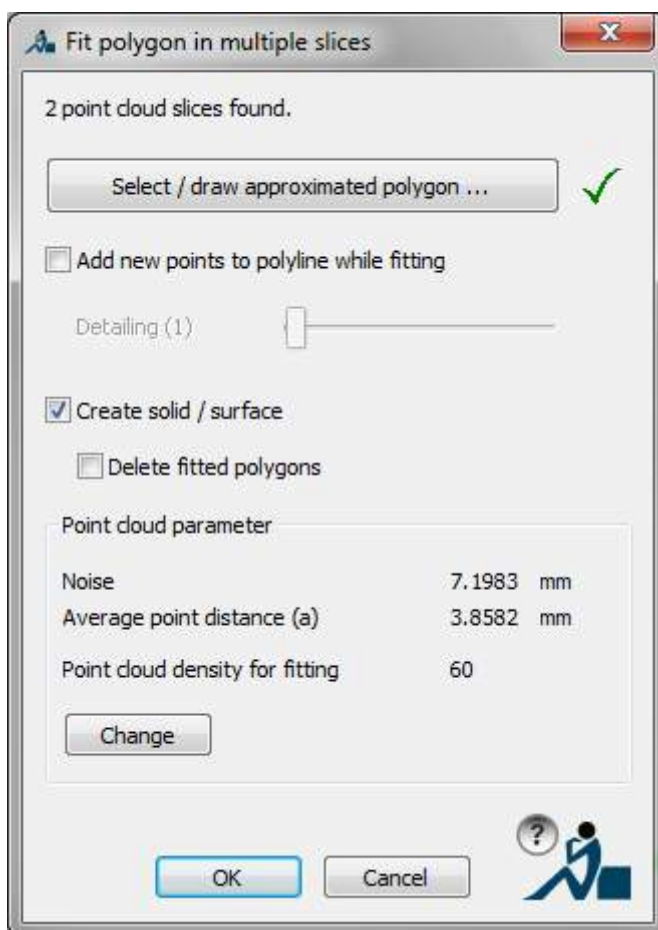
Click the button [Select/draw approximated polygon]. The dialogue box is then hidden, the view is changed to plan view in the current UCS and the first found slice is displayed on its own. The following prompt appears in the command line:

Specify first point of approximation or [Select objects/parameters/First slice/Previous slice/Next slice/Last slice]:

You can now change to another slice, for which you want to get the approximated polygon. Then draw an approximated polygon in this slice. If at the end you want to create a solid, then the approximated polygons must be closed, otherwise a 3D surface will be created.



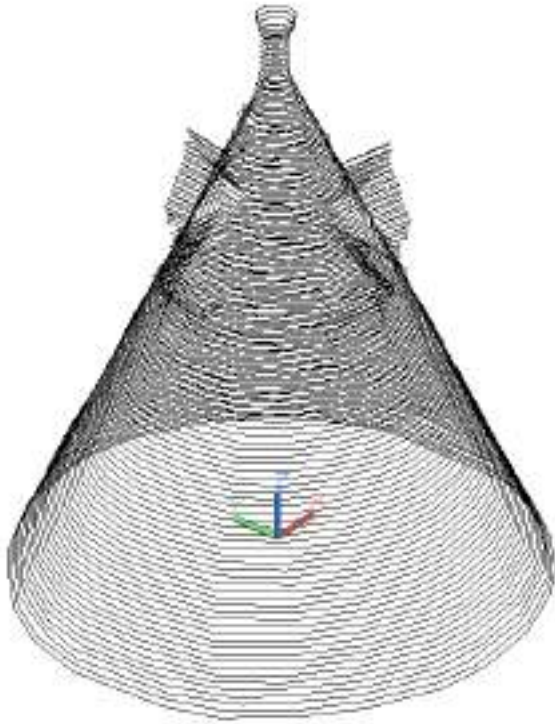
Quit with the option “Finish”. The dialogue box is once again displayed:



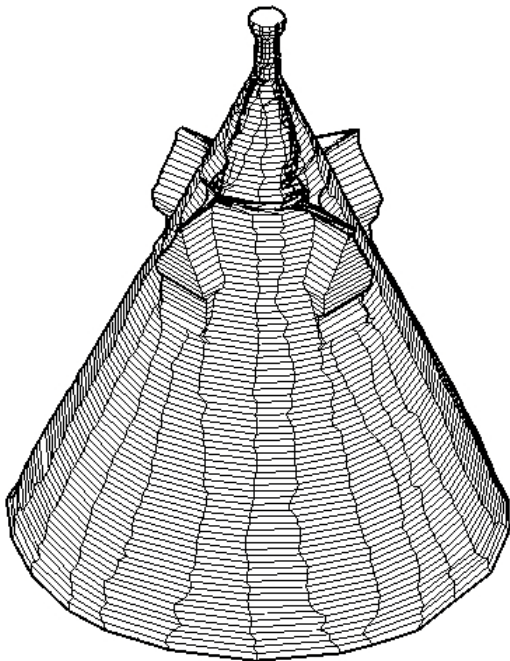
Select, if applicable, the available options and start the fitting with [OK].

The computation can – depending on the level of detail of the polygon – take a little time. The automatic insertion of additional polygon nodes during the fitting can significantly increase the computation time. Using the slider “Detailing” you can set whether just a few or many new polygon nodes should be inserted.

The results then look something like this:



Fitted polylines




Computed 3D solids in the hidden and realistic views.

Note: The creation of solids or surfaces can sometimes fail, dependant on the fitted polygons. That is connected with the AutoCAD algorithm.

Note: The quality of the results of fitting can be influenced by how the point cloud parameters are set. (→**Point clouds – Set** point cloud parameters...)

Modeling – Fit profile

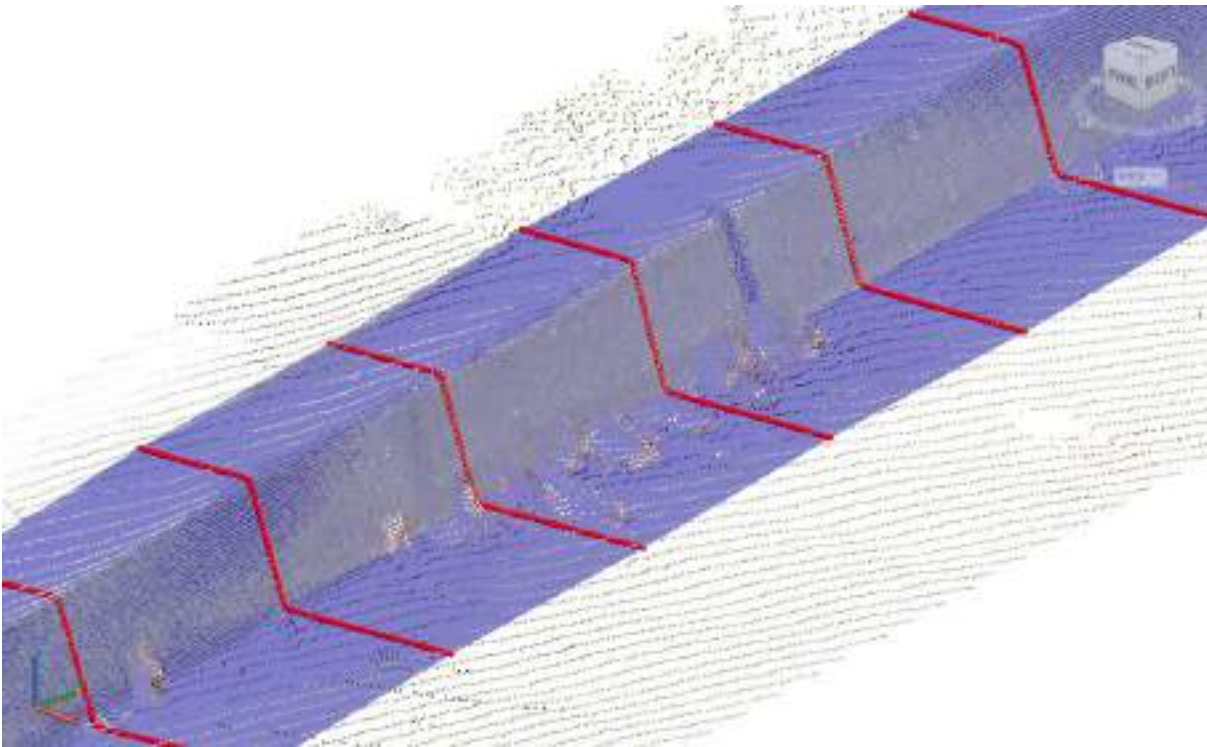
	Command name	PCPF
	Ribbon Tab:	As-Built Modeler

With this command you can fit a profile onto a point cloud and thereby model such things as balustrades, handrails, kerb edges or (steel) beams that are bent and twisted within themselves.

The profile cross section should be constant, but can be slightly wound and bent. The fitting algorithm automatically follows the course of the profile in the point cloud and in both directions, forwards and backwards.

The only thing that must be entered, without fail, is a 2D or 3D polyline, that describes as near as possible the correct profile cross section.

Hereto the example of a kerb edge:

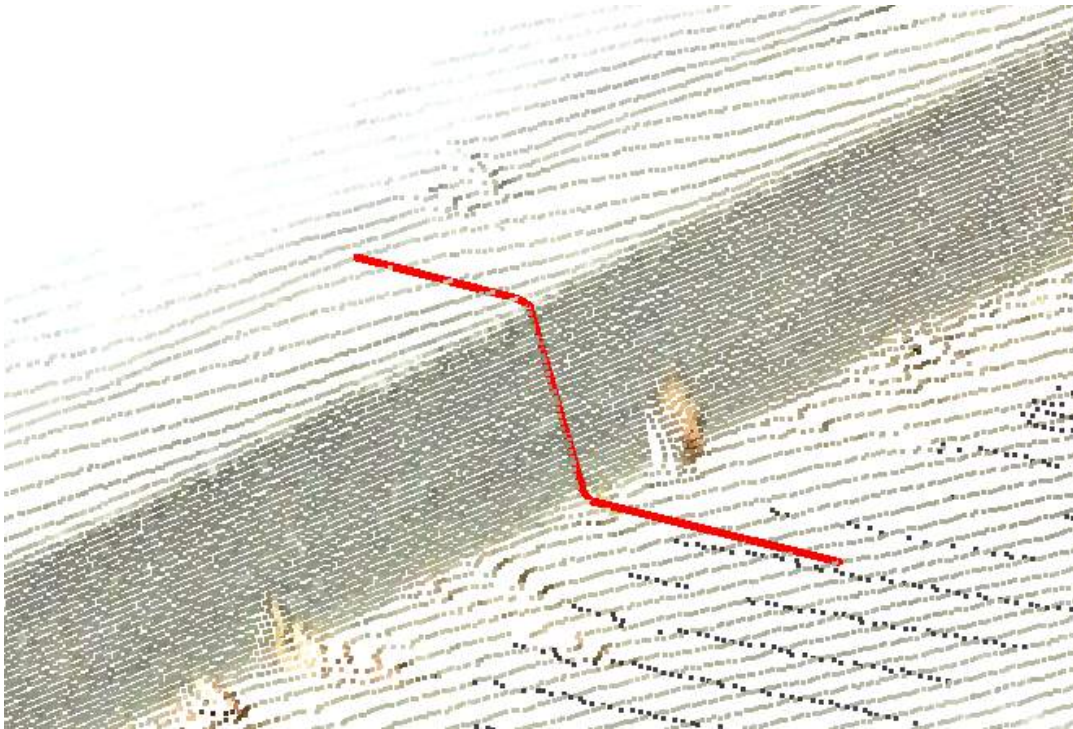


You can see in the image above, at fixed intervals, the fitted profile cross sections in red and a blue surface, which models the full profile (kerb edge).

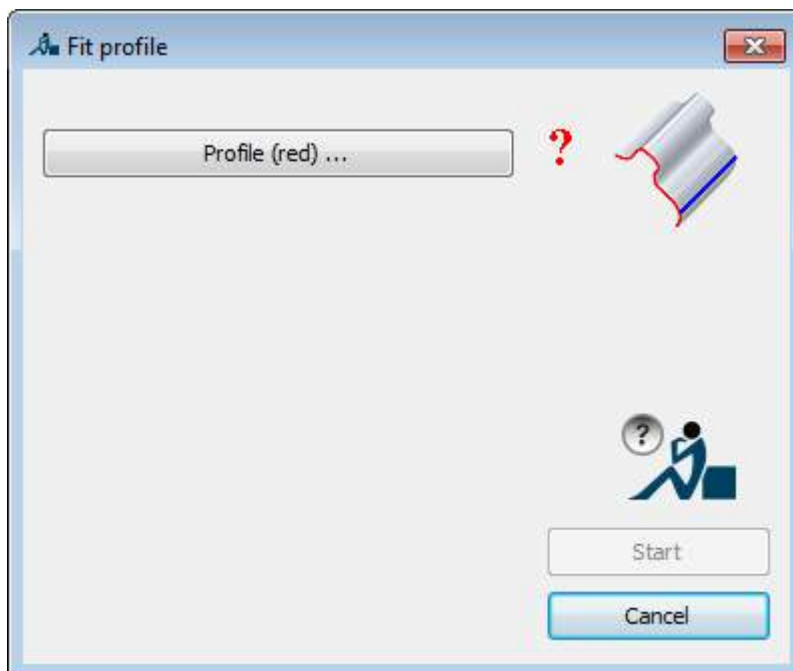
Before starting the command:

Load an AutoCAD point cloud into your drawing.

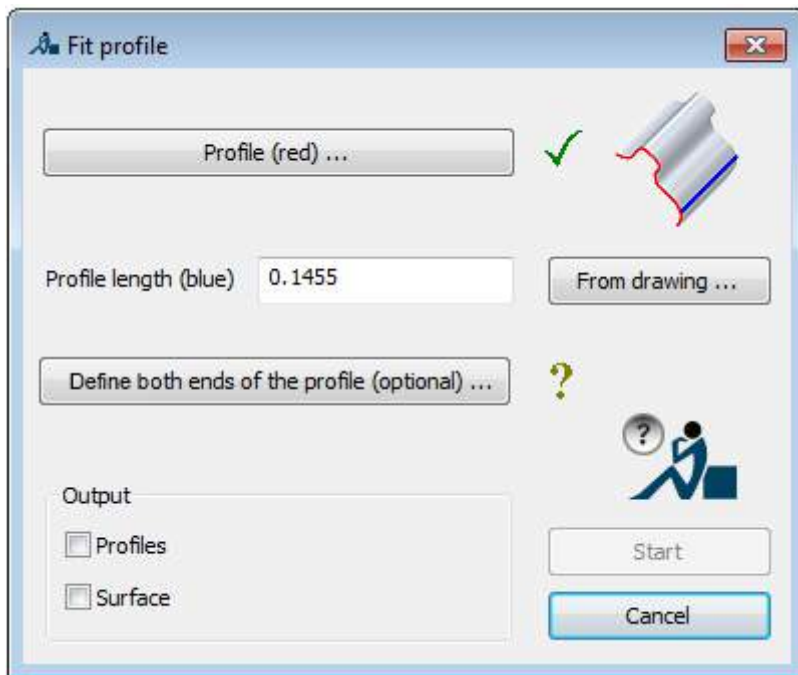
At any point on the kerb line draw a 3D polyline, which models the profile cross section.



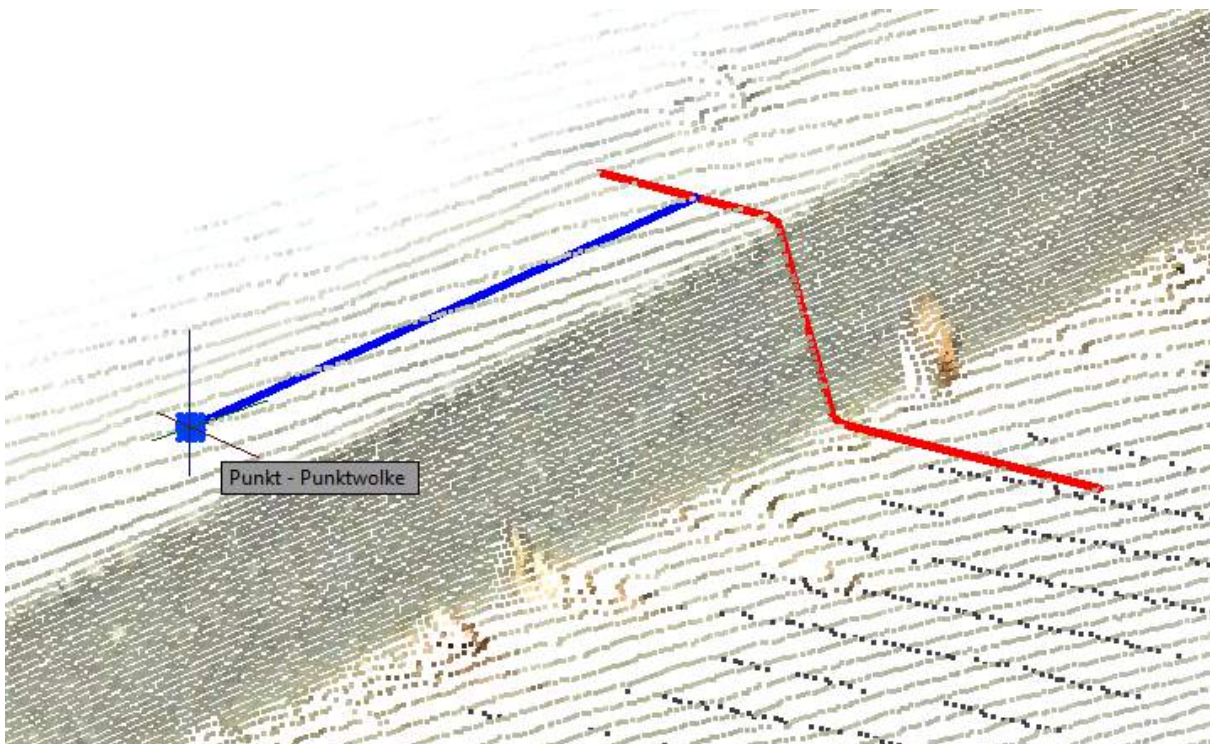
Now start the command.



In the drawing, select under the button [Profile (red) ...] the 3D polyline that you have just created.

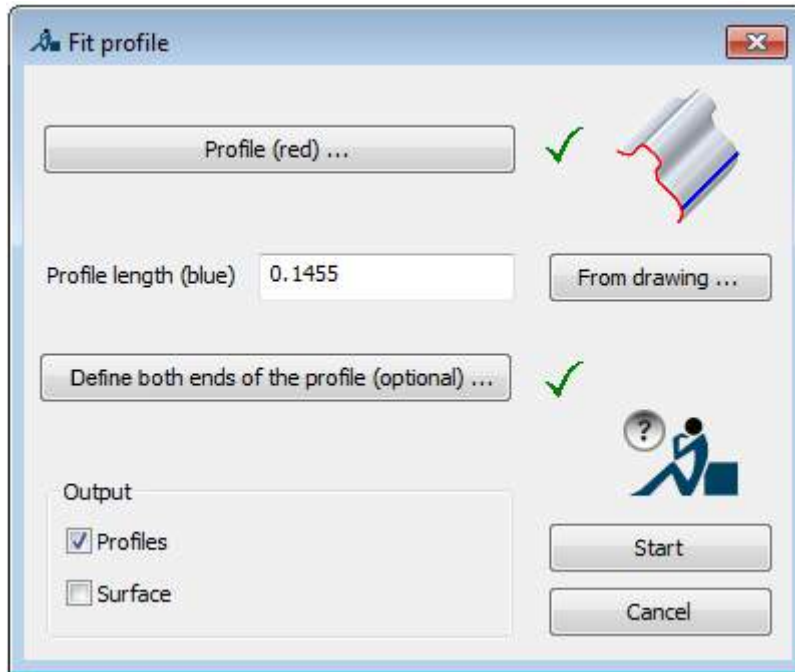


As the next step you must enter the distance between the profile sections, which must then be individually fitted onto the point cloud. The greater is this distance, the more robust is the procedure against interference and noise, small values can on the other hand react better to local variations. You must find the optimal distance by trial and error. You can accept the default value (1/3 of the polyline length of the cross section or pick the distance out of the drawing [From drawing ...]:



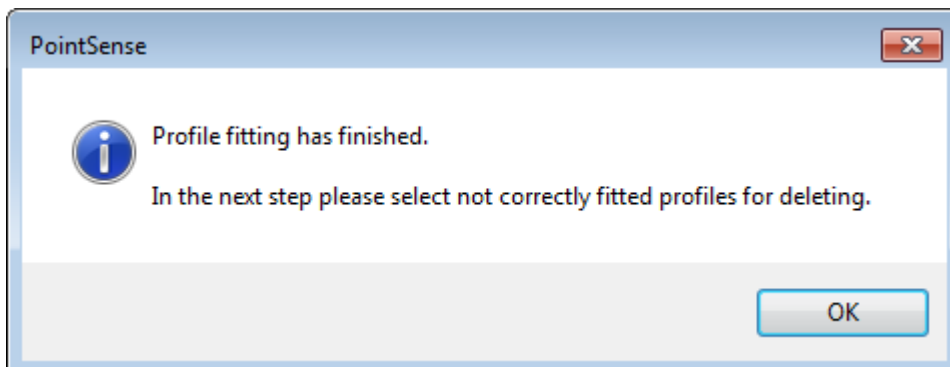
A further option is to define in the point cloud both of the end points of the total profile. If you don't enter both of the end points, the fitting will be carried out until, for

example, no more point cloud points can be found or until the starting point has been reached. Naturally you can cancel the fitting in both directions by pressing [ESC], but then there remains a large number of incorrectly fitted and unused profile cross sections, which have then to be manually deleted. For these reasons we recommend that both of the end points should be defined.

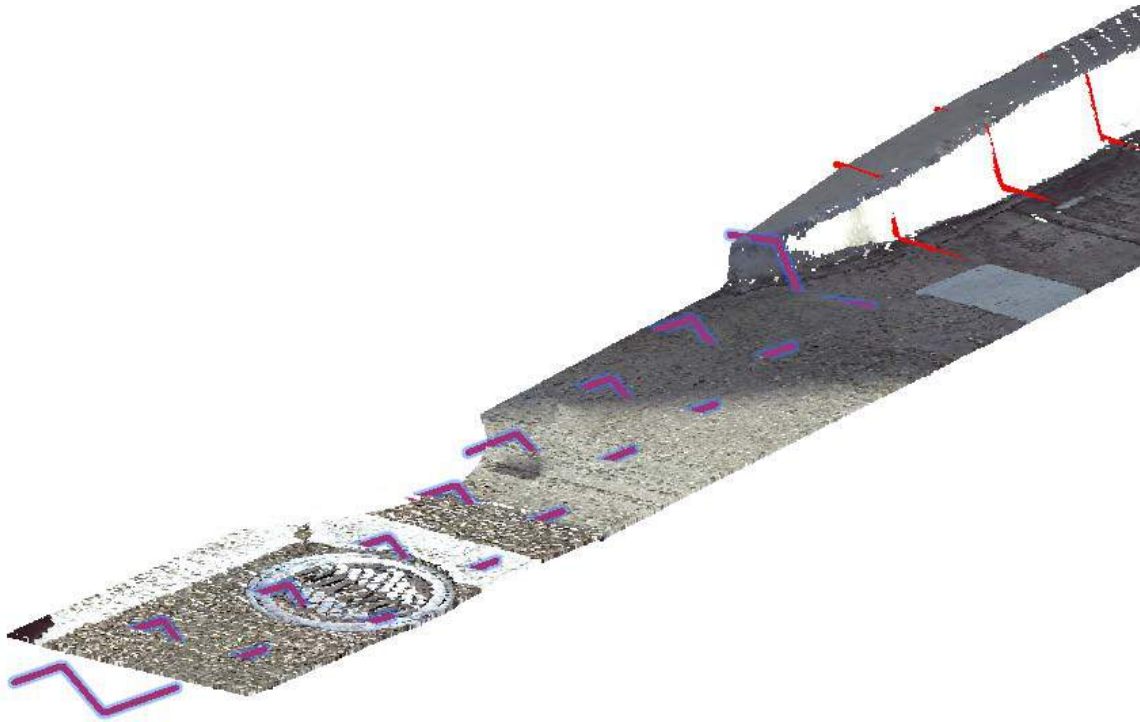


Lastly you need to decide what the command is to create. The individually fitted profile cross sections along the course of the profile with the distance between them “half profile length” and/or a fitted surface. If the default profile cross section is an open polyline then an AutoCAD surface will be created, but if the polyline is closed then an AutoCAD solid will be created.

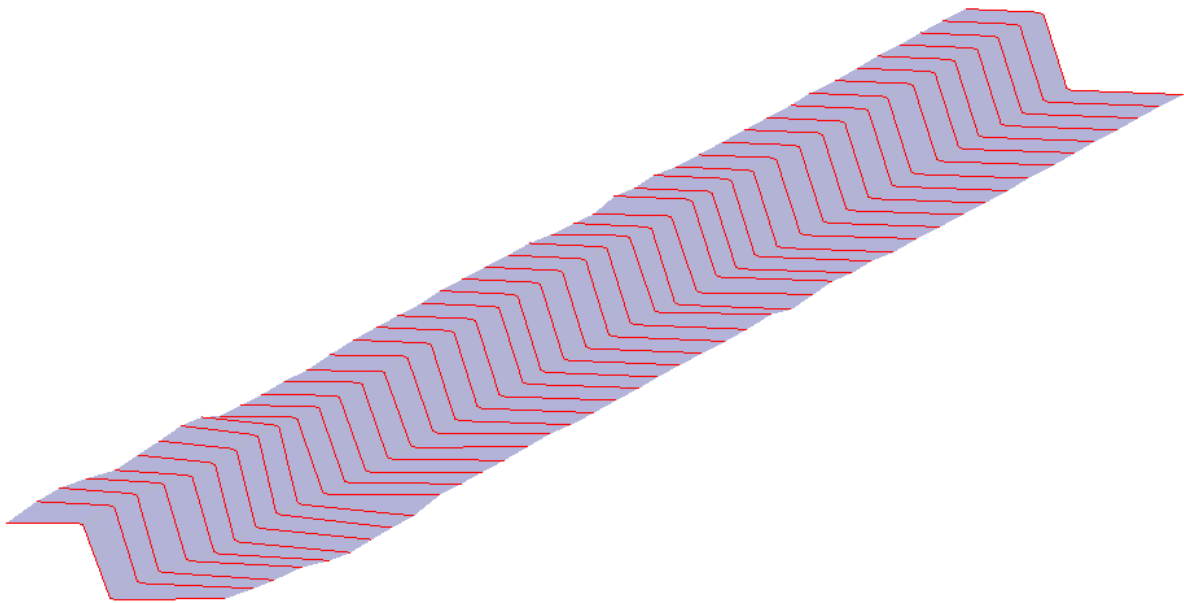
After the fitting process has been completed the following message will be displayed:



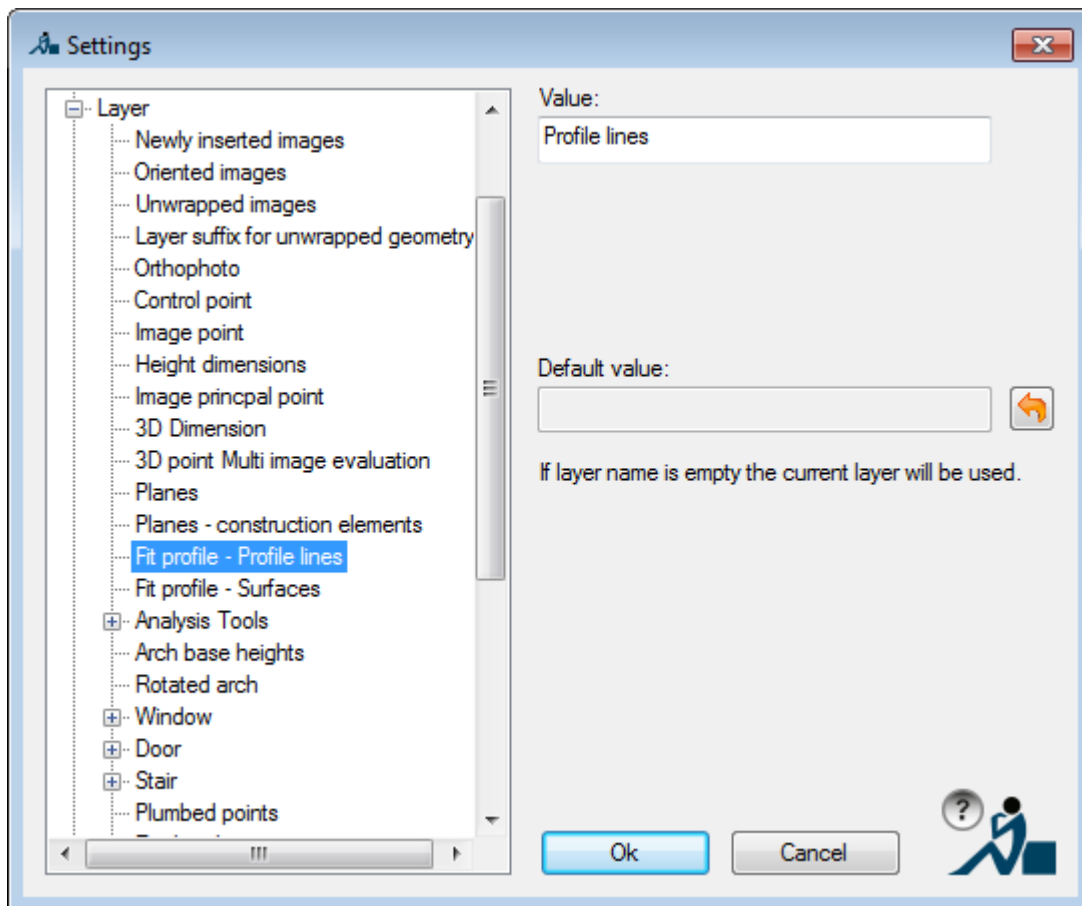
Now you can click on incorrectly fitted profile cross sections, so that they can be excluded from the creation of the surface:




As the result you receive in this example fitted profile cross sections and an AutoCAD surface:



The layers for the profile cross sections and the surfaces can, if required, be individually set under settings:



Modeling – Draw circle (3 points)

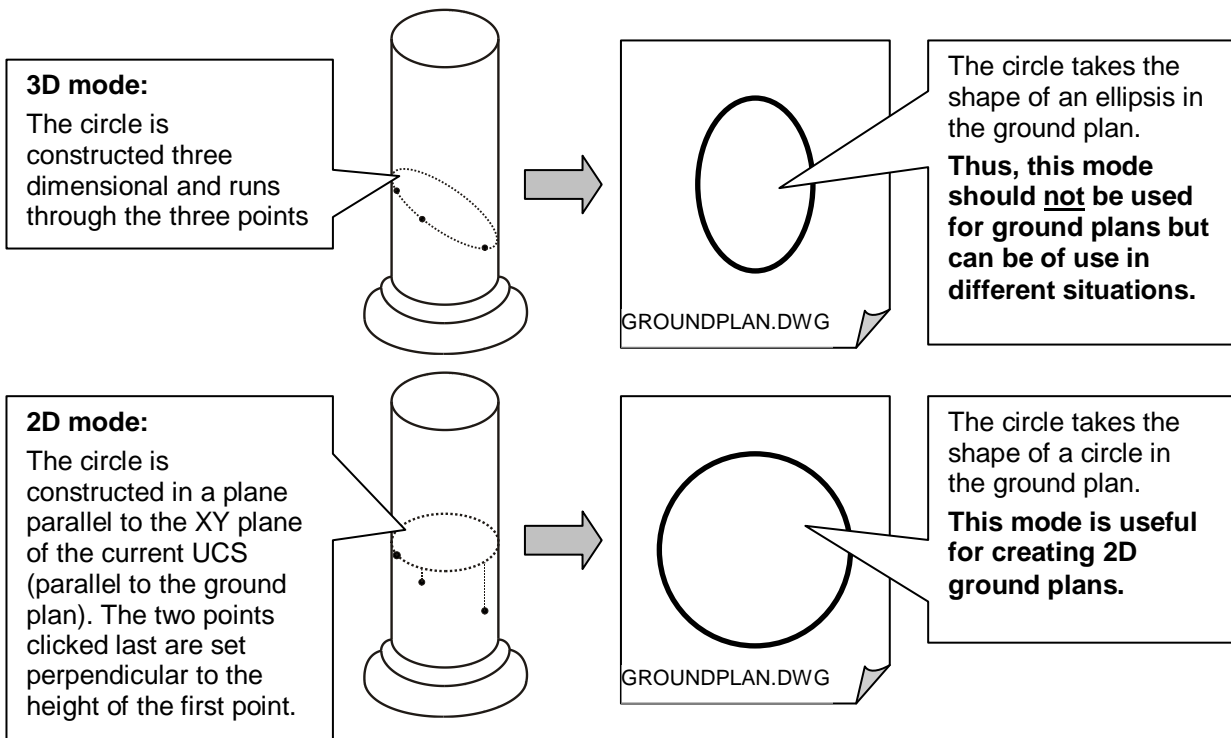
	Command name	PC3DCIRCLE
	Ribbon Tab:	As-Built Modeler

The command ‘PC3DCIRCLE’ draws a circle through three or two points to be given. You can choose between a real 3D circle (3D mode) and a projected circle (2D mode). The projected circle lies in a plane parallel to the XY plane of the current UCS with the Z coordinate of the first inserted point.

In contrast to the AutoCAD command ‘Circle’, which only accepts 2D coordinates, the FARO command ‘PC3DCIRCLE’ accepts 2D and 3D coordinates. Therefore, always use this command when you want to measure a circle with your instrument in the 3D mode.

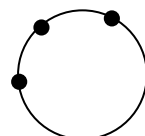
2D/3D mode

Following an example to explain the 2D and 3D modes further. A column is to be constructed. Its base should take the shape of a circle in the ground plan. Three points are clicked on the surface of the cylinder, (usually at different heights). The following outline shows the difference in reaction between both modes.

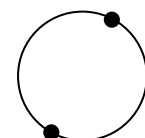


Construction 2points / 3points:

For 3points, any three points on the circle are clicked. With this the circle has been well-defined.




For 2points, two points have to be clicked which are exactly opposite each other. This option is advisable for the measurement of circular objects on the ground like covers or post holes (archaeology). This option is only available in the 2D mode.



The AutoCAD circle through the 3 (or 2) specified points will be created on the current layer.

Modeling – Draw arc (3 points)

	Command name	PC3DARC
	Ribbon Tab:	As-Built Modeler


The command “PC3DARC” draws a segment of an arc through three determined points. You can choose between a real 3D arc (3D mode) and a projected arc (2D mode). The projected arc lies in a plane parallel to the XY plane of the current UCS with the Z coordinate of the first determined point.

In contrast to the AutoCAD command “ARC”, which only accepts 2D coordinates, the FARO command “PC3DARC” accepts 2D and 3D coordinates. Therefore, always use this command when you want to measure an arc with your instrument in the 3D mode.

Also see → **Modeling – Draw circle (3 points)** for further explanation of the 2D and 3D modes. This command works in the same way, but instead of an arc a full circle is drawn through the measured points.

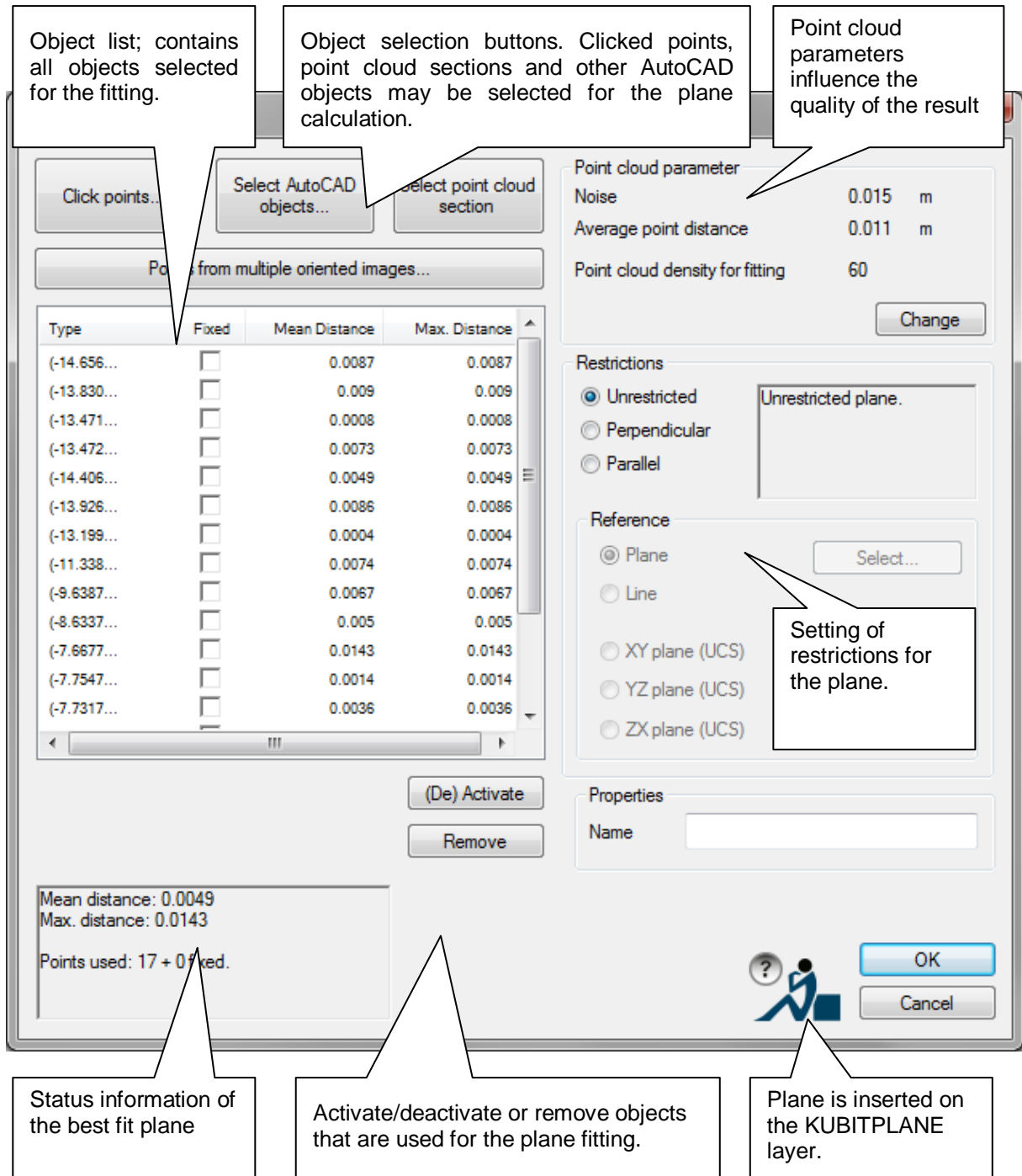
The AutoCAD arc through the 3 specified points will be created on the current layer.

Modeling – Plane – Fit plane

	Command Name:	KPFIT
	Ribbon Tab:	As-Built Modeler

A best fit plane is calculated for a selected set of points and is inserted into the drawing as kubit plane.

The calculation is done according to the least squares method. That means: the sum of the squares of the orthogonal distances of all selected points of the plane are minimized.



The dialog box is titled 'Fit plane' and contains several sections:

- Buttons:** 'Click points...', 'Select AutoCAD objects...', 'Select point cloud section', and 'Points from multiple oriented images...'. Callouts explain that the first three buttons are for object selection and that point cloud parameters influence the result quality.
- Table:** A table listing objects with columns for 'Type', 'Fixed', 'Mean Distance', and 'Max. Distance'. A callout identifies this as the object list.
- Point cloud parameters:** 'Noise' (0.015 m), 'Average point distance' (0.011 m), and 'Point cloud density for fitting' (60). A 'Change' button is located below these parameters.
- Restrictions:** Radio buttons for 'Unrestricted' (selected), 'Perpendicular', and 'Parallel'. A text box shows 'Unrestricted plane.'. A callout explains this is the setting of restrictions.
- Reference:** Radio buttons for 'Plane' (selected), 'Line', 'XY plane (UCS)', 'YZ plane (UCS)', and 'ZX plane (UCS)'. A 'Select...' button is next to the 'Plane' option.
- Properties:** A 'Name' text field.
- Actions:** '(De) Activate' and 'Remove' buttons.
- Status:** A box at the bottom left shows 'Mean distance: 0.0049', 'Max. distance: 0.0143', and 'Points used: 17 + 0 fixed.'. A callout identifies this as status information.
- Navigation:** 'OK' and 'Cancel' buttons at the bottom right. A callout notes that the plane is inserted on the KUBITPLANE layer.

When clicking OK, the kubit plane is inserted into the drawing. By default it is placed on the KUBITPLANE layer. By default the plane is indefinite and fixed in its supporting plane. Depending on the selected objects, the plane boundary results either from the boundary of the selected point cloud section (when exactly one point cloud section had been selected) or from the convex shell of all points and boundaries (for point cloud sections).

Object selection for the plane

You can use different types of objects for the plane calculation:

- Clicked points
- AutoCAD objects, lines and block references (insertion point)
- Point cloud sections
- Points from two oriented images (method spatial intersection); only available if the image management is turned on or at least two oriented images are in the drawing.

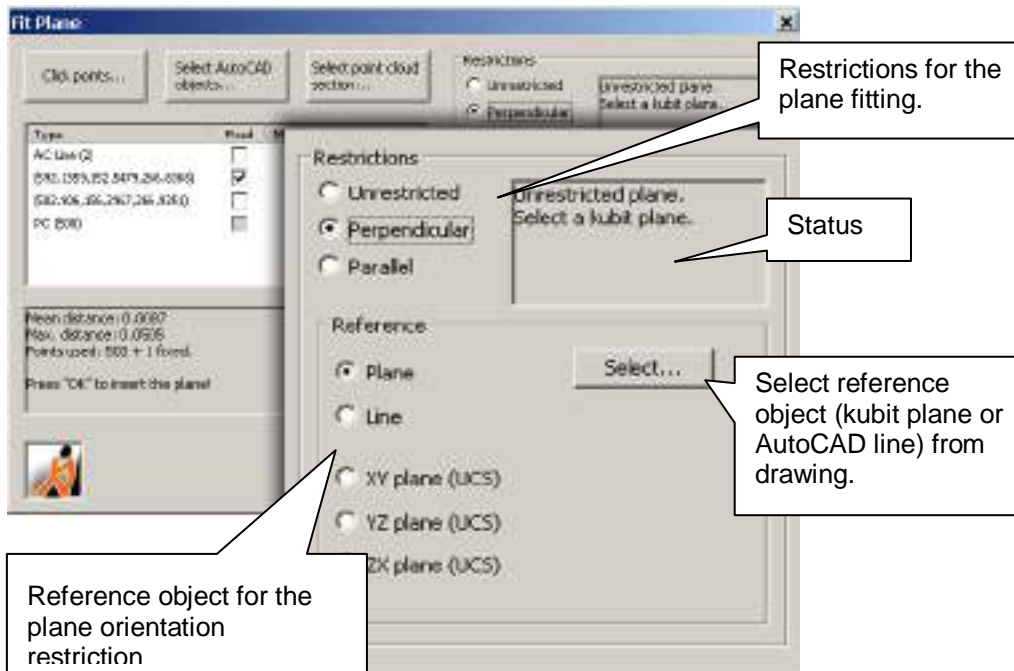
The object selection is controlled by three object selection buttons.

The objects used for the calculation are shown in the object list. The error information (mean and maximum distance from the plane) are displayed for each object in AutoCAD units. The accuracy of display depends on the AutoCAD settings. (Menu: Format → Units).

Certain objects (clicked points, AutoCAD points, block references and lines) can be fixed on the plane to be fitted. To do so activate the object in the column "Fixed" in the object list. The fixed objects become bases of the plane (error value=0). All other objects are used for the fitting process. The number of points that can be fixed depends on, among other factors, whether a restriction has been defined and if so which.

Restrictions


Here you can define certain restrictions for the plane to be fitted. To do so combine the alignment (parallel or vertical) with a reference object (kubit plane, AutoCAD line, UCS plane).



You find further information regarding the kubit plane in chapter **4.1 kubit Planes**.

Note: The quality of the fitting result can be influenced by setting the point cloud parameters (→**Point clouds – Set point cloud parameters...**).

Modeling – Plane – Plane (1-click)

	Command Name:	KUBT_ONE_CLICK_PLANE
	Ribbon Tab:	As-Built Modeler

With this command you can create a kubit plane with a single click, and, for example, use them in the later creation of façade drawings or solid models.

After starting the command:

Command: **KUBT_ONE_CLICK_PLANE**

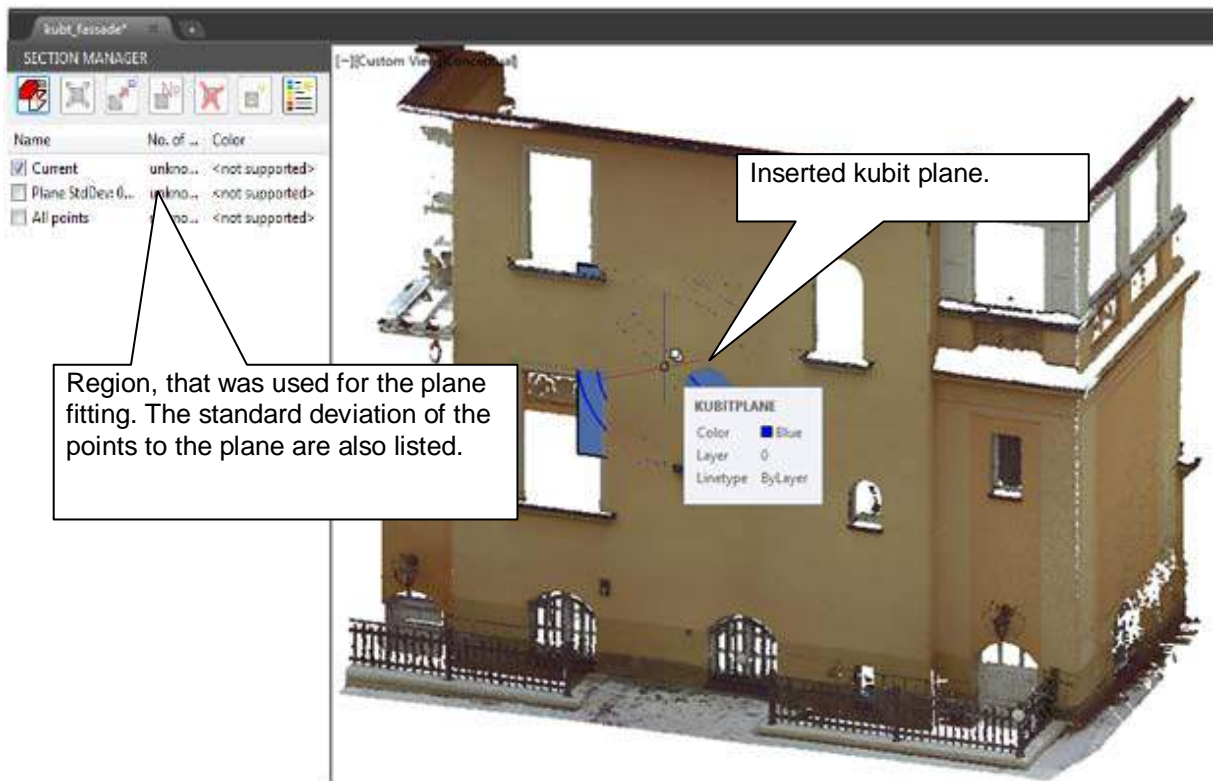
Select one point on (point cloud-) plane or [PArameter/Increase/Decrease] <PArameter>:

(Pick a point on the plane on which you want to model)

You will be prompted to pick a point on the plane on which you want to model:



When you have done this, around this position an optimum plane will be looked for, averaged and inserted into the drawing. For the points used in the plane fitting a section will be inserted in the section manager with the name “Plane fit, StdDev ...”, as long as the tick box in the parameter settings dialogue box has been activated:



The plane will be looked for within a search radius. The default value is set to 1.5 metre. The best value for the search radius should be approximately the size of the structure For the example above a facade e.g. 5 m.

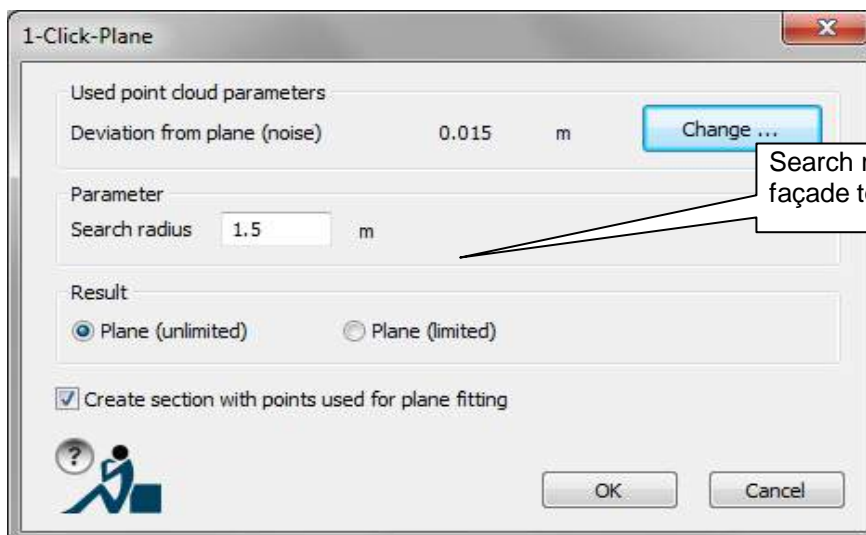
Run the command and select the option "Parameter":

Command: **KUBT_ONE_CLICK_PLANE**

Select one point on (point cloud-) plane or [PArameter/Increase/Decrease] <PArameter>: p

(Select the option Parameter)

A dialogue box opens:



There you can set the search radius in the “Search radius” box. Set the value to 5 m, for example and confirm it with [OK]. You will be again prompted to pick a point on the plane on which you want to model:

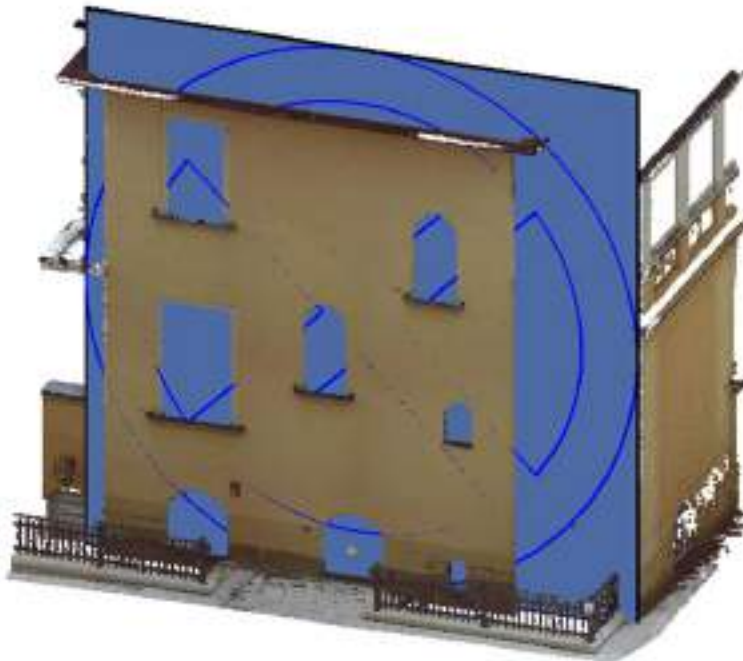
Command: **KUBT_ONE_CLICK_PLANE**

Select one point on (point cloud-) plane or [PArameter/Increase/Decrease] <PArameter>:

(Pick a point on the plane on which you want to model)

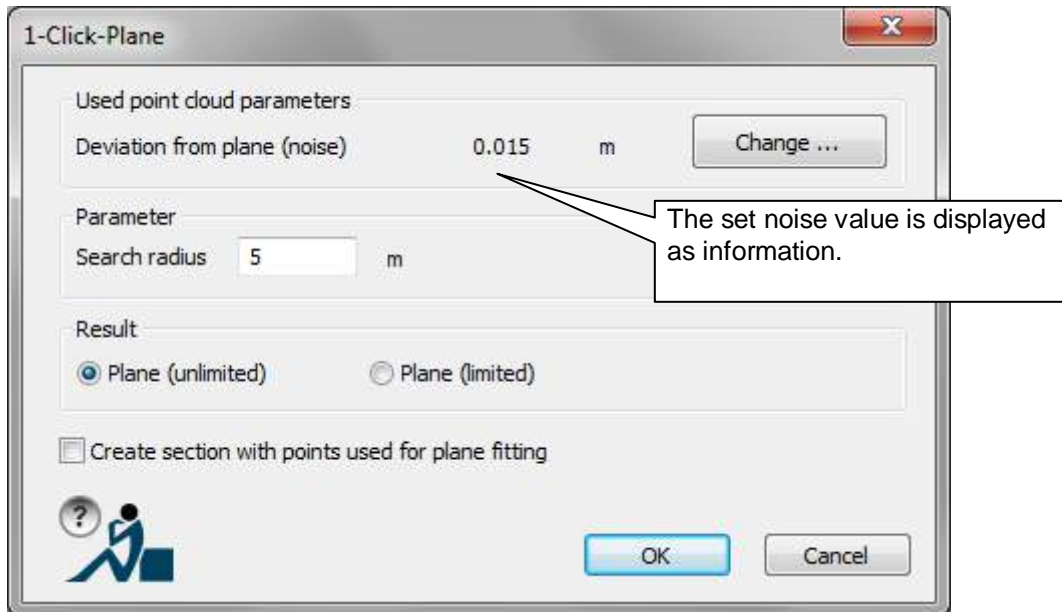


You can see that the size of the green circle representing the search radius has changed. The result is that you become an appropriately larger kubit plane.



Another possibility to change the size of the search radius is by the command-line options “Increase” and “Decrease”. Using this option, the search radius can be increased or decreased in approximately 20% steps.

As well as the search radius the command also uses the set value for the point cloud noise. This will be displayed as information in the Parameter dialogue box:



The point cloud noise is the deviation of the scanned plane from the ideal plane. The noise must take into account the surface properties (e.g. a textured render with at least 5 mm texture) and also the curvature properties of the surface (e.g. the swelling of a parquet floor of 5 mm). For the façade above a noise of 1 cm would be used. Your planes can deviate at the edges, within the set noise value, from the actually scanned plane, so try to set this value only as large as is necessary.


Output:

Hereby, you determine which AutoCAD object should be created from the computed plane.

- Unlimited plane: A user defined object of the type “KUBITPLANE” will be created. The plane will be considered as infinitely expanded.
- Limited plane: A user defined object of the type “KUBITPLANE” will be created. The plane is limited to the border of the plane.

You will find further information regarding the kubit plane in chapter **4.1 kubit Planes**.

Modeling – Plane – Draw plane

	Command Name:	KPCREATE
	Ribbon Tab:	As-Built Modeler

With this command you can draw a plane by clicking points. The first three selected points define the supporting plane. All further points just refine the boundary.

Select first point: *(Select first point)*

Select second point: *(Select second point)*


Select point: *(Select point*

and, if necessary, further points for the boundary)

The plane is then pasted into the AutoCAD drawing. By default it is put on the layer KUBITPLANE. The plane is indefinite and locked in its supporting plane.

You will find further information regarding the kubit plane in chapter **4.1 kubit** Planes.

Modeling – Plane – Intersection line (2 Planes)

	Command Name:	KPINTERSECT
	Ribbon Tab:	As-Built Modeler


This command creates the line of intersection between two kubit planes. For this the planes must not be parallel.

Select first plane for intersection:

Select second plane:

You will find further information regarding the kubit plane in chapter **4.1 kubit** Planes.

Modeling – Plane – Intersection point (3+ Planes)

	Command Name:	KPINTERSECTPOINT
	Ribbon Tab:	As-Built Modeler

This command creates the point of intersection between three or more kubit planes. The planes must not be parallel in pairs.

Select planes for creating intersection point (at least 3, pair wise non-parallel):


If more than three planes have been selected, a mean point of intersection is created. This point results from the center of all points of intersection of three planes each. In this case error values are displayed:

mean distance between planes and intersection point: 0.03

longest distance between planes and intersection point: 0.12

You will find further information regarding the kubit plane in chapter **4.1 kubit Planes**.

Modeling – Plane – Intersection lines (3 Planes)

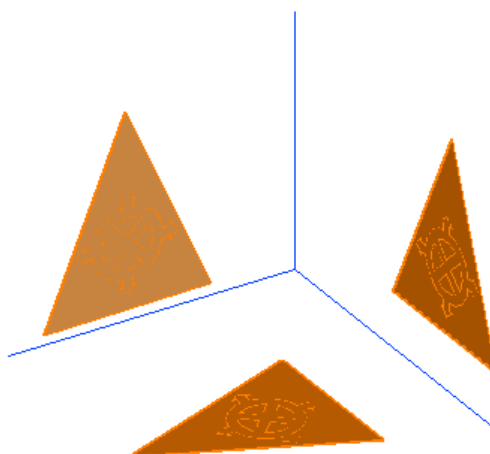
	Command Name:	KPINTERSECTTRIAD
	Ribbon Tab:	As-Built Modeler

This command creates a triad of intersection lines between three kubit planes. The planes must not be parallel in pairs.

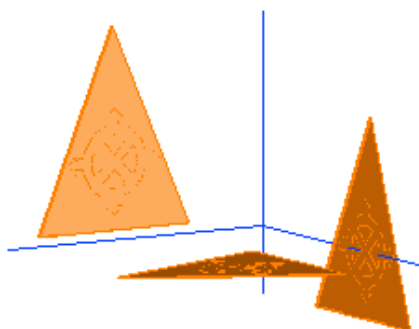
Select first plane:

Select second plane [UNdo]:

Select third plane [UNdo]:




The three lines for the edges are drawn on the current layer. The length of these lines is defined by the displayed boundaries of the kubit planes. Due to this fact it may happen that the edges of the triad do not end at the intersection point - as one might expect. See figure below.



You find further information regarding kubit planes and advice on how to model 3D using this command in **chapter 4** of this manual.

Modeling – Plane – Perpendicular projection of points onto plane

	Command Name:	KPPERPRETURN
	Ribbon Tab:	As-Built Modeler

With this command you may project 3D points onto a kubit plane. The result is an AutoCAD point object.

Select projection plane:


Direction of return:

(snap or measure a point)

Direction of return: *Cancel*

You find further information regarding kubit planes and advice on how to model 3D using this command in **chapter 4** of this manual.

Modeling – Plane – Extend (2 Planes)

	Command Name:	KPEXTEND
	Ribbon Tab:	As-Built Modeler

This command extends and trims two kubit planes to their common trimming edge. The processes of extending and trimming run symmetrically so that it does not matter which plane is selected first.

Select first plane to trim or extend:


Select second plane:

Note

You can also use planes to extend or trim line objects (lines, arches, and polylines). Use the AutoCAD commands Extend and Trim.

You will find further information regarding the kubit plane in chapter **4.1 kubit** Planes.

Modeling – Plane – Automatic rim

	Command Name:	KUBIT_REGION_GROW
	Ribbon Tab:	As-Built Modeler

With this command you can automatically define the border of a plane, for example to create façade drawings or to model an object from a compound collection of planes. To this end the command grows the plane by means of the point cloud plane. You can grow both an already existing kubit plane and also with just a single click define a new plane and grow it.

Alternative 1 (grow an already existing plane):



After starting the command:

Command: **KUBIT_REGION_GROW**

Select a plane or [PArAmeter/Automatic plane]:

(Select

The kubit plane that you want to bound)

Select the plane that you want to automatically border. Thereafter you will be prompted to pick a point inside of the point cloud plane which should be used to constrain the kubit plane.

Select one point on the (point cloud-) plane or [Back/PArAmeter]:

(Select

a point on the point cloud plane)

Select here for the above façade, for example, a point anywhere on the façade.

Alternative 2 (a new grown plane with a single click)



After starting the command:

Command: **KUBIT_REGION_GROW**

Select a plane or [PArmeter/Automatic plane]:

(Select

The option "Automatic plane")

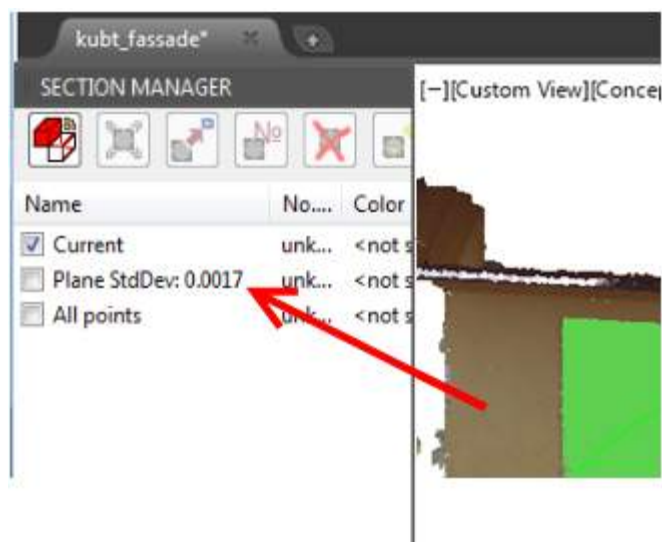
Select the option "Automatic plane". You will be prompted to pick a point on the point cloud plane, that you want to have modelled as grown kubit plane

Select one point on (point cloud-) plane or [PArmeter/Select plane/Increase/Decrease]

<PArmeter>:

(Select a point on the point cloud plane)

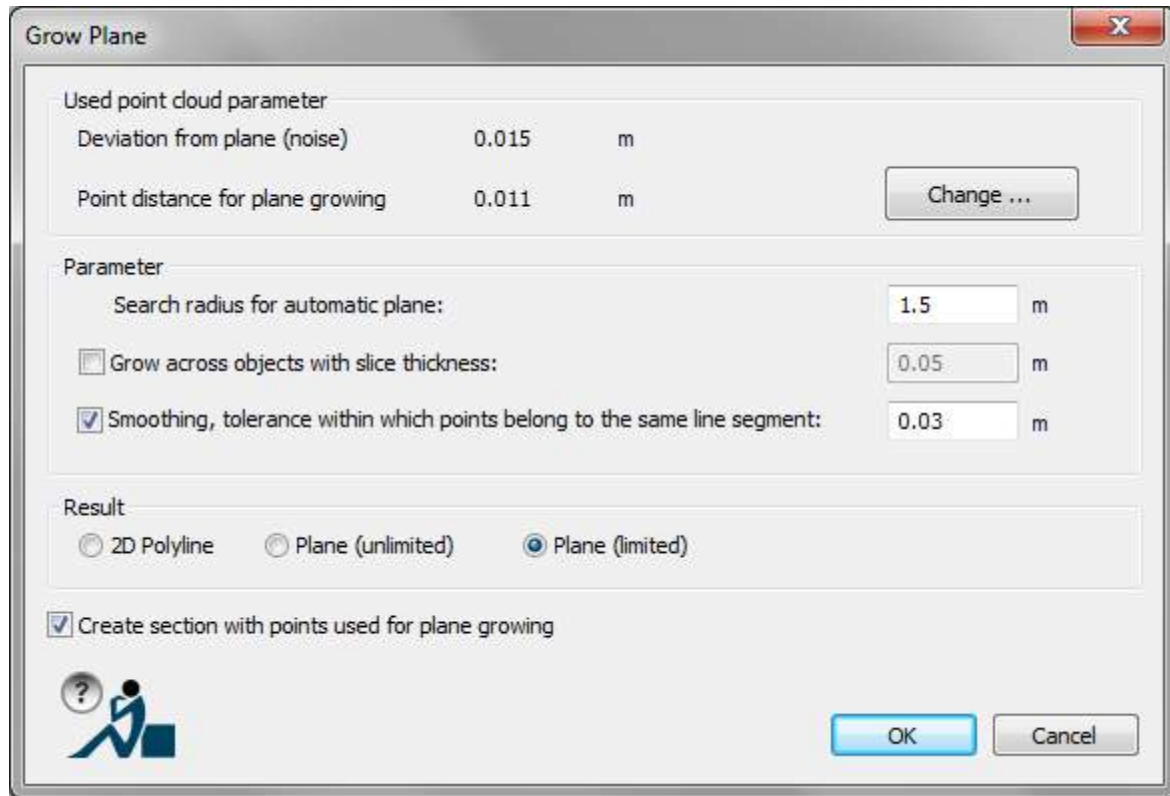
Thereupon you become the bordered kubit plane ensuing from the scanned point cloud plane. The command looks for the best plane and then averages it out. You will find the point data used in the fitting computation including the standard deviation data in the section manager under the name "Plane StdDev ...", as long as the tick box in the parameter settings dialogue box has been activated:



Parameters used for Alternative 2 (a new grown plane with a single click):

The command uses various parameters that can influence the quality of your results.

After starting the command select the option “Parameter”. The following parameter dialogue box opens:



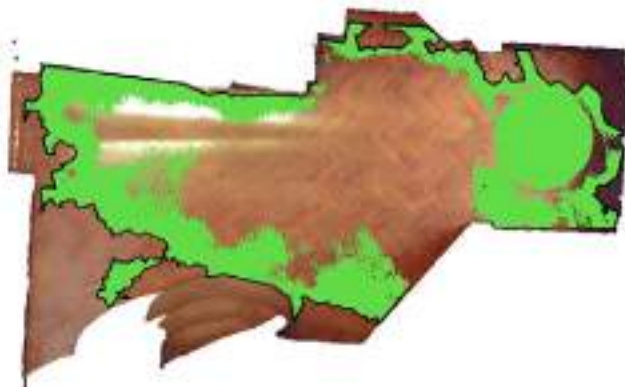
The alternative 2, which automatically creates a new plane with just a single click, uses when searching for the plane the parameters “Deviation from the plane (noise)” and “Search radius for automatic plane”.

Deviation from the plane (noise):

This parameter defines how large the deviation of the scanned point cloud plane can be from the ideal plane. This value must incorporate both the surface properties (e.g. a strongly textured rough render with 5 mm deviation) and also the curvature properties of the surface (e.g. an additional 5 mm deviation for the swelling of a floor).

The plane that has been found can deviate at the edges by about this set value. If this value is too large, then the ideal plane that has been found will be imprecise.

If the value is too small, then the surface can be only partially recognised and will be fringed towards the edges.



Search radius:

This parameter defines within which radius the plane should be looked for. This value should be approximately the size of the structure to be searched. For example for the above façade about 5 m. As the command automatically adapts this value, it mainly influences the speed of computation. The search radius can be can also be changed with the command-line options “Increase” and “Decrease”. Using these options, the search radius can be increased or decreased in approximately 20% steps.

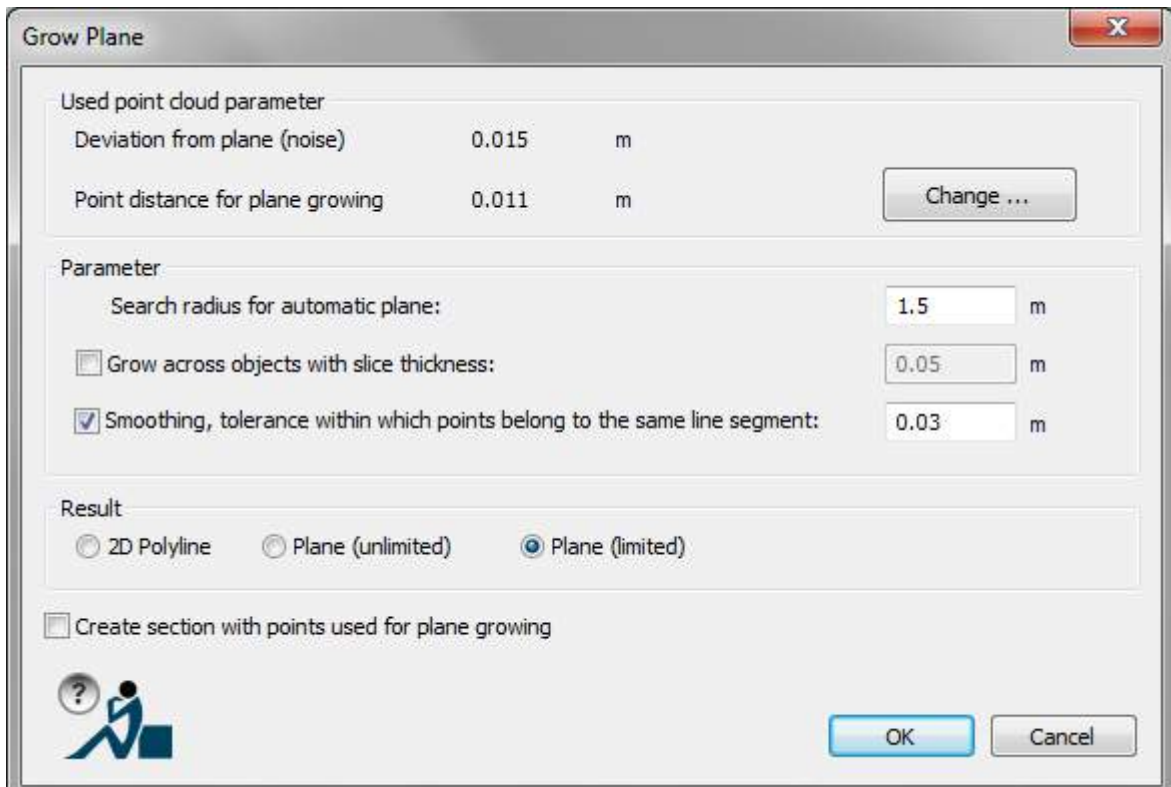
Output:

Hereby, you determine which AutoCAD object should be created from the expanded plane.

- 2D Polyline: The plane’s border is represented by a 2D polyline.
- Unlimited plane: A user defined object of the type “KUBITPLANE” will be created. The plane, when displayed, is represented by the computed border, but internally considered as infinitely expanded.
- Limited plane: A user defined object of the type “KUBITPLANE” will be created. The plane is limited to the computed border of the plane.

Parameters used by both alternatives:

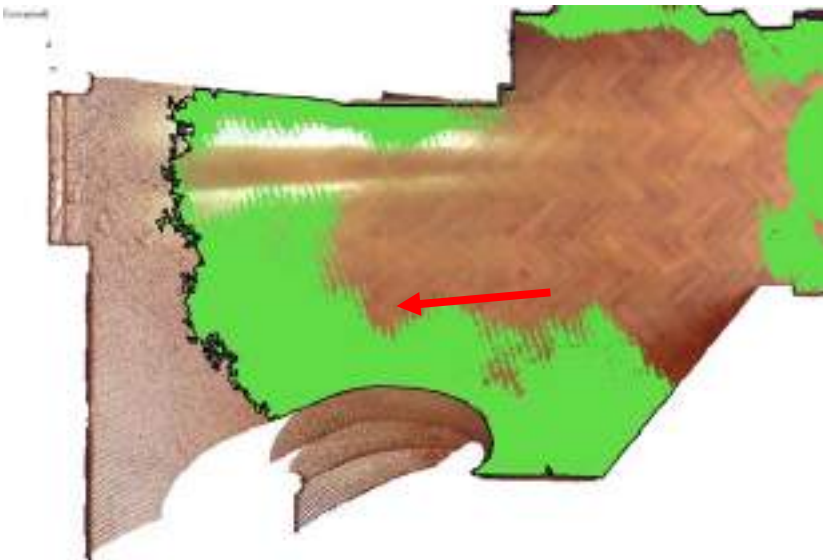
As well as the parameters that have just been explained “Deviation from the plane (noise)” and “Search radius for automatic plane”, all of the other parameters are used by both alternatives. You can also influence the result with the assistance of these parameters.



Point distance for plane growing:

This value defines how far points in your point cloud should be from one another so that they can be assumed to be related.

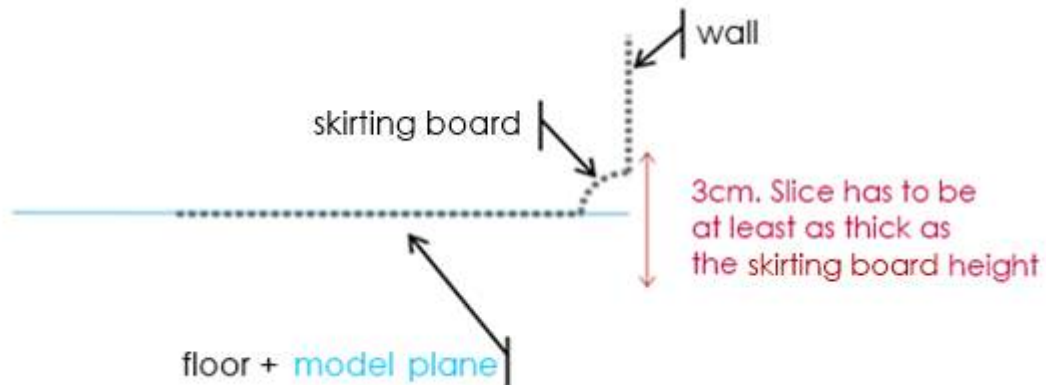
If this value is too small, then the plane will break off too soon at the edges. The plane will be fringed on the edges (holes will no longer be considered as being related):



If the value is too large, the border will be imprecise.

Grow across objects with slice thickness:

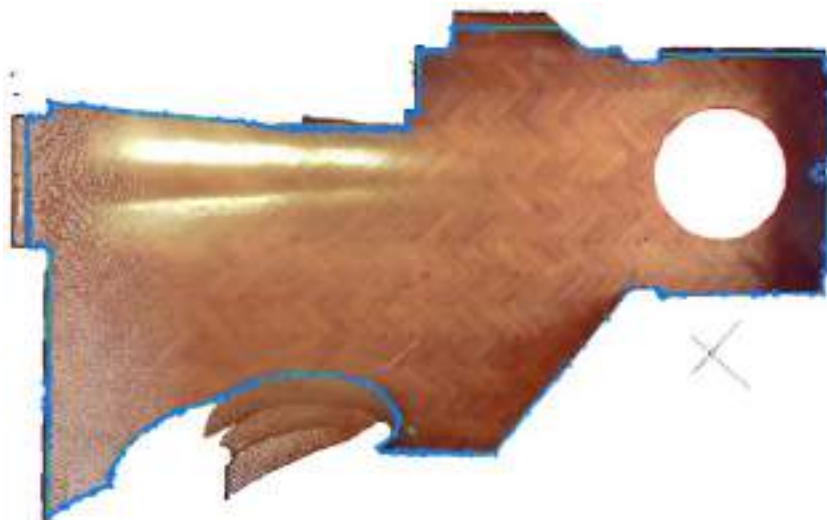
By default the points will be used to grow the plane that are above and below the plane and within the noise value. Should you wish, for example, to contour a room up to the walls, then you must consider that on the edges of the walls there are skirting boards. The plane border would normally end on these skirting boards. Which points, that lie above and below the plane, that are used to allow this object to grow outwards, will be influenced by this parameter.



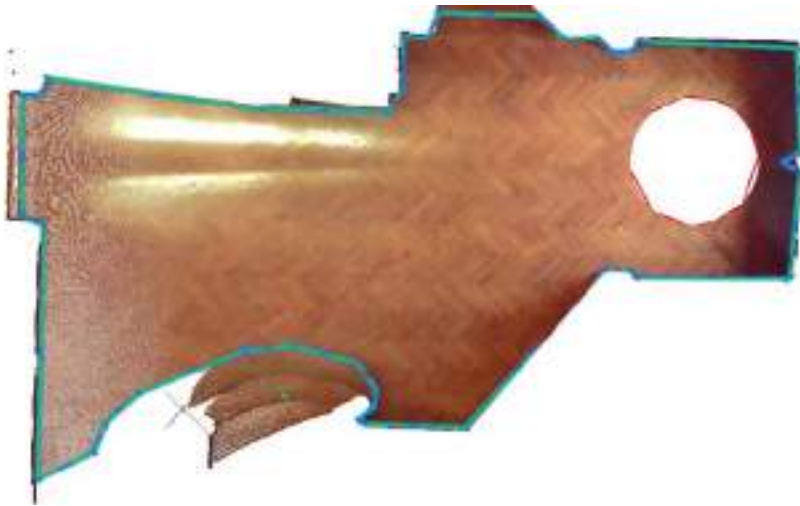
Smoothing, tolerance within which points belong to the same line segment:

This value controls by how much the resulting border should be smoothed. The larger this value is the fewer vertices has the resulting border.

For example, on a parquet floor with a smoothing deviation of 1 cm the following resulting border has 150 vertices:



However if you use a smoothing deviation of 3 cm, then the resulting bounding polygon has only 63 vertices:




Output options:

You can select between a 2D polyline or a kubit plane as output. If you choose the option 2D polyline, the holes within the surface will be displayed as red polylines.

You will find further information regarding the kubit plane in chapter **4.1 kubit** Planes.

Modeling – Plane – New Rim

	Command Name:	KPNEWRIM
	Ribbon Tab:	As-Built Modeler

This command allows you to draw a new boundary (rim) for a kubit plane. The supporting plane as well as all properties of the plane are kept when doing so.

Select a kubit plane: *(Select that plane for which you want to draw a new boundary)*

Select first point:


Select second point:

Select point: *(Draw the new boundary.)*

The clicked points will be projected onto the Supporting plane along the view ray.)

You will find further information regarding the kubit plane in chapter **4.1 kubit** Planes.

Modeling – Plane – Modify Rim

	Command Name:	KPMODIFYRIM
	Ribbon Tab:	As-Built Modeler

With this command you can modify the boundary of a kubit plane; for example “cut” an edge or enlarge the boundary on one side.

Select a kubit plane:

Click point on plane's rim:

(Click a point near the boundary)

Should the clicked point be too far away from the edge, the following error message will appear:

The point is not on the plane's rim!

Click point on plane's rim:

(Click closer to the edge.)

Else you can continue with changing the boundary:

Select point for plane's rim or [Finish]:

Select point for plane's rim or [Finish/Back]:

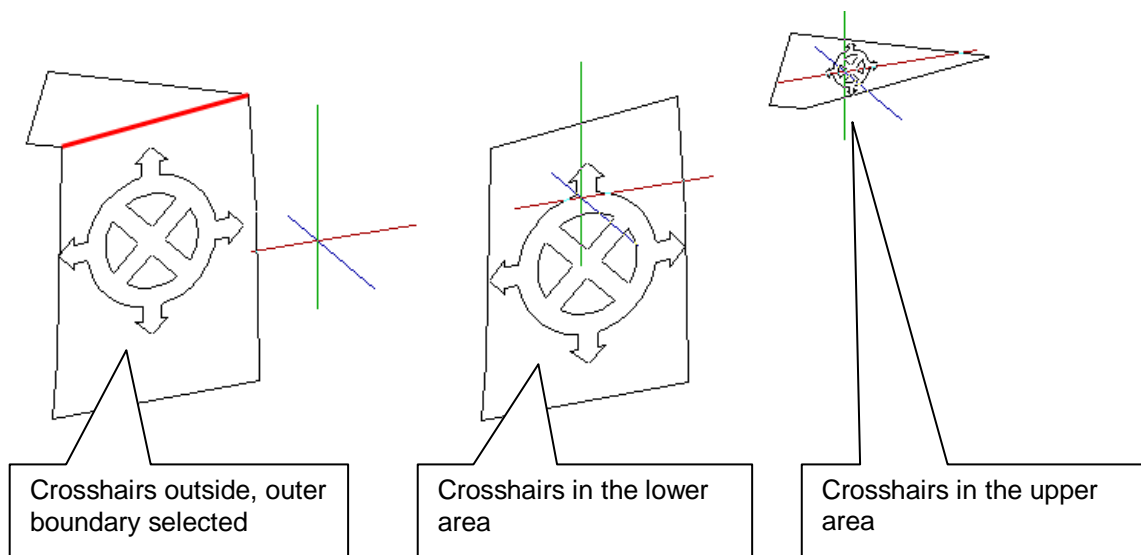
(Draw the new part of the boundary.)

Select Finish to finish the command.)


Should the last point already lie near the boundary, the command will be finished. Otherwise you are prompted again to click a point on the edge (or near it respectively).

To finish click point on the rim.

Now you have to decide which part of the plane you want to keep. In order to do so move the crosshairs inside or outside the respective boundary. Depending on where the crosshairs is situated, one of the three possible boundaries will be displayed. Then confirm your selection by clicking the left mouse button.



Modeling – Cylinder – Fit cylinder using 2 click points

	Command Name:	PCCC
	Ribbon Tab:	As-Built Modeler

You can generate an adjusting cylinder through a set of points using this command.

For estimating the rough position of the cylinder you click two points on the cylinder surface. Ensure you have enabled the object snap “node”.

Command: PCCC

>> Cylinder <<

Specify point on cylinder or [paraMeters] <paraMeters>: *(Snap first point on the cylinder surface)*

Specify point on cylinder or [paraMeters] <paraMeters>: *(Snap second point on the cylinder surface)*

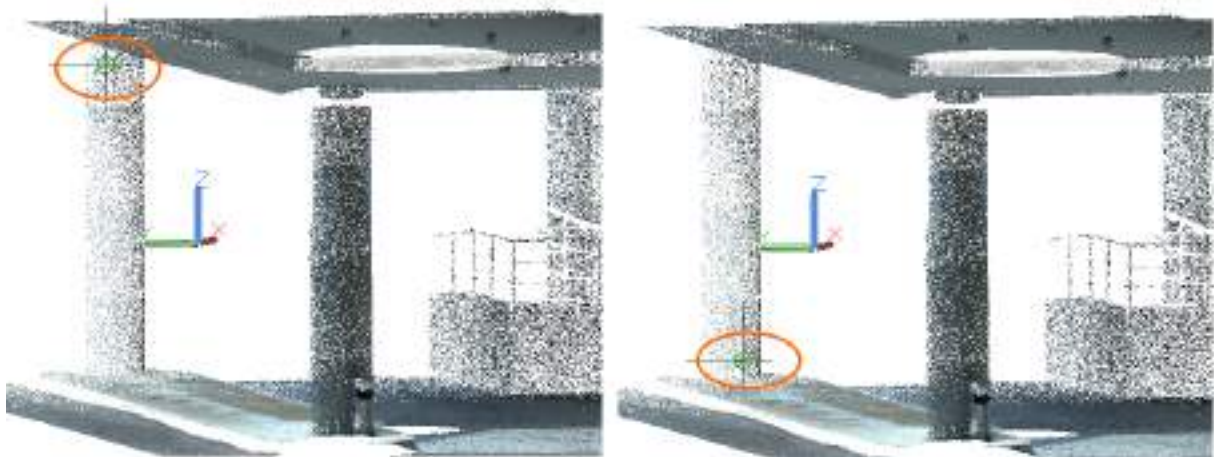
The cylinder will be fitted and a kubit cylinder is inserted in the drawing.

If the fitting fails following message appears:

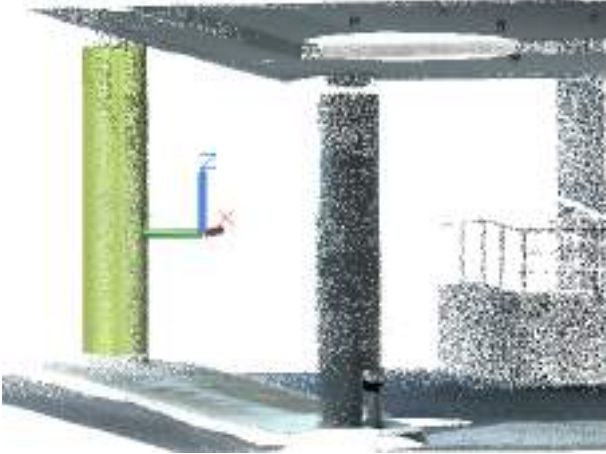
No cylinder detected. Please, check point cloud parameters (noise etc.)

There are different reasons for this, e.g.

- The point cloud parameters are not set correctly → **set point cloud parameters, can be accessed directly via command line option “paraMeters”**
- You did not snap the points on the surface of the cylinder to be fitted. Choose a different view or cut out a part of the cloud to help you snap the points properly. If you work with point clouds in PCG format you may also create a “Smart Section“ using the section manager.



1) Click two points on the cylinder surface




2) Cylinder fitted

If the cylinder fitting fails repeatedly with this command, you can use the command "PCFITCYL2" (→Modeling – Cylinder – Fit cylinder using visible points (clipped)). This command works with an exact cut of the point cloud sections which belong to the cylinder.

Note: The quality of the fitting result can be influenced by setting the point cloud parameters (**command line option "paraMeters" →Set point cloud parameters...**).

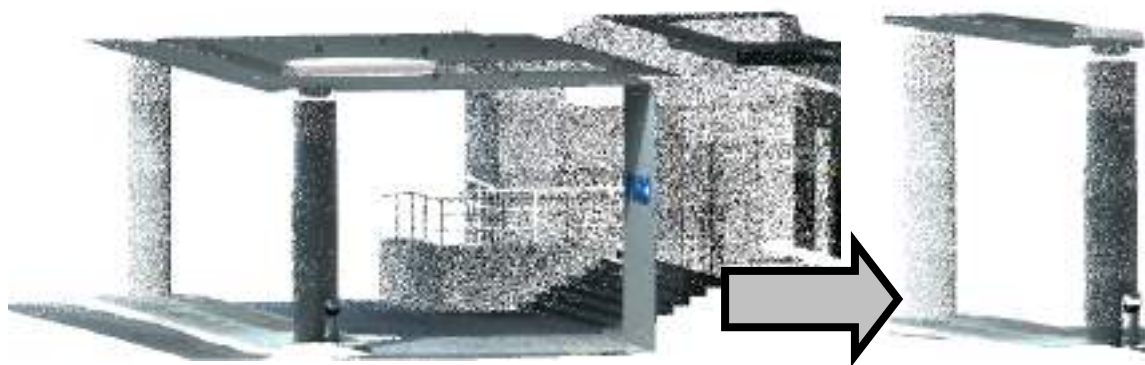
Modeling – Cylinder – Fit cylinder using visible points (clipped)

	Command Name:	PCFITCYL2
	Ribbon Tab:	As-Built Modeler

You can generate an adjusting cylinder through a set of points using this command.

Cut points belonging to the cylinder using the general commands to define section definitions. Single points that do not belong to the wanted cylinder, as for example a valve or a fragment of an adjacent cylinder, will automatically be filtered out.

Hint: If you want to fit many cylinders one after the other, a successive approach is recommended, e.g. beginning with the definition of a slice including many cylinders.



Name the slice in the section manager and then cut out several cylinders. Using the section manager you may always go back to the named slice to cut out additional cylinders. Furthermore, you may turn back to the previous section with the command →**Sections – Previous section definition**.



Fit the cylinders with command: **PCFITCYL2**. If there are more than 100.000 points in the selection you will be warned and asked whether you want to continue the fitting process.

Now, the outliers will be filtered out of the visible point set and the optimal cylinder will be calculated by minimizing the square distances.

Command: PCFITCYL2

Errors: 2.458 (mean) 18.9348 (max)

(error values in AutoCAD units)

Used points: 12113, Outliers: 404

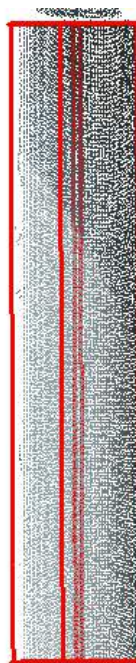
(adaptation information)

Finish (ESC or ENTER)?

(ENTER key, in order to generate the cylinder,

ESC in order to abort, the cylinder will not be generated)


The result will be an AutoCAD cylinder of the "kubit cylinder" type with fixed axis on the current layer.



You find further information according kubit cylinders in chapter **4.3 kubit** Cylinders of this manual.

Note: The quality of the fitting result can be influenced by setting the point cloud parameters (→**Set point cloud parameters...**).

Modeling – Cylinder – Fit cylinder

	Command Name:	KCFIT
	Ribbon Tab:	As-Built Modeler

A best fit cylinder is calculated for a selected set of points and is inserted into the drawing as kubit cylinder.

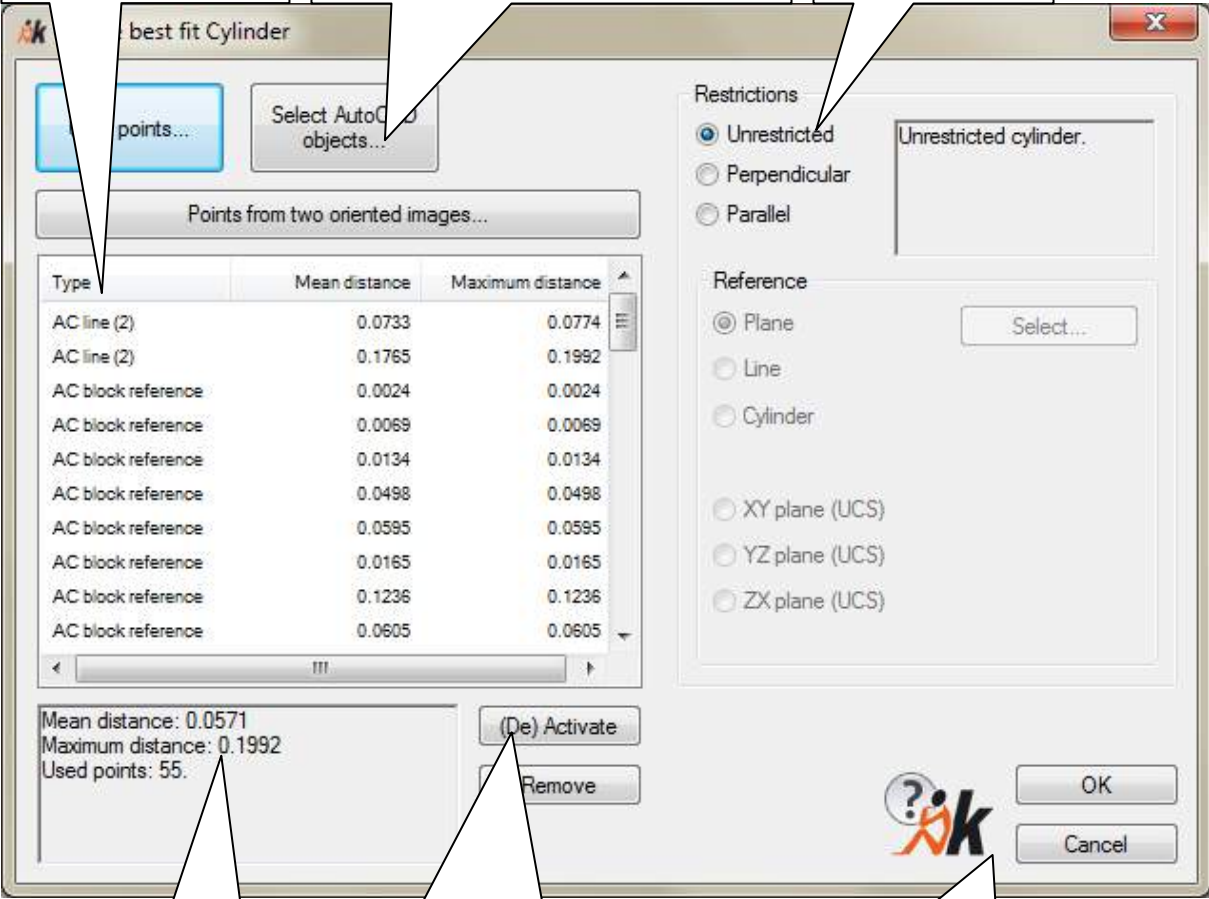
For a free cylinder at least nine points on the lateral surface are needed for fitting. With restrictions three points on the lateral surface are sufficient.

The calculation is done according to the least squares method. That means: the sum of the squares of the orthogonal distances of all selected points from the surface of the resulting cylinder is minimized.

Object list; contains all objects selected for the fitting.

Object selection buttons. Clicked points and other AutoCAD objects may be selected for the cylinder calculation.

Setting of restrictions for the cylinder.



Status information of the best fit cylinder

Activate/deactivate or remove objects that are used for the plane fitting.

Cylinder is inserted on the current layer.

When clicking [OK], the kubit cylinder is inserted into the drawing. By default it is placed on the current layer. The cylinder axis is locked by default. The extent of the cylinder results from the convex shell of all points used.

Object selection for the cylinder

You can use different types of objects for the cylinder calculation:

- Clicked points
- AutoCAD objects, points, lines and block references (insertion point)
- Points from multiple oriented images (method spatial intersection); only available if the image management is turned on or at least two oriented images are in the drawing.

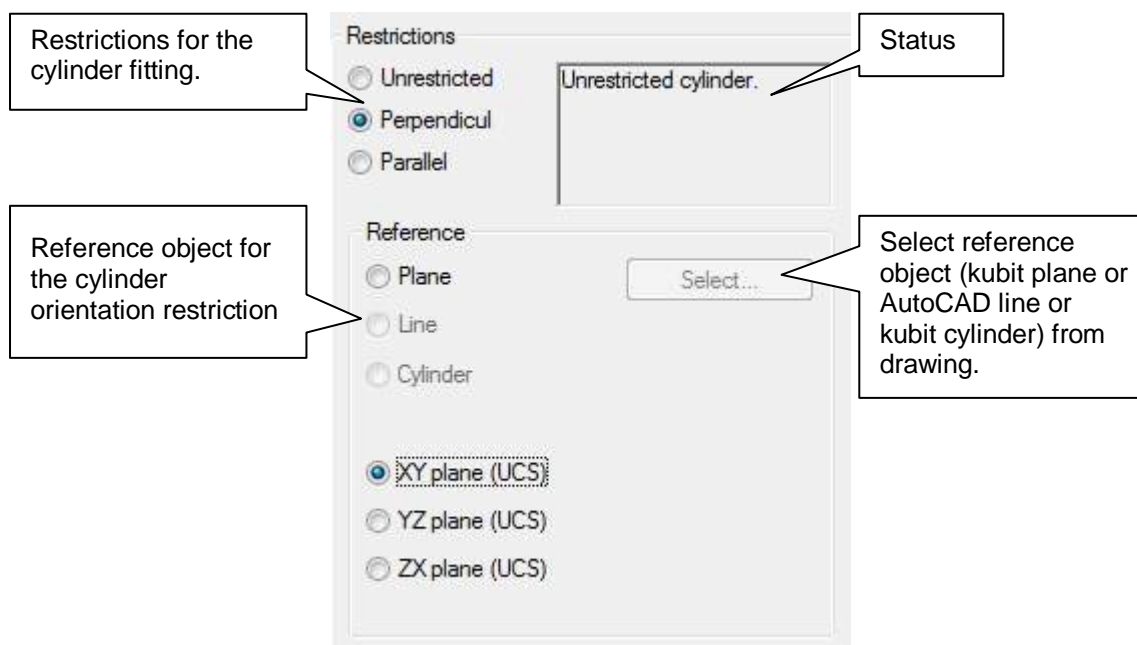
The object selection is controlled by the object selection buttons.

The objects used for the calculation are shown in the object list. The error information (mean and maximum distance from the cylinder surface) are displayed for each object in AutoCAD units. The accuracy of display depends on the AutoCAD settings. (Menu: Format → Units).

Certain objects (clicked points, AutoCAD points, block references and lines) can be fixed on the cylinder surface to be adapted. To do so activate the object in the column “Fixed” in the object list. The fixed objects become bases of the cylinder surface (error value=0). All other objects are used for the fitting process. The number of points that can be fixed depends on, among other factors, whether a restriction has been defined and if so which.


Restrictions

Here you can define certain restrictions for the cylinder to be fitted. To do so combine the alignment (parallel or vertical) with a reference object (kubit plane, kubit cylinder, AutoCAD line, UCS plane).



You find further information according kubit cylinders in chapter **4.3 kubit** Cylinders of this manual.

Modeling – Cylinder – Draw cylinder

	Command Name:	KCNEW
	Ribbon Tab:	As-Built Modeler

By clicking the axis points and stating the radius you generate a new cylinder with this command.

Command: KCNEW

Specify start of cylinder axis:

Specify end of cylinder axis:

Diameter <1.000.000>

*(pull the cylinder to the wanted size,
enter the diameter*


or ENTER key in order to take over the last diameter)

You may change the attributes of the cylinder at any time via the AutoCAD properties window.

The generated cylinder is not fixed to its axis.

You find further information according kubit cylinders in chapter **4.3 kubit** Cylinders of this manual.

Modeling – Cylinder – Generate AutoCAD 3D solid

	Command Name:	KCSOLID
	Ribbon Tab:	As-Built Modeler

You may transform kubit cylinders into AutoCAD solids with this command.

Command: KCSOLID

Select cylinder:

Select objects: *(click on cylinder to be transformed)*

Erase or keep original cylinders? [Keep/Erase] <Keep>: K

(K or ENTER key, in order to keep the original cylinder,


E in order to delete the original cylinder.)

1 cylinder(s) successfully converted.

Kubit cylinders serve as design aid. They are custom entities, with which you can only work when using the our software. If you want to forward the project to a third party, you have to change the objects into a format, which is compatible with the bare AutoCAD. Therefore the last step in editing the measured cylinders is to transform them into AutoCAD solids. Those cannot be changed in their geometry. Therefore the transformation should only take place when the geometry is correct.

You will recognize the transformation, when the cylinders do not look hollow but solid.

Modeling – Truncated cones – Fit circular truncated cone

	Command name:	FCCREATECIRCLE
	Ribbon Tab:	As-Built Modeler

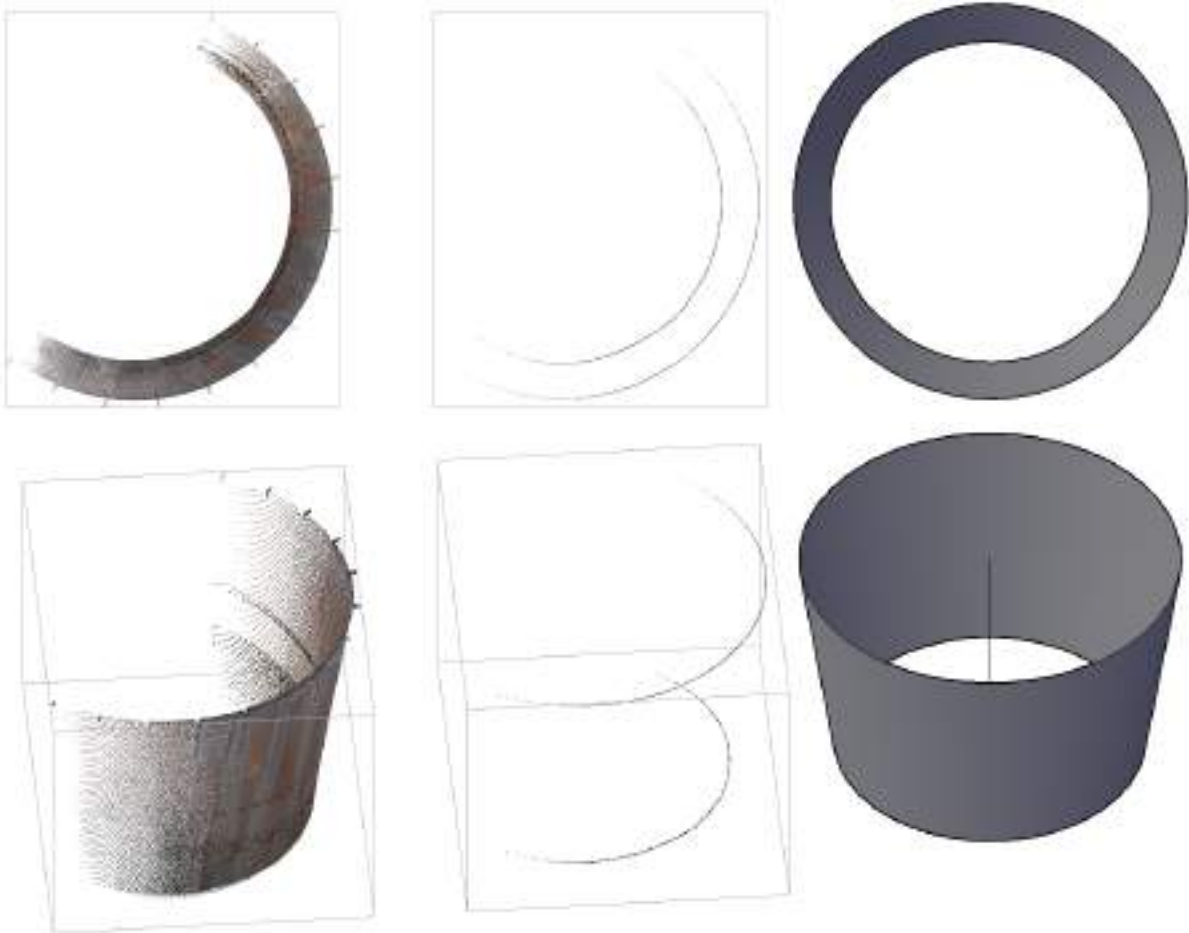
Truncated cones with a circle as their base shape can be modelled with this function. This kubit cone is especially suitable for modelling from point clouds. The cone's axis can be locked, so that when stretching or shortening lengthwise the position of the surface skin on the point cloud stays as it is (AutoCAD properties window). Furthermore kubit cone can be used as a reference surface for a deformation analysis and to create a deformation true solid (→**Modeling - Analyze frustum of a cone in a point cloud**).

The creation of the cone is done in three steps:

1. As the first step, the circle is defined in 3D space, that will determine the profile and the position of the axis of the cone. The circle requires at least three construction points on the circumference of the circle. From the selected points the average plane is determined, to which the cone's axis will be perpendicular, and then the circle is computed. With more than three points the averaged circle is determined. The number of points that have already been selected is displayed during input.
2. The height from the profile either upwards or downwards as well as the slope angle of the surface skin is determined from the subsequently entered points. It is sufficient to use just one point. For better accuracy it is recommended to select multiple points. These points have no influence on the cone's axis.
3. At the end it is stated whether the cone's axis, when creating the cone, should be fixed (that means locked). "Fixed axis", as used here, means that the truncated cone can only be modified in its height. This property can also be separately edited in the properties dialog.

For an upright standing cone shaped tank one would proceed as follows:


1. The point cloud will be loaded that will be used to model the scanned tank (→Point cloud - Insert point cloud).
2. Next the visible point cloud region would be cut back to two thin slices. For this use the command →**Sections - Define slice (UCS)** (PCS) or →**Sections - Define multiple slices** (PCMSC)
3. Now the truncated cone is created. Here, to determine the circular profile, a few points in the lower thin point cloud slice are selected. To define the height, the points in the upper thin slice are used. The cone's axis should be fixed (locked).
4. Lastly, the whole of the point cloud region is again made visible and the grips to stretch the already created kubit cone can be used to again move the upper and lower cap surfaces to fit it onto the whole of the cone shaped region of the point cloud. Thereby the position of the surface skin and the axis stay as they were, it's just the length that is adjusted.



Shown in the figure above, from left to right, in top view and as perspective view, are the following:

- the point cloud of the scanned truncated cone (tank or tower),
- the two thin point cloud slices and
- the resulting circular truncated cone.

Modeling – Truncated cones – Fit elliptical truncated cone

	Command name:	FCCREATEELLIPSIS
	Ribbon Tab:	As-Built Modeler

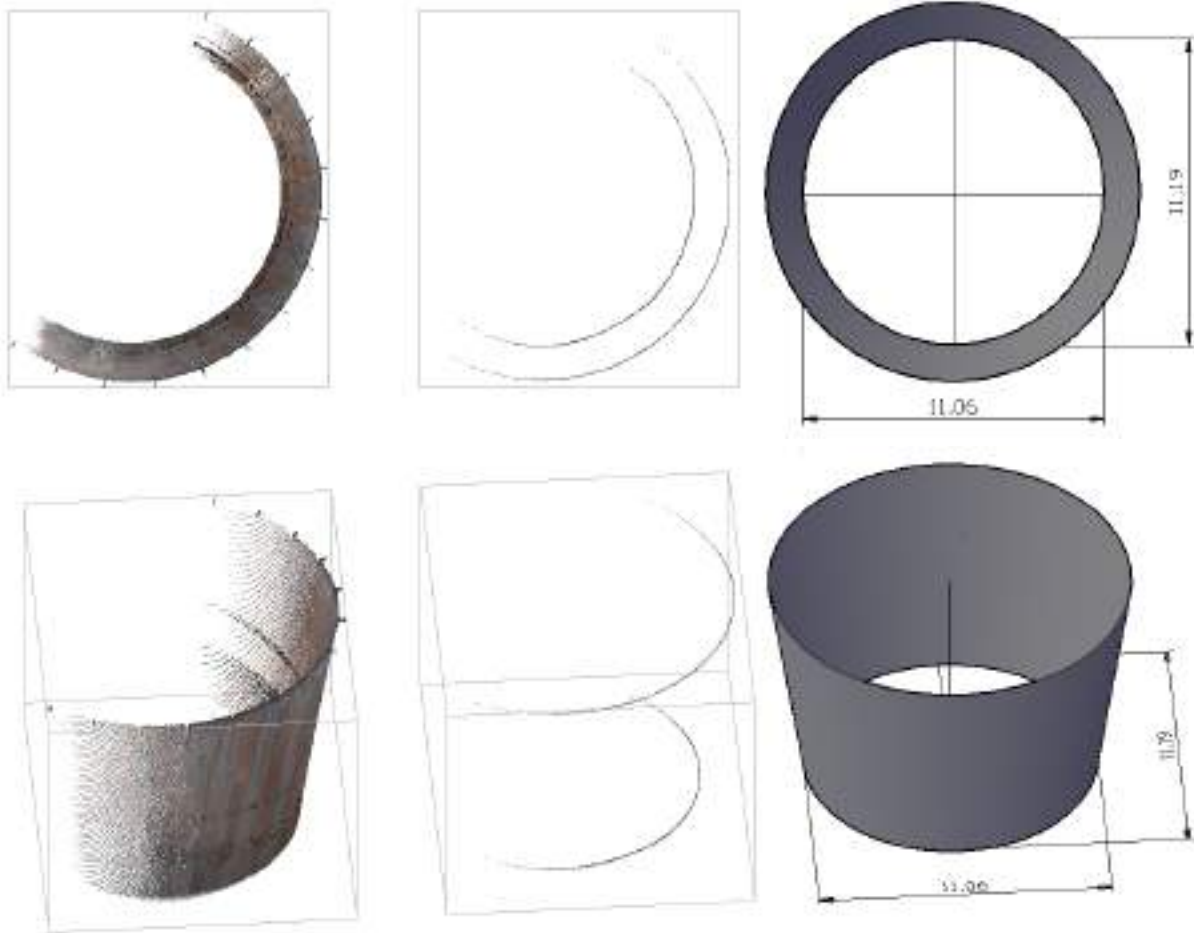
Truncated cones with an ellipse as their base shape can be modelled with this function. This kubit cone is especially suitable for modelling from point clouds. The cone's axis can be locked, so that when stretching or shortening lengthwise the position of the surface skin on the point cloud stays as it is (AutoCAD properties window). Furthermore kubit cone can be used as a reference surface for a deformation analysis and to create a deformation true solid (→**Modeling - Analyze frustum of a cone in a point cloud**).

The creation of the cone is done in three steps:

1. As the first step, the ellipse is defined in 3D space, that will determine the profile and the position of the axis of the cone. The ellipse requires at least five construction points on the circumference. From the selected points the average plane is determined, to which the cone's axis will be perpendicular, and then the ellipse is computed. Should more than five points be selected the average ellipse to fit the points will be determined. The number of points that have already been selected is displayed during input.
2. The height from the profile either upwards or downwards as well as the slope angle of the surface skin is determined from the subsequently entered points. It is sufficient to use just one point on the surface skin. For better accuracy it is recommended to select multiple points. These points have no influence on the cone's axis.
3. At the end it is stated whether the cone's axis, when creating the cone, should be fixed (that means locked). "Fixed axis", as used here, means that the truncated cone can only be modified in its height. This property can also be separately edited in the properties dialog.

For an upright standing cone shaped tank one would proceed as follows:


1. The point cloud would be loaded that will be used to model the scanned tank (→Point cloud - Insert point cloud).
2. Next the visible point cloud region would be cut back to two thin slices. For this use the command →**Sections - Define slice** (UCS) (PCS) or →**Sections - Define multiple slices** (PCMSC)
3. Now the truncated cone is created. Here, to determine the elliptical profile, a few points in the lower thin point cloud slice are selected. To define the height, the points in the upper thin slice are used. The cone's axis should be fixed (locked).
4. Lastly, the whole of the point cloud region is again made visible and the grips to stretch the already created kubit cone can be used to again move the upper and lower cap surfaces to fit it onto the whole of the cone shaped region of the point cloud. Thereby the position of the surface skin and the axis stay as they were, it's just the length that is adjusted.



Shown in the figure above, from left to right, in top view and as perspective view, are the following:

- the point cloud of the scanned truncated cone (tank or tower),
- the two thin point cloud slices and
- The slightly elliptical truncated cone that was created with labelled axes dimensions.

Modeling – Truncated cones – Create a truncated cone

	Command name:	FCCREATE
	Ribbon Tab:	As-Built Modeler

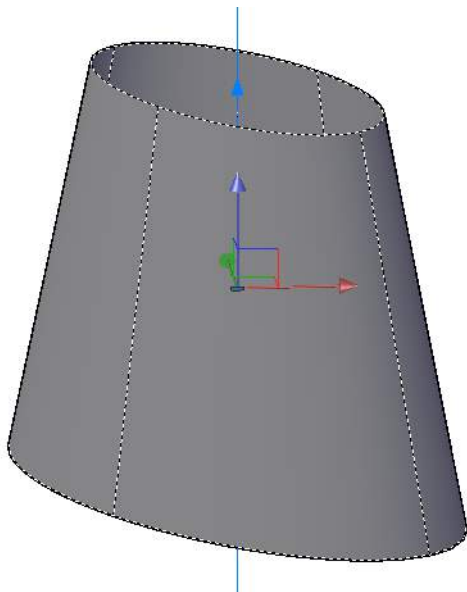
Truncated cones with an ellipse as their base shape can be freely created with this function.

Note: This command is not very well suited for modelling truncated cones in point clouds. To do that it is better to use the commands → **Modeling – Truncated cones – Fit circular truncated cone** and → **Modeling – Truncated cones – Fit elliptical truncated cone**.


The cone's axis can be locked, so that when stretching or shortening lengthwise the position of the surface skin on the point cloud stays as it is (AutoCAD properties window).

The creation of the cone is done in five steps:

1. Firstly the ellipse is created parallel to the UCS X-Y plane, which will represent the profile of the truncated cone. First pick the centre point of this base ellipse.
2. Now pick the end point of the first semi-axis.
3. Then the length of the second semi-axis can be entered.
4. Now the height of the truncated cone is entered and
5. the input of the taper angle to the first semi-axis.
6. At the end it is stated whether the cone's axis, when creating the cone, should be fixed (that means locked). "Fixed axis", as used here, means that the truncated cone can only be modified in its height. This property can also be separately edited in the properties dialog.



Modeling – Truncated cones – Explode to AutoCAD 3D solid

	Command name:	FCTOSOLID
	Ribbon Tab:	As-Built Modeler

With this command you can convert kubit cones into AutoCAD solids (3D solids).

Command: FCTOSOLID

Select cone:

Select objects: *(click on the cone(s) to be converted)*

Delete or retain cone? [Retain/Delete] <Retain>:D

*(R or ENTER key, to retain the original cone,
D, to delete the original cone)*


1 cone successfully converted.

1 cone deleted.

kubit cones serve as modelling and construction aids. There are application specific objects, which you can only work with in connection with our software. If you wish to hand over the project to a third party, you must first bring it into a format that pure AutoCAD can recognise. As the last step in the processing of the surveyed cone is the job of converting it into an AutoCAD solid (3D solid). They can then no longer accept any changes to their geometry, therefore this step should only be undertaken when the geometry is correct.


You recognise the conversion in that the cone appears to be no longer "hollow", but is displayed as a solid object.

Modeling – Fit 3D solid

	Command name:	PCFITSOLID
	Ribbon Tab:	As-Built Modeler


This command fits any 3D solid into a point cloud.

Help – Manual

	Command Name:	PCHELP
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

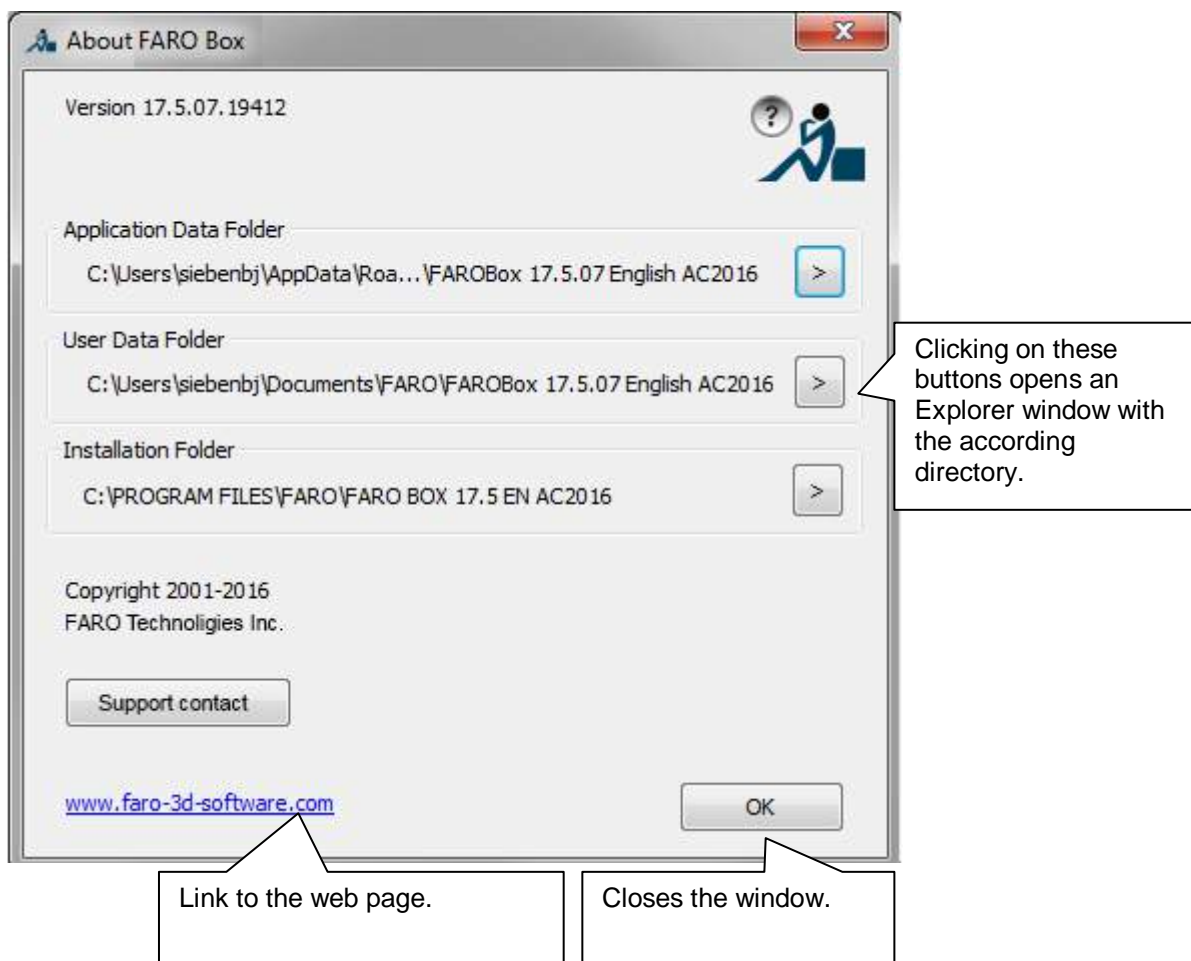
This command opens the manual as PDF file.

As-Built – About


	Command Name:	KUBTABOUT
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

This provides information about the current version and about important paths. Furthermore it enables the fast access to data within the most important software directories, like the application data directory, the installation directory and the directory which contains the configuration file.

The following window will open:

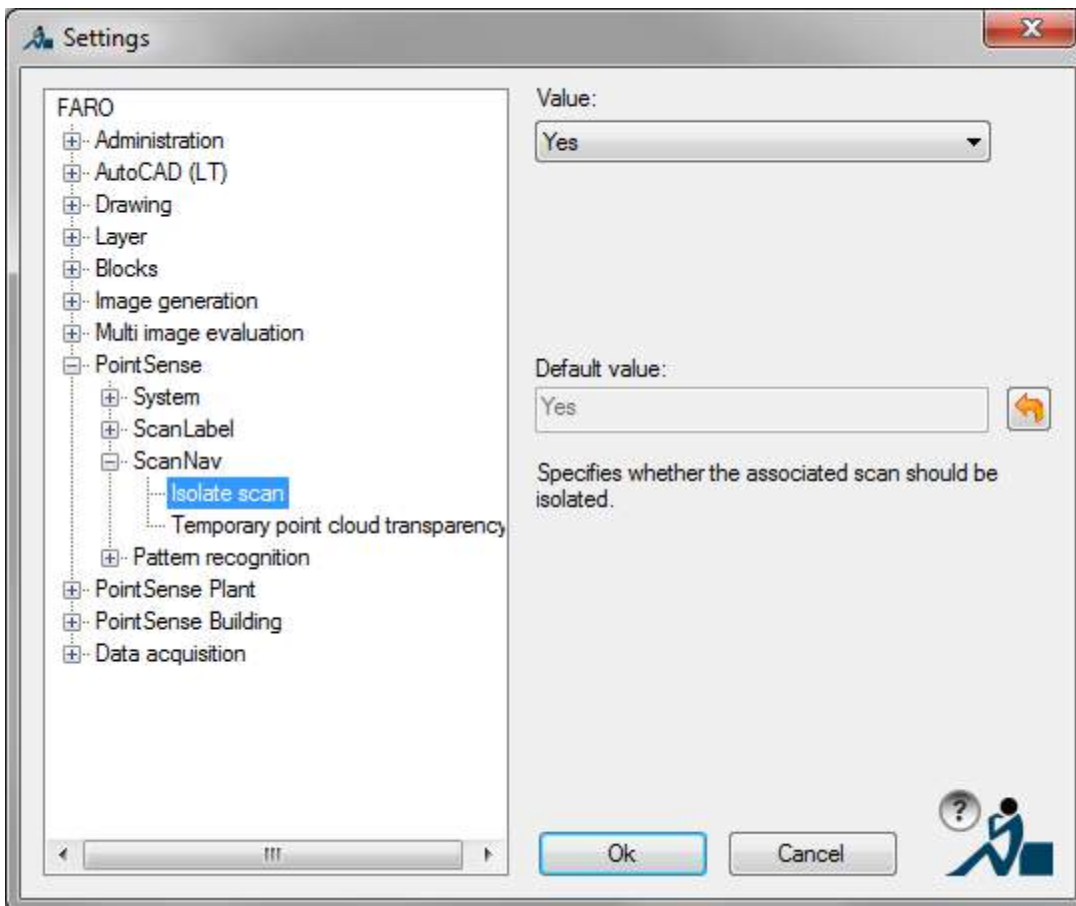



As-Built – Settings

	Command Name:	KCSETTINGS
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

Here you can display and edit important configuration data of As-Built. As a result, you can adjust As-Built to your individual needs.

Clicking the command will open following window:



On the left side you see all key entries. 'Value' on the right side gives you the actual value of a selected key entry, which you may also change. 'Default value' states the standard setting of As-Built. If you want to reset to the standard setting of a value click .

The following table provides an overview over the possible adjustments, their meaning and possible settings.

Entry	Possible values	Description
Administration		
Configuration file for company guidelines	Path and file name	Path and name of the configuration file with the company guidelines. Also see chapter “Adaption and Configuration of As-Built”
Check for updates automatically	Yes/no	
Show Getting Started dialog	Yes/No	
Drawing		
Drawing unit	Millimetres ... Parsec	Here you set the length unit for your AutoCAD drawing. If one drawing unit corresponds exactly one meter you have to select „meter“ here. This setting which also can be done in other places in As-Built also influences the size of As-Built blocks and is used for the image orientation in As-Built.
Block scaling factor	Any number > 0	Multiplication factor for block sizes and 3D dimensions. With this value you can adapt the size of the As-Built blocks. The factor refers to the size in the template file and the drawing unit currently set.
Layer		
Object(s) for which the layer is stated	Any name	Layer name for an image, a block or geometry.
Blocks (List of As-Built blocks)		
Block name	Any string	Name of the block that is to be used for certain As-Built elements. Corresponds to the name of the file in the directory DWGBASE that is to serve as template.
Attribute	Any string	Name of the attribute in a block template which As-Built uses for certain commands.
Image generation		
Automatic generation of image names		
Prefix	Any string	Prefix for the automatic generation of image names. [Name of the new image] = [Prefix][Name of the original image][Suffix]
Suffix	Any string	Suffix for the automatic generation of image names. [Name of the new image] = [Prefix][Name of the original image][Suffix]
Image type	Tif/ bmp / jpg / png	Image format (file extension) for the automatic generation of image names. Typical are "tif", "bmp", "jpg" and "png". If you leave this field empty the file format and extension of the original are applied.
Image path	Folder	Default path (folder) for the automatic storage and naming of images.

		<p>You could for instance write "D:\Project\Images".</p> <p>If you leave this field empty the folder of the original image is assumed as destination for the images.</p> <p>Specifications relative to the location of the original image are allowed as well. Following syntax is applied:</p> <ul style="list-style-type: none"> • „\Images“ – directory “Images“ directly under the root of the current drive, that means the drive of work directory of AutoCAD. Generally this value will be dissolved to “C:\Images“. • “.\Images“ – The point in front of the back slash means the directory of the original image. If the original image is stored under “D:\Project“ the resulting image will be stored under “D:\Project\Images“. • “..\Images“ – “..“ means one level above the directory of the original image directory. If the original image is stored under “D:\Project“ the resulting image will be stored under “D:\Images“.
Path type	Absolute, Relative, No path	<p>Path type of the image file that shall be used when saving the image in the drawing: "Absolute": the whole path of the image is stored. "Relative": the image path is stored in relation to the drawing file. "No path": no path is stored (makes only sense if image and drawing are in the same folder).</p> <p>“Path type“ is not related with the “Image path“.</p>
Use TIFF compatibility mode	yes / no	TIFF saving option for compatibility with AutoCAD (LT) 2000 to 2006
Directory of camera parameter files	Any string	If you leave this field empty (default) the folder of the original image is used. Otherwise the defined folder is searched for the camera parameter files (*.vp2).
Use automatic zoom to image point	yes / no	Image rectification / image orientation: Do you want to enable automatic zoom to the image point after the control point has been selected, as soon as an estimation for the position can be calculated? Default: yes
Multi image evaluation (3D Drawing)		
Maximum distance of the image rays (in drawing units)	0.1	Maximum distance of the image rays (in drawing units) for the 3D drawing with multiple oriented images. If the distance is larger than this value a warning message is displayed. This Threshold refers to a distance of 10 metres from the camera position and is being scaled accordingly.
Minimum angle of the	10	Minimum angle of the image rays (unit Degree) for

image rays (unit degree)		the 3D drawing with multiple oriented images. If the angle is smaller than this value a warning message is displayed.
Point Clouds		
System		
Set object snap for point clouds	yes / no	Object snap settings will automatically be adjusted to be suitable for point clouds.
Set system variable PLINETYPE	yes / no	Specifies whether the system variable PLINETYPE shall be modified at AutoCAD startup to create only polylines that are compatible with As-Built.
Default value for making point clouds transparent	Number 0 to 90	While drawing and modeling you can set the point clouds transparent using the according option in the Section manager. The default value defines the transparency level that is set in the Section manager when opening a new drawing.
ScanLabel		
Size (in metres)	Any number > 0	Size of a Scan Label in (meters) (>0)
Colour	number >=1 and <=255	Colour of the layer the Scan Labels are stored on (AutoCAD index colour)
Text style	string	Name of the text style for the Scan Label annotation
Layer name	string	Name of the layer the Scan Labels are stored on
ScanNav		
Isolate scan	yes / no	Specifies whether the selected scan shall be isolated for scan navigation
Temporary point cloud transparency	Number 0 to 90	Defines the temporary transparency level when selecting a ScanLabel
Pattern recognition		
Desired block size in metres	Any number > 0	This value defines the size of cubic blocks for point queries. It influences mainly the performance of the pattern recognition. One block should have a size to hold 1 to 10 million points. The default setting for this value is 3.5 metres and is optimized for working with buildings and plants. Only change this value if you want to work with a different structures size, e.g. terrain models or structures in mm-sizes. The unit of this value is metres!
Plant		
Angle tolerance	number	Angle tolerance for fitting beams
Text height for export commands	number	Text height (in drawing units) for exporting pipe lines. Command PCEX (Export to Polylines) or export of beams, command PCEXSTEEL
Building		
Construction		
Wandbruch	yes / no	Break wall lines with window and door commands?

Stairs		
Dimension unit	Millimeters ... Parsec	Unit for dimension annotation
Arrow size	number	Scaling factor for stair case arrow blocks
Feature Data		
Viewer mode	yes / no	Do you want to run the feature data management as a viewer? In viewer mode no data can be added or modified. It can only be viewed. No license needed.
Decimal marker for numbers	comma or point	Decimal marker for numbers when exporting data to text files
Column separator	Any string	Column separator when exporting data to text files (';' or ',' or 'TAB', ...)
Comment sign	Any string	Comment sign when exporting data to text files ('#' or '\$' or '*', ...)
Table Export Type	One file per table/ one file with one universal table / One file with all tables	Data export to tables chose the option that suits your needs
Export XML scheme	yes / no	Export XML scheme?
Code page	ASCII / Unicode UTF8 / Unicode UTF16 (Little Endian) / Unicode UTF16 (Big Endian)	Code page for data export tot ext files ('ASCII' or 'Unicode UTF8' or 'Unicode UTF16 (Little Endian)' or 'Unicode UTF16 (Big Endian)')
Label size	Any number	Height of a label for area objects in the drawing (in AutoCAD units)
Pattern recognition		
Tolerance Polygon recognition	Any number	Tolerance for the recognition of gaps when creating As-Built area objects (in AutoCAD units) (distance in which are assumed to be the same)
Tolerance Pattern recognition	Any number	Tolerance for the recognition of gaps when recognizing door symbols (in AutoCAD units) (distance in which are assumed to be the same)
Total Station		
System		
Observation point – Precision of coordinates	integer	Number of decimals to be used for calculation of coordinates from measured values
Commandline send mode	"1" or "2"	"1" or "2", mode "2" can be used experimental, if coordinates are sent fragmentary to command line.
Show 3D label	Yes / no	Should 3D labels been shown?


Use As-Built drawing template path	Yes / No	Set the path to locate drawing templates to the As-Built template folder. This will be the default path in the "New drawing" dialog box.
Points and Protocol		
Instrument station protocol	Yes / no	Turns the protocol function on or off Yes: A protocol file will be created in which the coordinates of the stationing data, the control points, the observation points and forward steps are recorded. The file has the name of the drawing with the ending .TPR and lies in the directory of the current drawing. No: No protocol file will be created
Control points protocol	Yes / no	
Observation points protocol	Yes / no	
Points by intersection protocol	Yes / no	
Stake out protocol	Yes / no	
Observation point - Type	Block / AutoCADpoint	Display type of observation points in the drawing: AutoCAD point object or Observation point block
Observation point - Number	Unique per drawing / Elective Zero at new station	Numbering definition of observation points: - unique per drawing or - starting with Zero when new station
Orientation		
Scale tolerance	any number	Tolerance limit for the scale used for Free Orientation. If value exceeded traffic lights switch from green to yellow.
Deviation of coordinates	any number	Tolerance limit for coordinate deviation used for Free Orientation. If value exceeded traffic lights switch from green to yellow.
Standard deviation	any number	Tolerance limit for standard deviation used for Free Orientation. If value exceeded traffic lights switch from green to yellow.
Stake Out		
Maximum distance*	any number	Deviation from desired set value. If value exceeded a warning message will be shown.
Construction		
Break wall	Yes / no	Shall the wall lines be broken when inserting windows or doors?
Instrument		
Average refraction coefficient	any number default=0.13	All total station measurements are corrected regarding refraction/earth curvature using this coefficient. If the value is set to 1.0, no correction will be applied.
Approach measurement point automatically	Yes / no	If the measurement point is approximately known, it will be approached automatically by motor drive. E.g. Free orientation, over-determined control point measurement.
LeicaTPS1200GeoComTimeout	Milliseconds value	Time duration [msec]. Duration, how long As-Built waits for response after sending a GeoCOM command to the total station. Only valid for instrument profile "Leica TPS1200 total stations"
WMS Distance offset	any number	Distance from Disto front face to instrument origin

		which has to be added to the Disto measurement
GeodimeterDelay	any number	Communication delay time in milliseconds for Geodimeter instruments.
Leica delete measurement after receiving	Yes / no	At some Leica instruments the stored angle and distance measurements has to be deleted before starting a new angle observation. Otherwise the old vertical angle will be send.
LeicaDelay	any number	Communication delay time in milliseconds for Leica instruments (time between receiving of a measurement and sending of delete command). This setting has only consequences if value "Leica delete measurement after receiving" is set to "yes".
Leica prism search mode	"1" or "2"	"1" for Leica PowerSearch or "2" for Leica SearchTarget
TopconDelay	any number	Holding time in milliseconds between command for changing the measuring mode and command for releasing a measurement.
DolmazonVectographSD Offset	any number	DolmazonVectograph BC9000 - offset for slope distance
Dolmazon4DLasergraph samplingrate2D	any number	Number of generated points per second in 2D tracking mode
Alternative Bluetooth mode	Yes / no	Workaround for an error that sometimes occurs with Leica instruments: When working with Bluetooth the communication sometimes "friezes" in such cases only switching to the alternative Bluetooth mode solves the issue
Sounds		
Soundtype	Sound file / system beep /no sound	You can choose between "Soundfile" if you want to be notified by a sound file or "Systembeep" if you want to be notified by the computer's system beep. The first has to be selected if the computer does not have system beep (e.g. notebooks)
Error	File path	Path for WAV file for sound notification when error message
Success	File path	Path for WAV file for sound notification when sending of measured value of hand-held laser was successful
BelowZtolerance	File path	Path for WAV file for sound notification when measured below Z tolerance
AboveZ tolerance	File path	Path for WAV file for sound notification when measured above Z tolerance
Measurewithoutactivecomm and	File path	Path to the WAV file for a notification sound in case a measurement was initiated without an AutoCAD command being active.
GNSSCountdownstart	File path	Path for WAV file. This sound is played, when the countdown for the measurement starts.
GNSSAveragingstart	File path	Path for WAV file. This sound is played when collecting of coordinates for the averaging starts.
GNSSMeasurement	File path	Path for WAV file. This sound is played, when the

finished		coordinate measurement is finished.
----------	--	-------------------------------------

The values as given here in the table are saved in the FARO configuration file. That file is named **FaroConfigCurrentUser_en.cfg** and is located in the application folder under \FARO. Every time you start As-Built these values are read; every time you finish As-Built these values are brought up to date. Should this file be destroyed (e.g. breakdown during writing process), As-Built automatically uses the back-up copy, which is created under the name **FaroConfigCurrentUser_en.cfg.bak** after every use. Thus you can continue working. Changes made during the last use of As-Built are, however, lost. In case the configuration file is missing, As-Built automatically creates a new one that is equipped with As-Built default settings. A message will appear on the screen.


As-Built – Getting Started ...

	Command Name:	KUBT_GETTING_STARTED
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

This command opens a dialog where you can access tutorials, videos and further information about As-Built for AutoCAD.


The “Getting Started” dialogue box appears by default when starting AutoCAD. By selecting the option “Show this dialog at startup” you can switch off this behaviour for the currently installed program version.

As-Built – Knowledge Base...

	Command Name:	
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

This command opens the FARO Knowledge Base product page for As-Built for AutoCAD in an Internet browser. Here you will find useful information about the product and help for helping yourself.

As-Built – Software Update

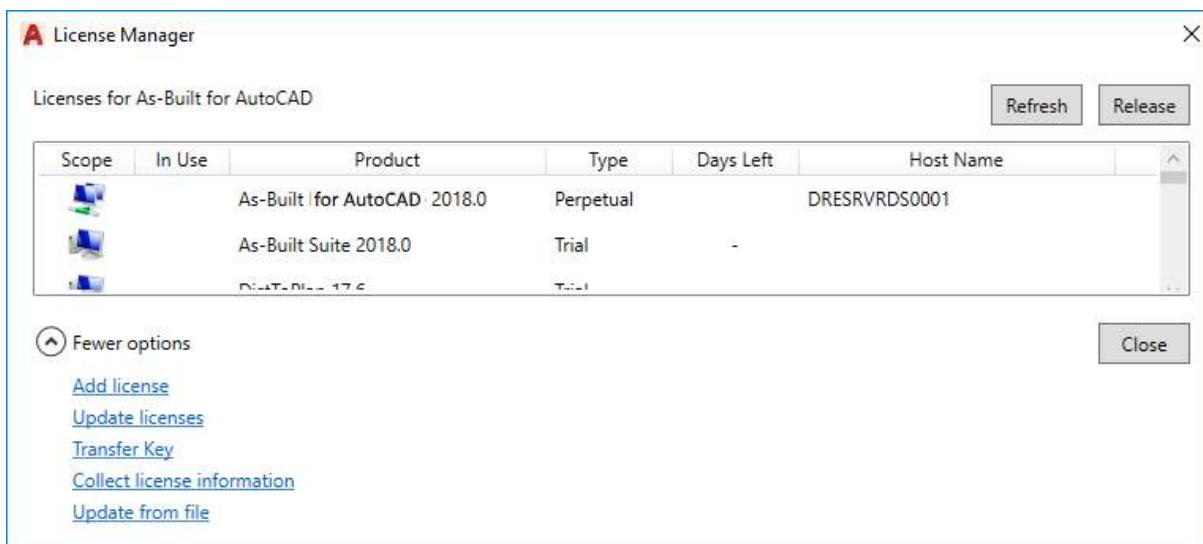
	Command Name:	_KUBT_VERSIONCHECK
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

This command checks whether there is a newer version of the software. If yes, a download is offered.

As-Built – Licence Management ...

	Command Name:	KUBT_SHOW_LICENCE_INFO
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

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 Knowledge Base:

- [for As-Built for AutoCAD](#) 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 – Customize User Interface

	Command Name:	KUBT_CONFIG_PRODUCTS
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

As-Built for AutoCAD is a product with extensive functionality for the evaluation of point clouds, photogrammetry, feature data acquisition as well as a total station interface.

This command allows you to show and hide menus and Ribbon tabs to make the user interface easier to navigate.

Customize As-Built user interface ✕

Which parts of the user interface do you want to activate? 

Current AutoCAD profile: <<Unnamed Profile>>

As-Built Modeler

As-Built Analysis

As-Built Plant

As-Built Building plans

As-Built Photo

As-Built Total Station-interface

As-Built Feature data management

As-Built FAMOS Extension (unsupported language)


As-Built ARCHIKART Extension (unsupported language)

Viewer mode for feature data management

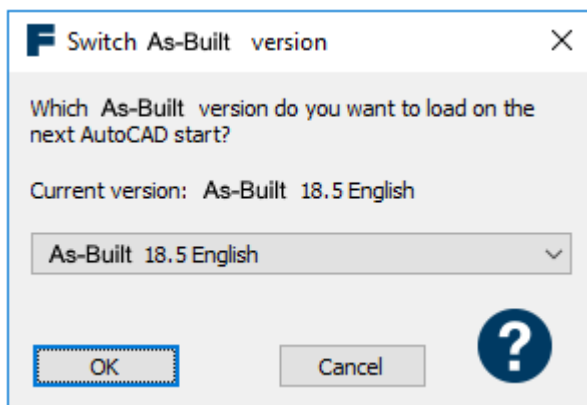
OK
Cancel

You also have the option to run the feature data management in viewer mode. In this mode you may look at feature data without needing a license. But you are not able to add or edit feature data. Please refer to the [Feature Data manual](#) for more information. This way As-Built for AutoCAD can be distributed free of charge together with the structured data.

As-Built – Version Manager

	Command Name:	KUBT_SELECT_KUBIT_BOX_VERSION
	Ribbon tab:	As-Built Modeler, Building Plans, Photo, TS Design

If you have more than one As-Built for AutoCAD version installed on your computer, you can select in the Version Manager with which version you want to work. After calling the command the following dialogue box is displayed:




You can select the appropriate version from the pick list or remove the currently loaded version (Select "**None (unload current version)**"). After confirming the selection with [OK], you will be prompted to restart AutoCAD so that the changes can take effect.

Should you ever have completely unloaded As-Built, then none of the As-Built commands will be able to be called from the AutoCAD menus and ribbon tabs. If this should be the case, you will only be able to start the Version Manager from the AutoCAD command line, to reload a version of the As-Built for AutoCAD. Enter KUBT_SELECT_KUBIT_BOX_VERSION.

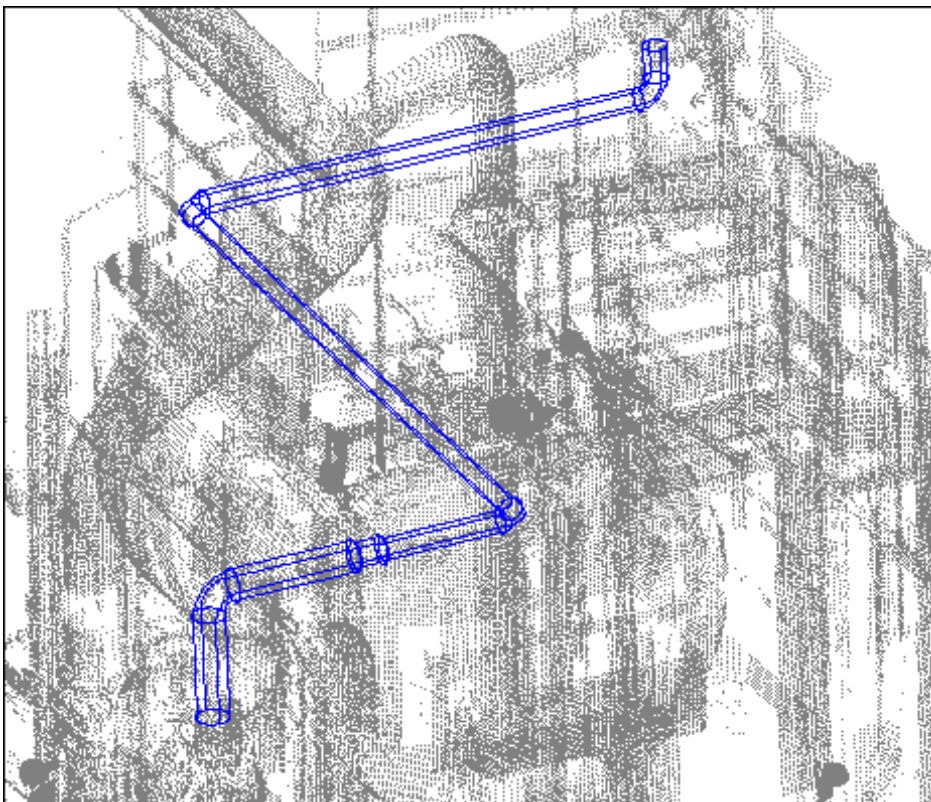
10.2 As-Built Analysis Tools

Analysis - Clash Detection

	Command Name:	PCCLASH
	Ribbon Tab:	As-Built Plant

With this command you can detect clashes between AutoCAD 3D solids and the point cloud. Clashes are clustered, the resulting clusters appear in the Section Manager (see → **Sections – Section Manager**).

The clash detection is always performed with the currently visible part of the point cloud!



Important note

For clash detection you will need a PTC point cloud or a so called SmartSection. If command is called with an Autodesk point cloud active, an error message will display, saying that there is no current point cloud.

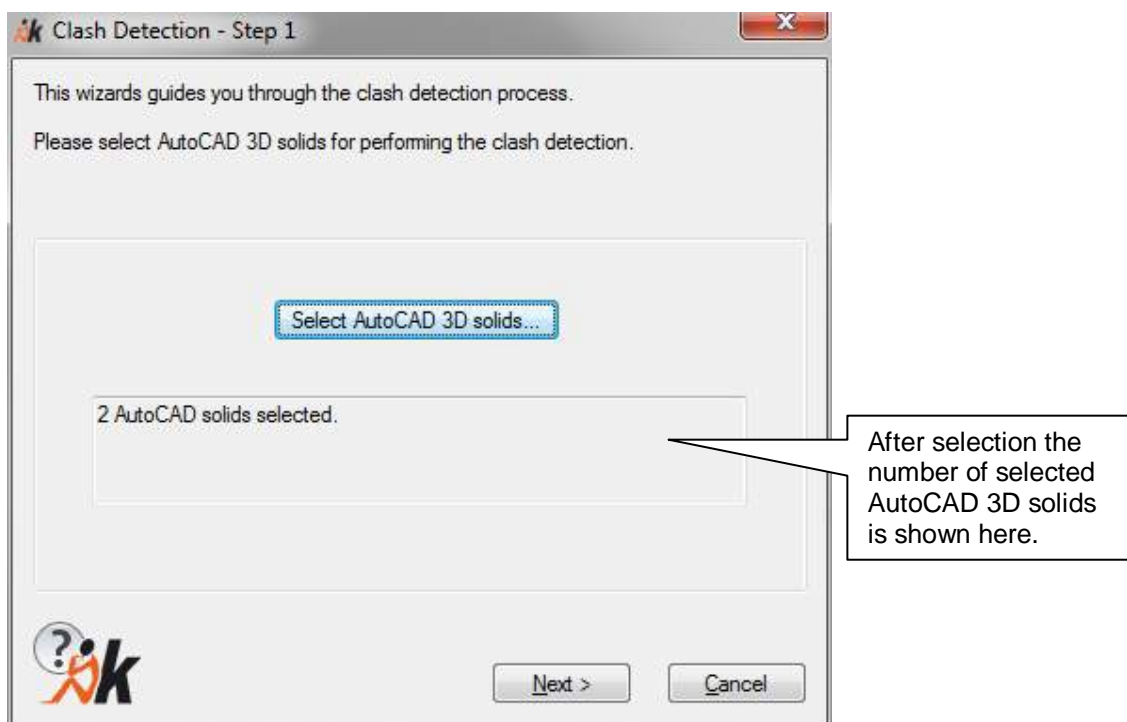
If you are working with Autodesk point clouds you first have to define a section of the cloud, which you want to run the clash detection with and convert this section temporarily into PTC format using the → „**SmartSection**“ command from the section manager.

A wizard guides you through the process of clash detection.

Step 1:

You need to select one or more AutoCAD solids, with which you want to perform the clash detection.

After you have clicked the “Next >” button, the clash detection will be performed. This may take a while, depending on the number and complexity of the solids and the number of currently visible points.

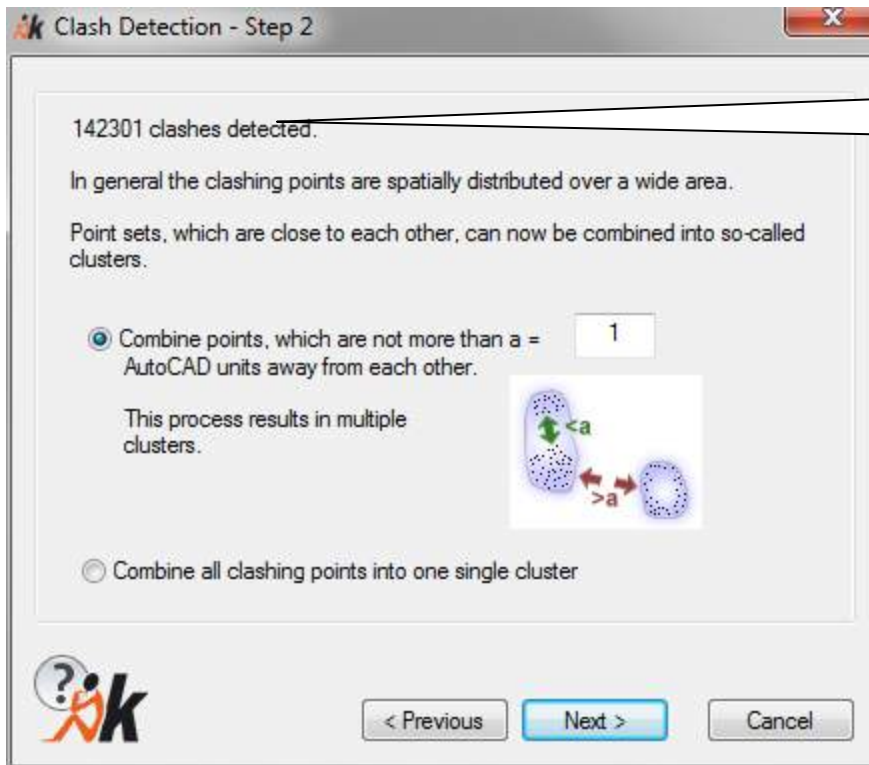


Step 2:

You can specify if the clashing points should be clustered or not. Clustering means, that points which are close to each other are combined in one point set (cluster), while points which are far away are not included in the cluster.

You need to specify a distance (in AutoCAD units), which defines the minimum distance between different clusters. In other words: If two points have a distance greater than the distance specified, they will belong to different clusters.

Each cluster results in a *temporary* section in Section Manager).

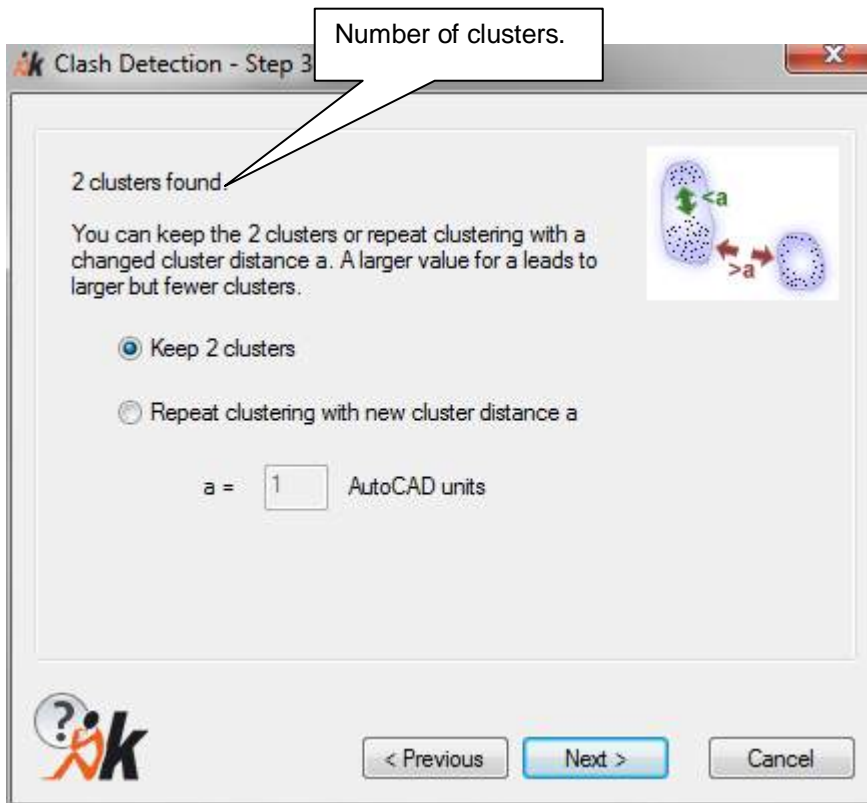


How many points clash with the selected solids.

Step 3:

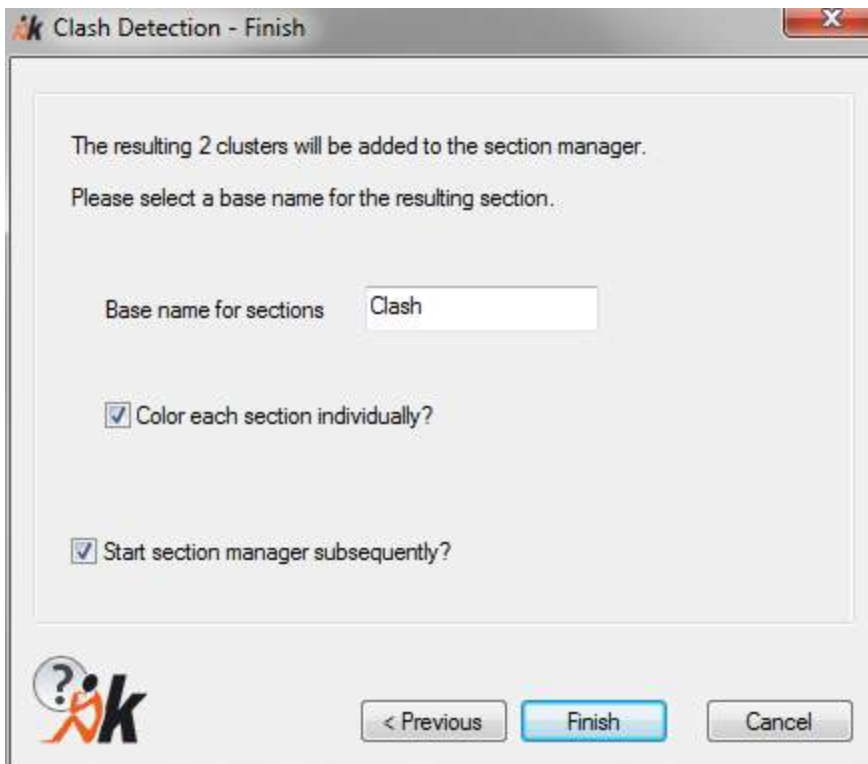
The resulting number of clusters is shown. If you like to have more or less clusters, you can change the cluster distance and repeat the operation. Increasing the distance tends to result in less clusters and vice versa.

Please consider that there is no “correct” number of clusters. It depends on your purpose, whether you need many clusters with few points or few clusters with many points in each cluster. Feel free to play around with clustering.



Step 4:

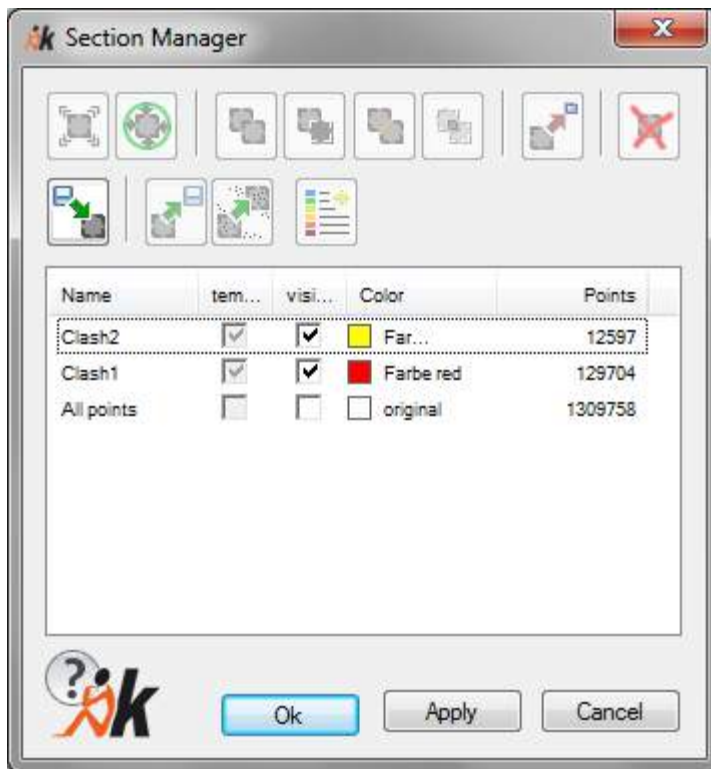
In the last step you need to specify a base name for the resulting clusters. The cluster's names are completed by consecutive numbers.




If you have checked the „Color each section“ box, each section will get its own color, using an AutoCAD index color, starting with 1 (red).

If you have checked the „Start section manager“ box, the Section Manager (see **“Sections – Section Manager”**) with the newly created clusters will appear after pressing „Finish“. Now you can show or hide the clusters, color them individually, rename or export them.

Special attention should be paid to the fact that the resulting cluster sections are *temporary*. That means, that they are valid only as long as your current AutoCAD session is running! If you want to make the clusters (or any other temporary sections) permanent, you need to export them (page 97f →Export).



Analysis – Analysis of arbitrary surfaces/solids

	Command name:	PCDISTCOL
	Ribbon tab:	As-Built Analysis

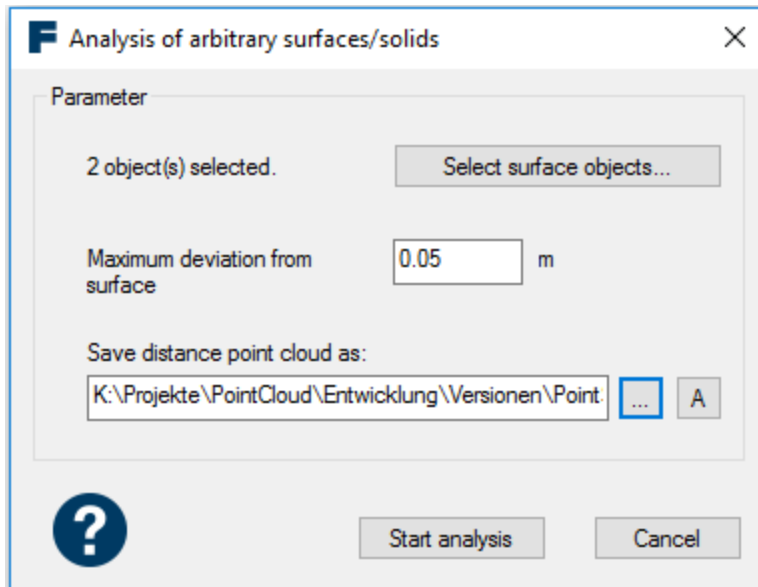
This command creates a new point cloud, in which all of the points can be coloured in relation to their distance from AutoCAD standard surface geometry.

The following AutoCAD object types are supported:

- 3D Solid
- Region
- Surface

As well as these objects kubit planes, kubit cylinders and kubit cones are also supported.

After starting the command, the following dialog opens:



For the distance analysis, select one or more surface objects. The maximum deviation from the surface defines the range within which the distances from points to the object are determined. Points outside this range will not be considered.

The determined distances are stored in the distance point cloud (in E57 format). This is designed exclusively as a data storage device and is not suitable for any other use.

You can select your own path and name as the storage path of the distance point cloud or use the [A] button to accept the automatic suggestion. The suggested path of the distance point cloud is formed from the path of the original point cloud and the addition "_deviation_[unit]. e57".

The button [Start analysis] starts the distance calculation and opens the dialog "Colorize an analyzed point cloud". With this you can configure a distance-coded colorization for the previously analyzed point cloud. A new AutoCAD point cloud is created, which is inserted into the drawing.

You can find a detailed description of the "Colorize an analyzed point cloud" dialog under → **Analysis – Colorize an analyzed point cloud**. The dialog can also be

called up later with the PCDISTCOLEEDIT command independently of the analysis command.

A suitable legend can be created with the command PCDISTCOLLEGEND (→**Analysis – Create Legend**).

Notes on creation of appropriate objects

Convert AutoCAD mesh

Mesh → **CONVTOSOLID** → 3D solid

Mesh → **CONVTOSURFACE** → Surface

Convert AutoCAD Polyface Mesh

Polyface Mesh → **EXPLODE** → 3D surfaces → **CONVTOSURFACE** → Surface (planar)

AutoCAD Block reference (providing that it contains surface geometry):

Block reference → **EXPLODE** → further conversion dependant on the objects present in the block reference

Notes on practical application

Dependant on the number of points and the number and complexity of the selected objects, the creation of the deviation point cloud can take anything between a few minutes to several hours. The following notes help with the optimisation of the computational overhead and thereby the computation time.

1. Point cloud clipping: Using the region commands isolate the parts of the point cloud that are relevant for the computation, in this example the beam and part of the wall.

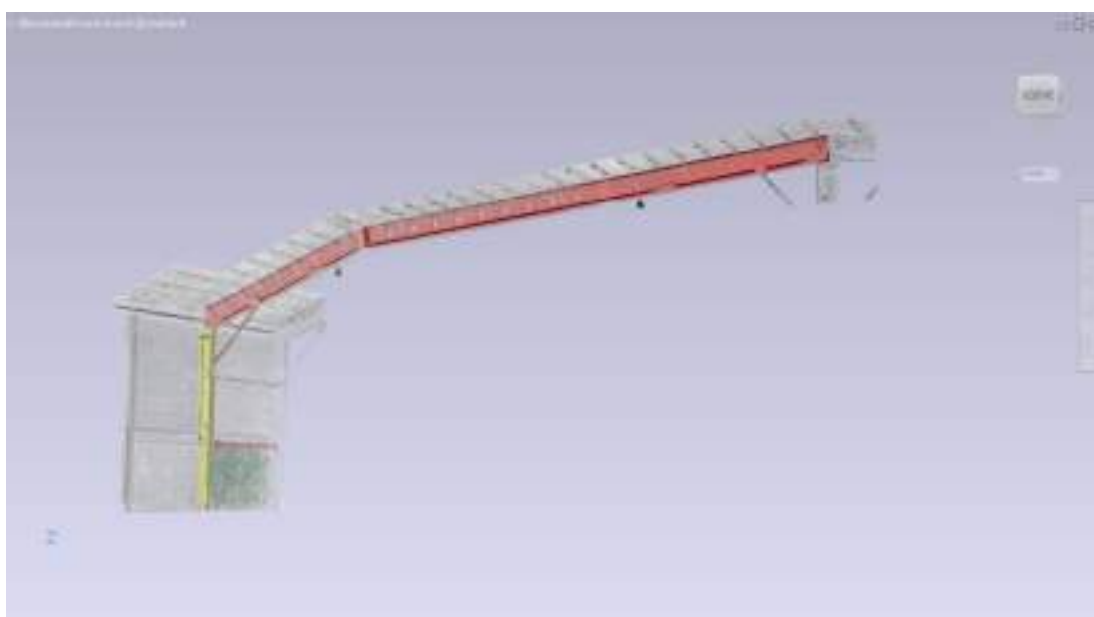


Figure 2: Clipped point cloud

2. Reduce the point density: Set the parameter “Point cloud density for fitting” (command: **PCSETPARAMS**) to a low value. The default value is 100. For ReCap data, that have been created from terrestrial laser scans, a reduction of the point density to a value less than 10 should be used. This accelerates the computation, but it also leads to a less dense point cloud results:

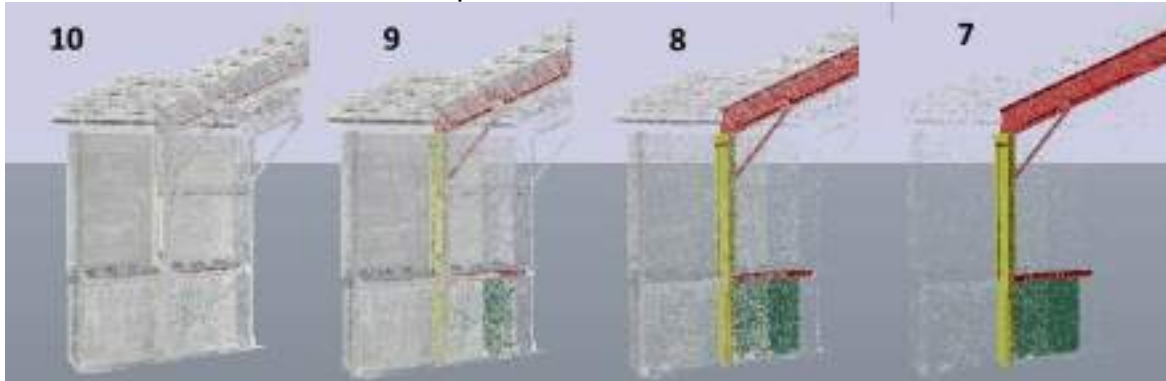



Figure 3: Appearance of the point cloud density

Tip:

If you want to more accurately analyse geometric primitives such as planes, cylinders or truncated cones, then use the dedicated analysis commands

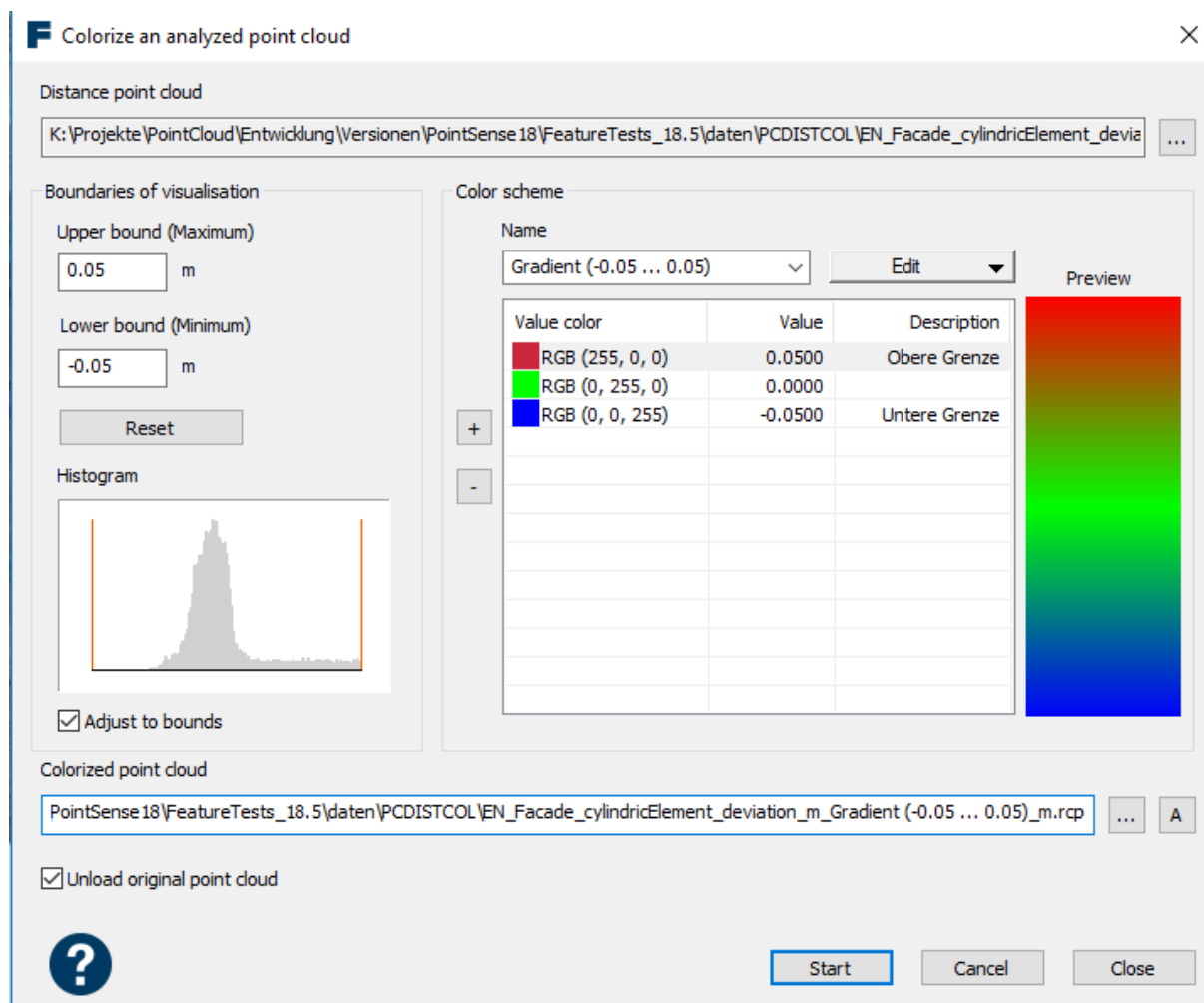
(**PCPLANEANALYSIS** , **PCCYLINDERANALYSIS** , **PCFRUSTOCONICALANALYSIS** ).

Analysis – Colorize an analyzed point cloud

	Command name:	PCDISTCOLEDIT
	Ribbon tab:	As-Built Analysis

This command creates a new AutoCAD point cloud (RCP/RCS) from an analysis or distance point cloud (E57) (→ Analysis – Analysis of arbitrary surfaces/solids). A user-defined color scheme is used for the colorization.

The following dialog opens when the command is started:

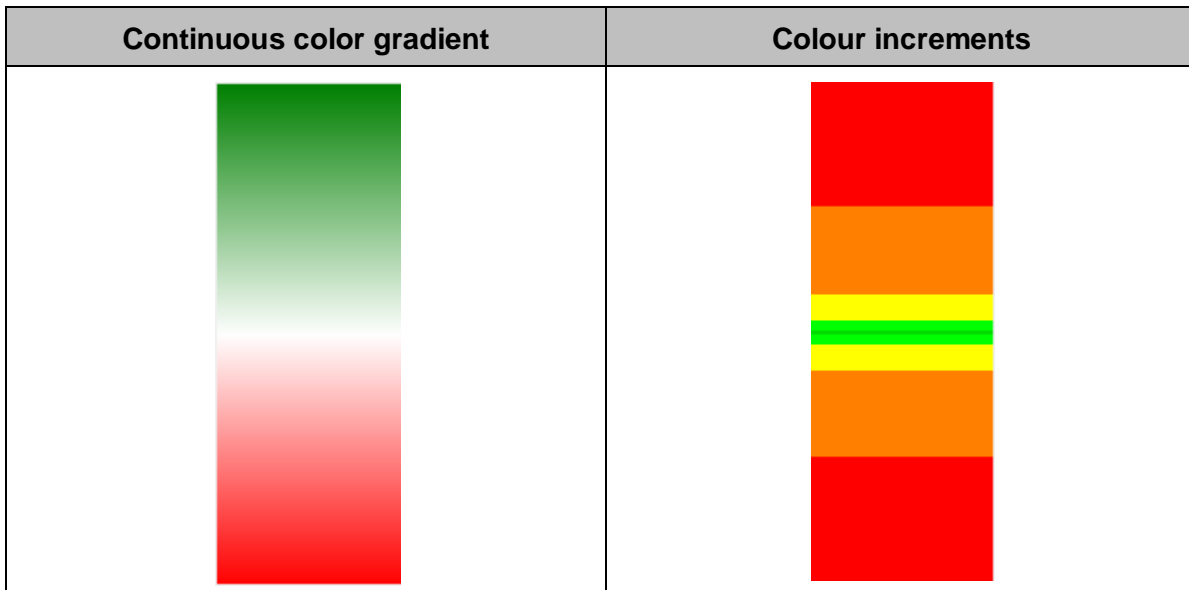


The distance point cloud from the previous analysis is already preselected. Otherwise, please select the desired file yourself.

The boundaries of the visualization determine the range within which points are included in the new point cloud. Points outside the boundaries are cropped off. The [Reset] button allows you to restore the boundary values with which the distance analysis was originally performed.

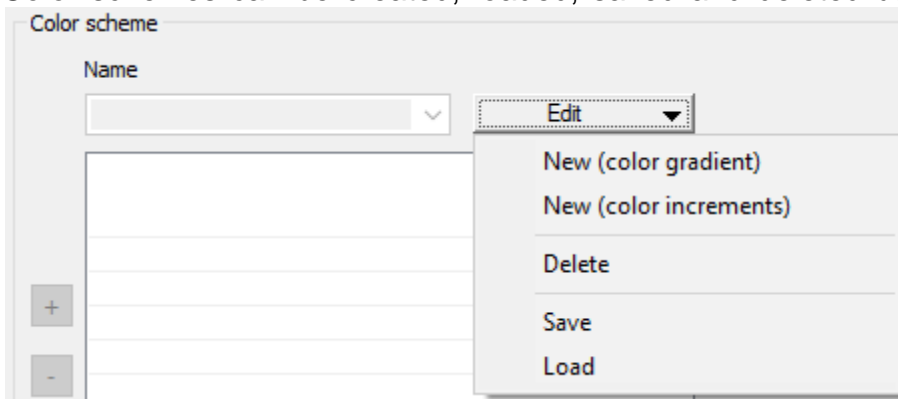
For easier estimation of these values, the histogram (occurrence distribution) of the distance values is displayed.

The colour scheme for the colorization of the point cloud can be flexibly configured. There are basically two different types:



In continuous color gradient, colors are set for discrete distance values, between which color gradients are calculated. The graded color scheme, on the other hand, colors distance areas with only one color and sharp borders to the adjacent fields.

Color schemes can be created, loaded, saved and deleted using the "Edit" button.



The [+] and [-] buttons can be used to add or remove rows for values (ranges) and colors. The individual colours, values or descriptions can be edited directly in the lines by double-clicking.

Finally, specify the path for the result - the colored AutoCAD point cloud. The [A] button can be used to generate a default path and name. The suggestion is the path of the distance cloud - supplemented by the name of the color scheme used.


Optionally, point clouds present in the drawing can be unloaded if desired.

After all settings have been made, start the colorization with the [Start] button. The newly created point cloud is inserted into the drawing and displayed.

[Cancel] aborts the command. [Close] quits the dialog and saves the settings made for a new execution of the command.

A matching legend can be created with the command **PCDISTCOLLEGEND** (→Analysis – Create Legend).

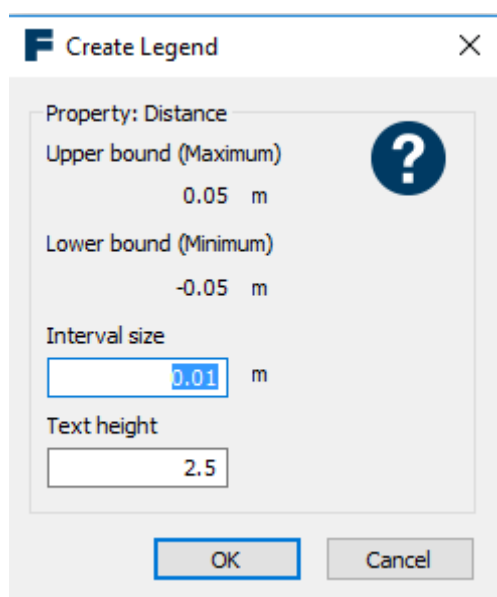
Analysis – Create Legend

	Command name:	PCDISTCOLLEGEND
	Ribbon tab:	As-Built Analysis

This command creates a legend for a point cloud created and colored using As-Built analysis commands. It represents the color scheme defined by → **Analysis – Analysis of arbitrary surfaces/solids** (PCDISTCOLEEDIT).

If you have not performed an analysis and simply want to create a legend for your point cloud colored with AutoCAD tools according to elevation or intensity, you can also use this command.

The dialog box shows you for which property (distance, elevation or intensity) a legend is created.




The interval size and text height are configurable. The interval size only influences the label subdivision, while the text height determines the size of the entire legend. The text height must be specified in drawing units.

After confirming the settings with [OK] the insertion point (bottom left corner of the legend) is requested. The legend is inserted in the XY plane of the current UCS. All drawing objects in the legend are grouped together. Various display parameters such as overall size, text height or line color can be modified using AutoCAD tools.

The format for the displayed values (in the dialog as well as in the legend) can be controlled by the AutoCAD system variable DIMZIN.

Tip: The height of the inserted legend can be estimated using the following formula:
 Height = 3 x text height x number of intervals.

Analysis – Plane – Analysis of planes in a point cloud

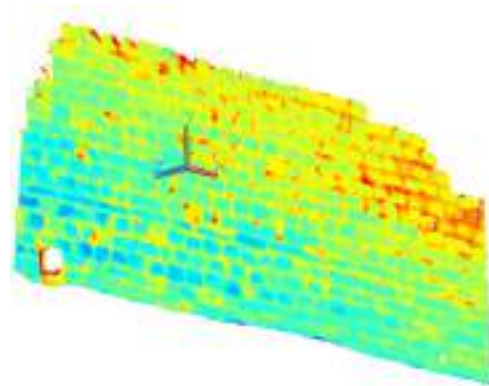
	Command Name:	PCPLANEANALYSIS
	Ribbon Tab:	As-Built Analysis

With this function parts of a point cloud can be analysed and modelled with respect to a reference plane (e.g. walls, floors, excavations or terrains). **Deformations** can be determined and displayed. It also allows you to create **AutoCAD solids** from those objects that deviate above or below the optimum plane and display them. The AutoCAD command `_MASSPROP` can be used to **determine the volumes** of the solids.

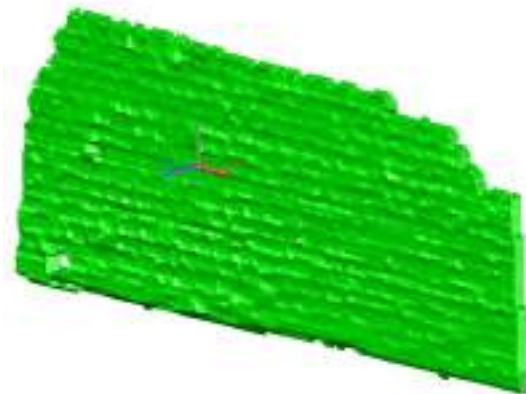
This command has many options and they can impact on each other. We therefore recommend that you read through the whole of this section before you start working with this powerful feature.



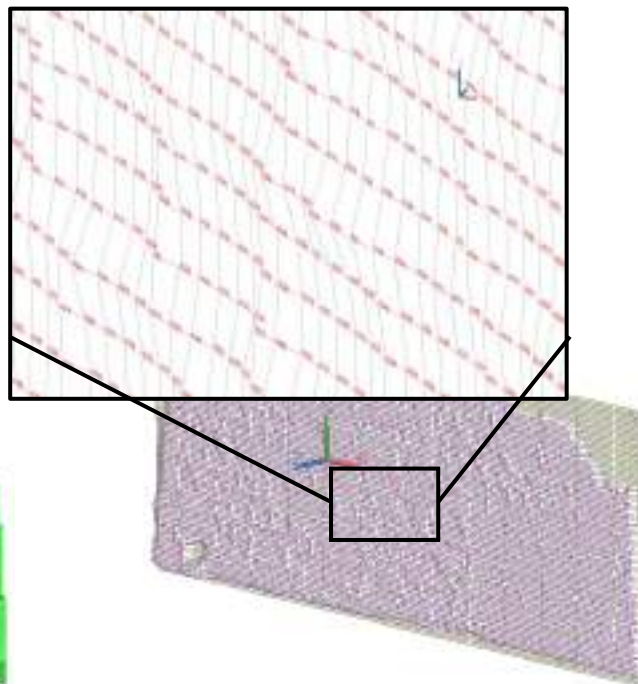
Original point cloud



Coloured in accordance with the variance from the reference plane



Modelled AutoCAD solid

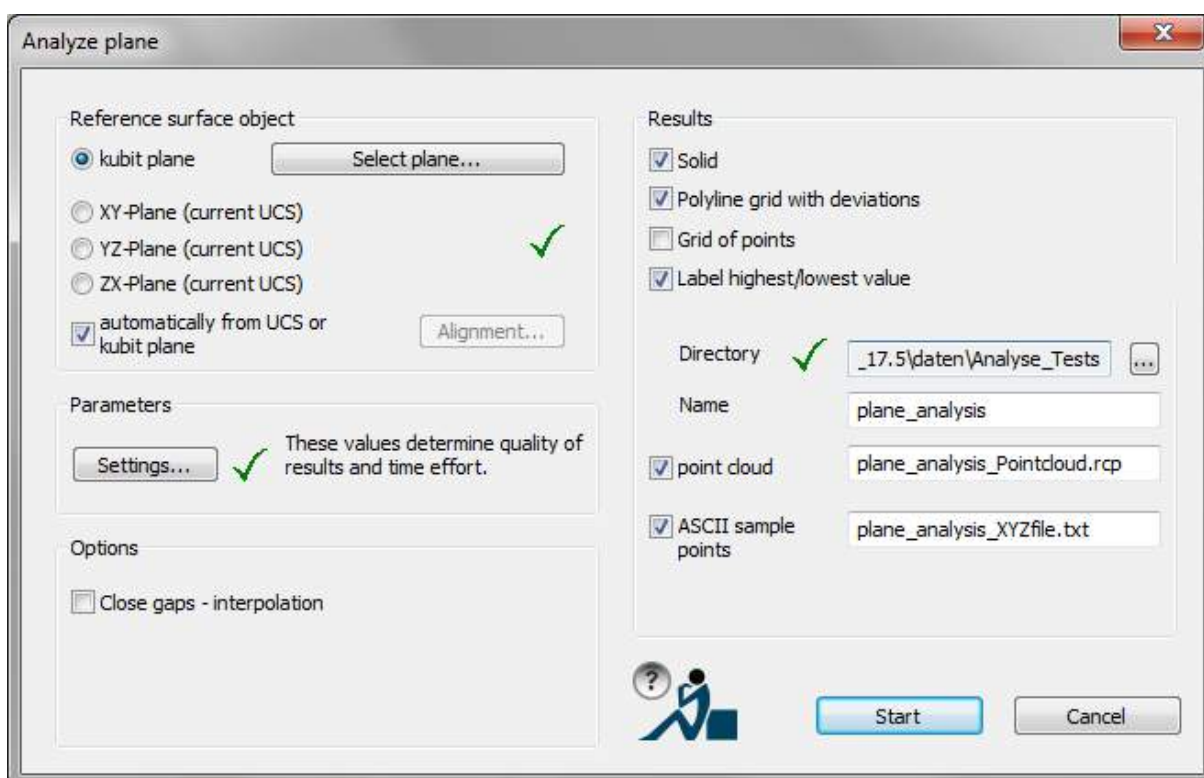


Variance displayed as 3D polyline grid

Next open a drawing with an AutoCAD point cloud. Now you can analyse the point cloud using the three UCS planes (XY, YZ or ZX) as the reference plane or to a not yet aligned kubit plane. To fit planes to the point cloud use the command **→Modeling/Construction - Plane - Fit plane** (KPFIT).

Hint: The property "limited" or "unlimited" of the kubit plane is taken into consideration during the analysis. Further information on kubit planes and their properties can be found in section **4.2 Planes**.

The command PCPLANEANALYSIS opens a dialogue box where you can enter the basic settings to be used in the flatness analysis. Buttons with a question mark behind them, must first be clicked so that the button [Start analysis] can be enabled and the analysis can now start. Here you can see a fully completed dialogue box:

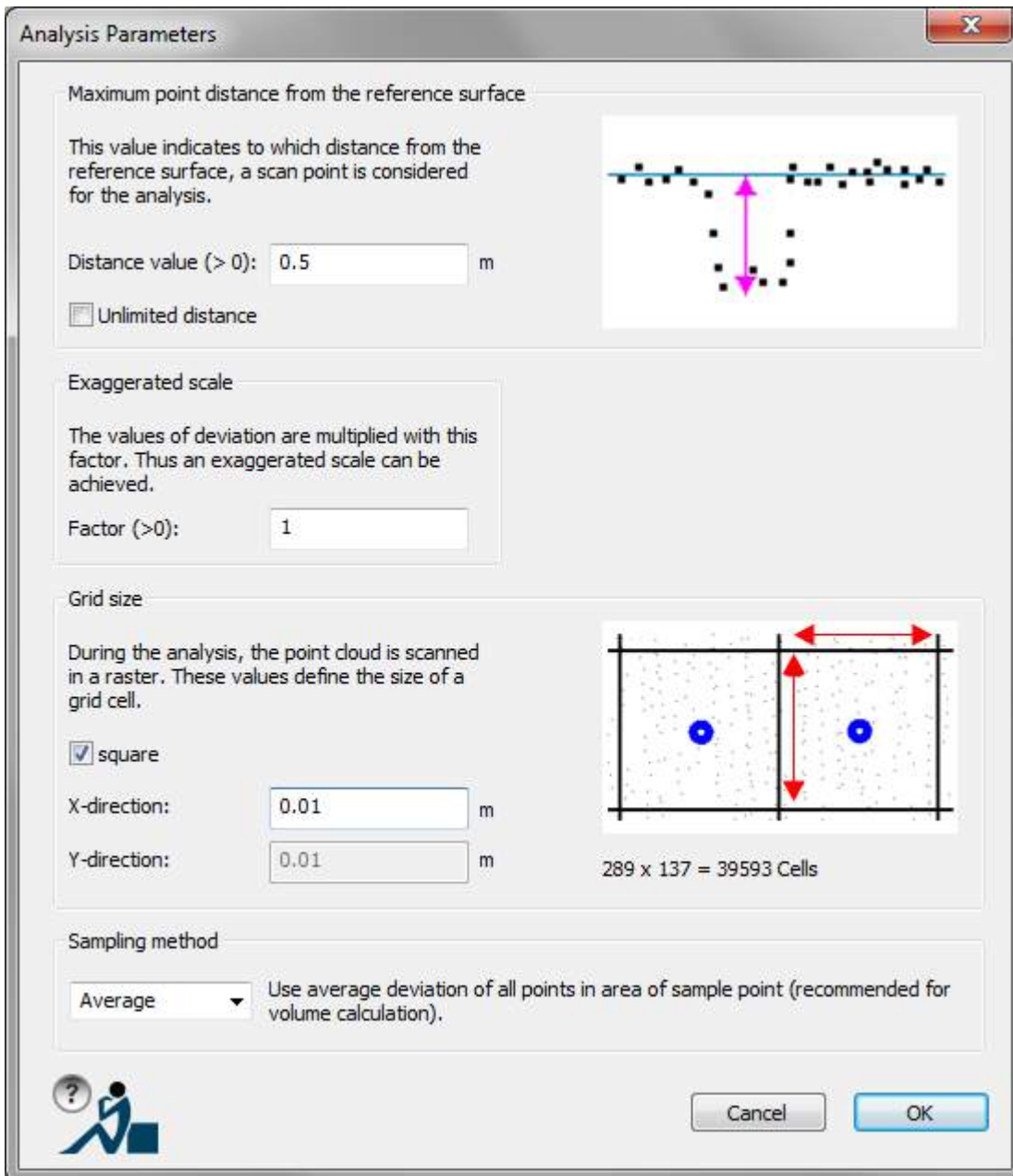


In the **"Reference surface object"** pane is the most important input. The plane that will be used as the reference plane for modelling, volume computations and/or deformations analysis is defined here.

If you want to use a kubit plane as the reference plane, select the option "kubit plane" and the button **[Select plane...]** will then be enabled. By clicking on this button the dialogue box will be temporarily hidden and you can click on the desired plane in the drawing.

The option "Automatically from UCS or kubit plane" refers to the **alignment of the grid** for the analysis/modelling. If it is ticked the grid will be automatically aligned to the axes of the selected reference plane. If it is not ticked, then after clicking on the button **[Alignment...]** you can yourself define the axes orientation by clicking on three points in the drawing.

In the **"Parameters"** pane, the second most important dialogue box lies behind the **[Settings...]** button. Here the parameters are set that will have a significant influence on the desired results of the analysis and modelling.



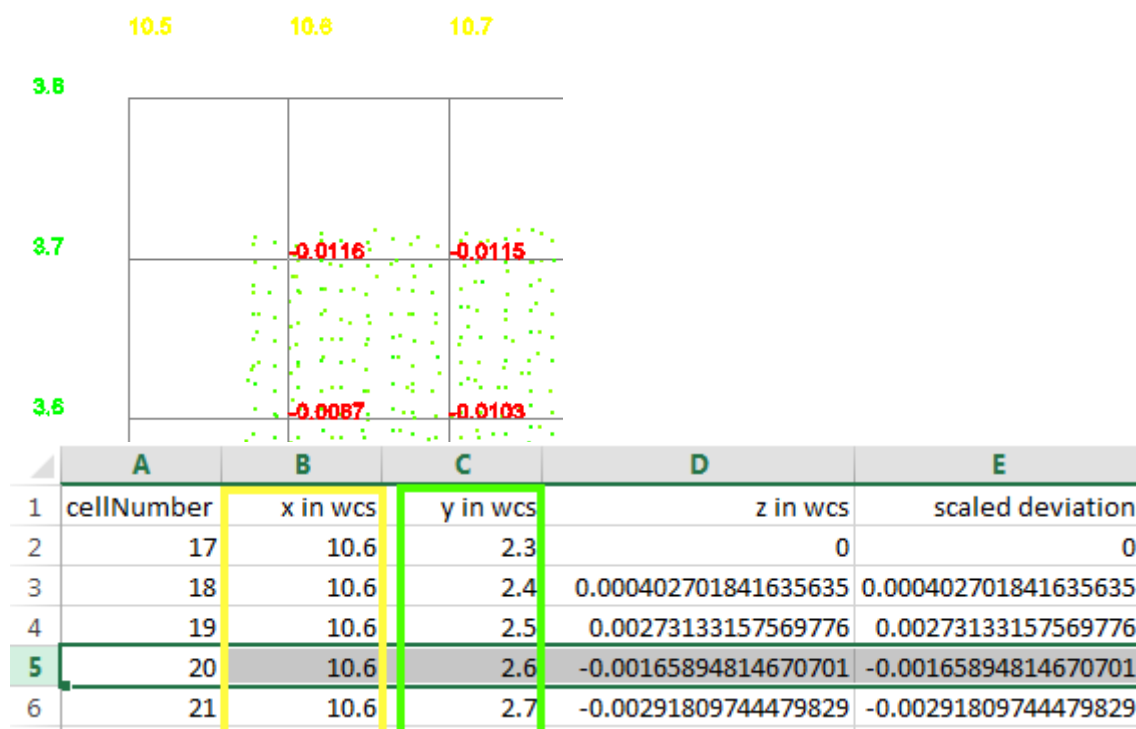
How these parameters influence the results will be explained below.

When you have set all of the parameters and selected the results that you want, you can start the analysis and modelling by clicking on the **[Start analysis]** button in the main dialogue box.

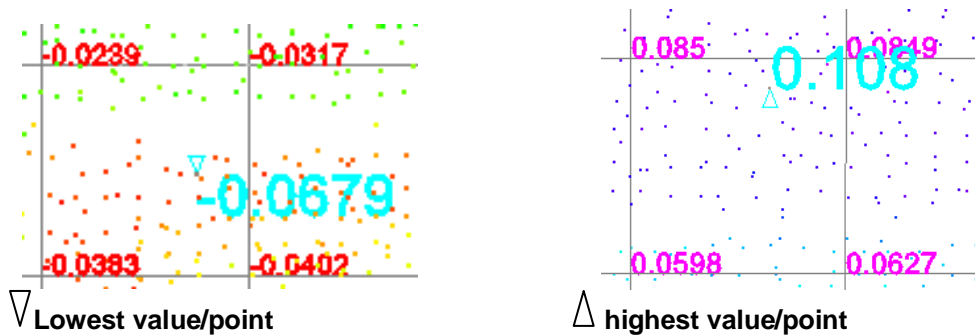
What results are possible?

in the "Results" pane you can choose depending on what task you are doing: Choose either flatness analysis or modelling (e.g. for volume computation), depending on which results that you want to generate:

- Solid:** Select this option when you want to model walls, floors, excavations or terrain and compute volumes. One or more AutoCAD solids will be created that represent the variances between the reference plane and the point cloud data. Should there be variances on both sides of the reference plane, then two solids will be created. Please note, if you want to create only one solid, then it is best to have the reference plane located somewhat outside of the point cloud. Otherwise it can lead to problems when modelling in AutoCAD, in the situation where the solid that is to be created is very thin and lies very close to the reference plane. The AutoCAD command **_MASSPROP** can be used to **determine the volume** of each of the solids. Otherwise you can with the aid of the AutoCAD commands for boolean operations further model the two solids (see the examples further below). (AutoCAD menu "Modify" ("Solid editing" → "Union"/"Subtract"/"Intersect").
- Polyline grid with deviations:** Creates 3D polylines, which form a grid that precisely nestles onto the point cloud, with the variance values shown as text on the vertices of the grid. Additionally the outermost grid lines are annotated with the x/y values, which are to be found in the ASCII table of the cell points. Thereby the deviation values from the ASCII table are to be found again in the drawing. The nomenclature is on a separate layer and can be hidden when they are not needed.

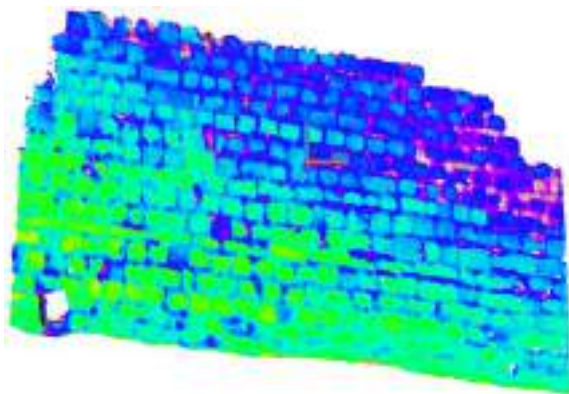


- **Grid of points:** Creates AutoCAD points on the vertices of the grid. A thinned out point raster is created, in comparison with the point cloud. How new points are defined in respect to the variance between the point cloud and the reference plane is explained below in a diagram for the sampling methods. By exporting these points as a DXF or DWG file they can then be further modelled in other systems. **Warning:** Please only use this option for smaller numbers of points, less than 10,000 points, otherwise AutoCAD will have problems dealing with such a large number of objects.
- **Label highest/lowest value:** This option creates an annotation, there in the point cloud where both of the absolute extreme points lay, in relation to the reference surface:



The tip of the small triangle marks the precise point. The value corresponds to the respective deviation from the reference surface.

- **Point cloud:** Creates a new point cloud file in E57 format, in which the deviations of the points from the reference plane are stored as intensity values. This point cloud is designed exclusively for data storage. It is used to create a colored AutoCAD point cloud (RCP) - like the one below - with the help of the PCDISTCOLEEDIT command (→**Colorize an analyzed point cloud**), which is automatically started in the next step.



- **ASCII sample points:** Creates an ASCII file (.txt) with all of the information of the polyline grid. They can be imported into other CAD systems and statistical software:

	A	B	C	D	E	F	
1	cellNumber	x in wcs	y in wcs	z in wcs	scaled deviation	cell status	
2	1	-1,26	-0,8	-0,374726745	-0,374726745	interpolated cell	
3	2	-1,26	-0,78	-0,367146304	-0,367146304	interpolated cell	
4	3	-1,26	-0,76	-0,367685248	-0,367685248	interpolated cell	
5	4	-1,26	-0,74	-0,352560606	-0,352560606	interpolated cell	
6	5	-1,26	-0,72	-0,353897607	-0,353897607	interpolated cell	
7	6	-1,26	-0,7	-0,355021704	-0,355021704	interpolated cell	
8	7	-1,26	-0,68	-0,348030334	-0,348030334	interpolated cell	
9	8	-1,26	-0,66	-0,358009711	-0,358009711	interpolated cell	
10	9	-1,26	-0,64	-0,359913085	-0,359913085	interpolated cell	

13205	13204	2,32	-0,58	-0,32377708	-0,32377708	valid cell	
13206	13205	2,32	-0,56	-0,32567931	-0,32567931	valid cell	
13207	13206	2,32	-0,54	-0,327182436	-0,327182436	valid cell	
13208	13207	2,32	-0,52	-0,328035691	-0,328035691	valid cell	
13209	13208	2,32	-0,5	-0,328120116	-0,328120116	valid cell	
13210	13209	2,32	-0,48	-0,330340374	-0,330340374	valid cell	
13211	13210	2,32	-0,46	-0,329626149	-0,329626149	valid cell	
13212	13211	2,32	-0,44	-0,328014568	-0,328014568	valid cell	
13213	13212	2,32	-0,42	-0,323993064	-0,323993064	valid cell	
13214	13213	2,32	-0,4	-0,325013864	-0,325013864	valid cell	
13215							

With the entries for "**Directory**" [...] and "**Name**" you define the file name and the path for the generated point cloud and ASCII file. The individual composite file names parts can be likewise edited. They are updated as soon as the main name is changed in the text box "Name".

All of these new results (solid objects, the deviation grid with its annotations, ...) that have been directly created in the drawing are automatically placed on a new layer, so that you can show and hide them separately. You can change the name of the default layer by using the As-Built command → **Settings** → Layer → Analysis Tools .

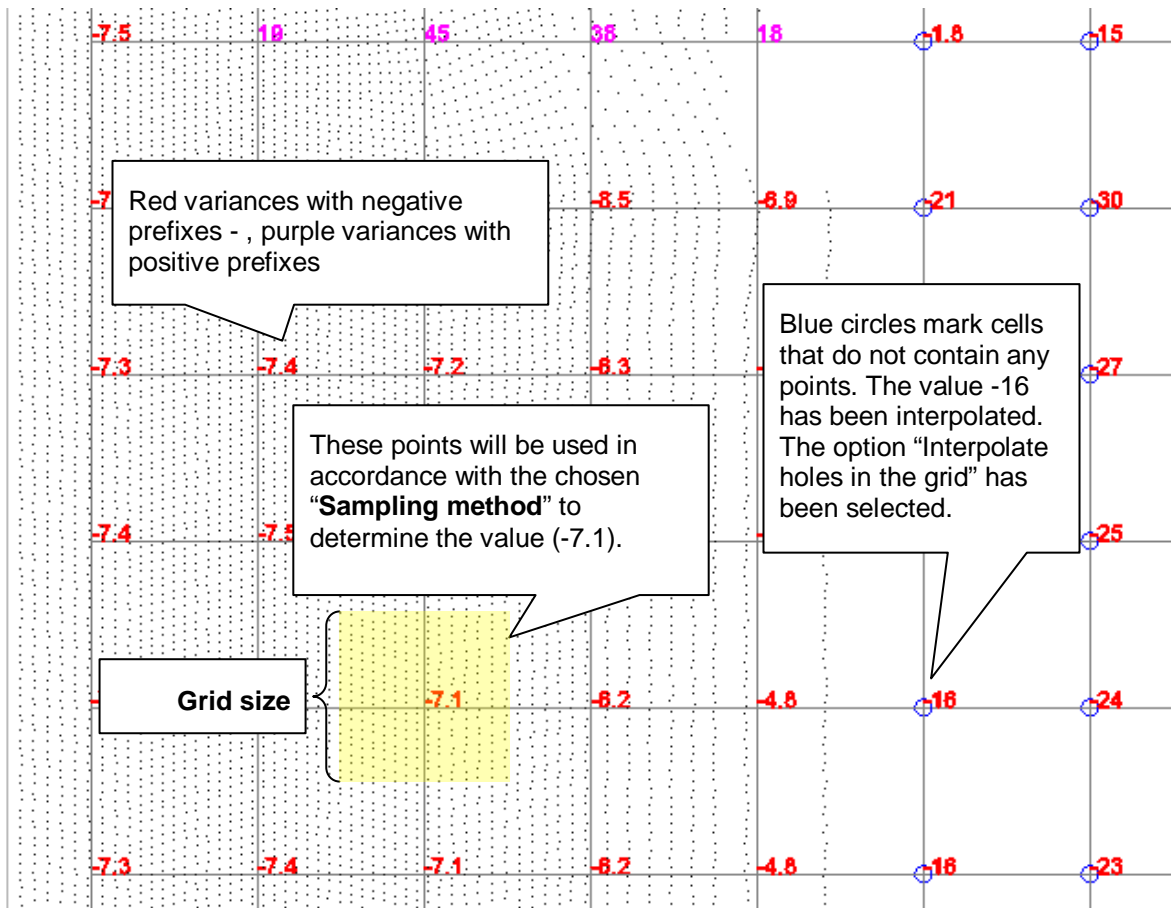
How do the parameters influence the results?

The parameter "**Maximum point distance from the reference surface**" of the Analysis Parameters dialogue box determines which points in the original point cloud should be used in the calculation. All of the points with a distance smaller than the value entered here will be used. If you can first clip out the point cloud, then you can enter here a relatively large number or simply just select the option "Unlimited distance". Only visible points are taken into consideration.

"Exaggerated scale": A value of 2 or 3 for the exaggerated scale significantly increases the displayed "hills" and "valleys" of the solids and polyline grids and make the variances much easier to see. The actual variance is simply multiplied by this factor. A factor of 1 gives the original variance. **Warning:** If you are modelling to scale and want to determine non-exaggerated volumes, then set this factor to a value of 1.

The **grid size** in the X and Y directions determines how many grid cells will be used in the calculation. If you set values that will require a very long time to compute (over 10000 cells), then when you close the parameter dialog box with [OK] a warning message will be displayed. The grid size influences the accuracy of the volumes, as does the resolution when modelling and the deformations analysis. *For computing more precise volumes we recommend that a smaller grid size and the "Average" sampling method be used.*

To explain the usual settings we shall take a look at the point cloud together with the generated polyline grid and the variance values. You should see something similar to this:



There are four sampling methods:

- **Average:** Averages all points within the area defined by the grid size around the grid position (recommended for volume computations)
- **Minimum:** The smallest absolute variance in the area around the grid position.
- **Maximum:** The largest absolute variance in the area around the grid position.
- **Nearest point:** The value will be used of the nearest point to the grid position.

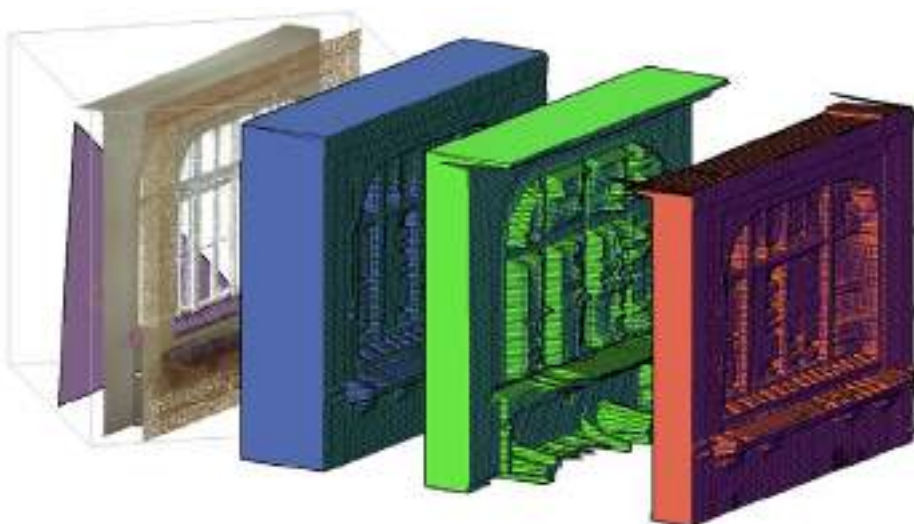
A **3D solid** will be created when the appropriate option has been selected. In the example shown below the **option "Interpolate holes in the grid"** has been selected, so that the missing upper corners and the scan holes in the wall will be "filled".

Note: The resolution of the surface of the solid is determined by the grid size. The exaggerated scale factor is also applied to the solid. Set this value to one when you want to create a model that is a true copy of the original.



From a point cloud, in which for example both sides of a wall have been scanned, you can with the sampling methods **Minimum** and **Maximum** determine the **solid for the wall** by combining together two plane analyses.

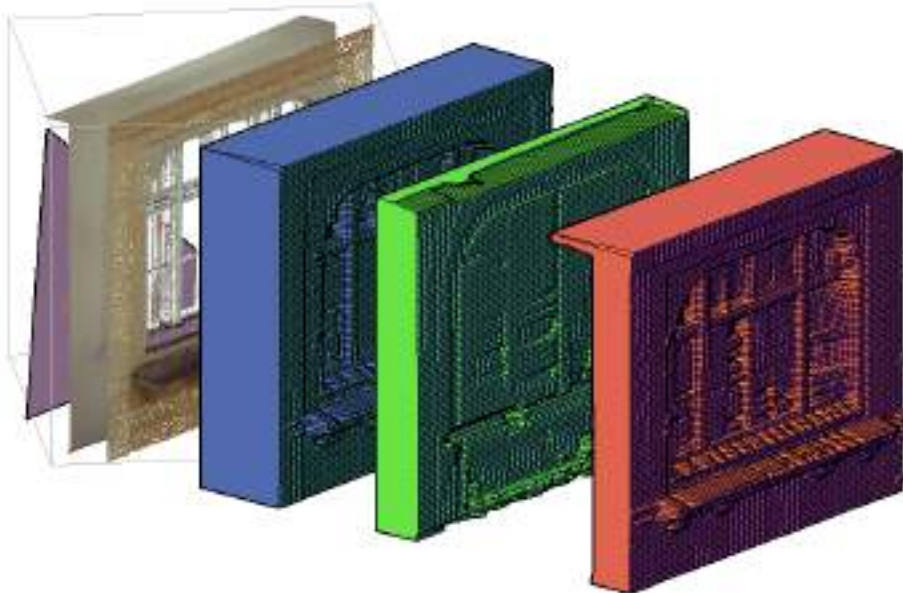
First the point cloud that is to be modelled is isolated. Next a kubit plane is placed outside of the point cloud, so that the kubit plane has approximately the same alignment as the wall and has a certain distance from all points in the point cloud (e.g. this can be done using fitting and parallel offset of the plane). Now to create the solid the analysis is carried out once using the sampling method Minimum and once using the sampling method Maximum. Using the two solids that have just been created the minimum solid will be subtracted from the maximum solid, with the aid of the AutoCAD command "SUBTRACT", and the remaining solid is that of the wall.



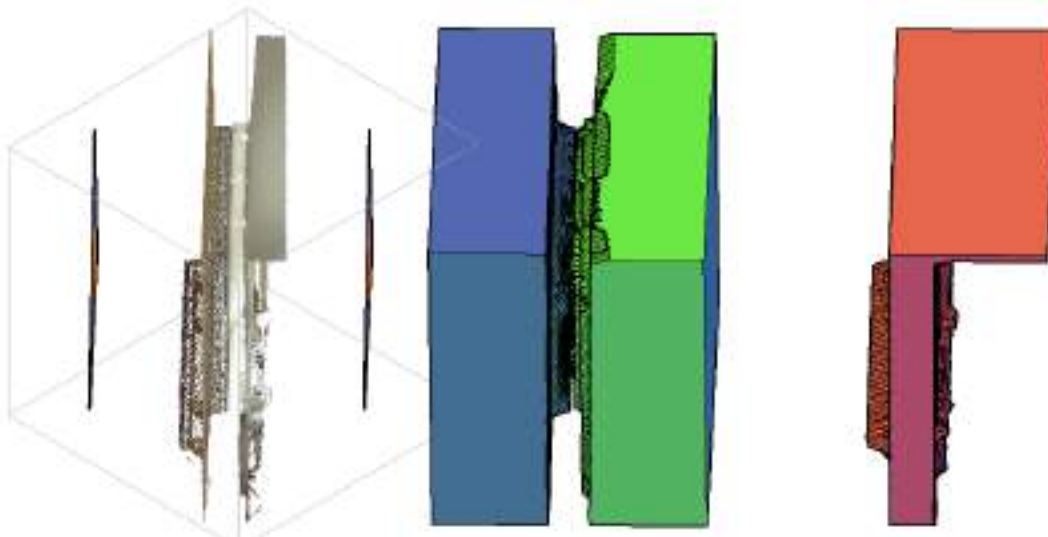
In the above figure you can see from left to right:

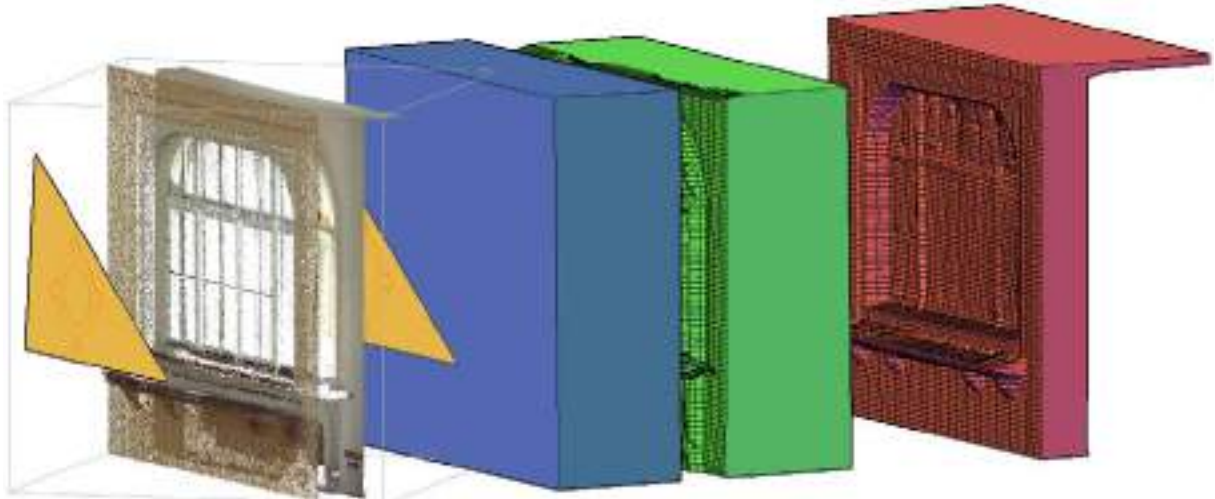
- purple - the kubit plane,
- the scanned point cloud of both sides of a window in a wall,
- blue – the maximum solid,
- green - the minimum solid and
- red - the resulting wall, produced by subtracting the minimum solid from the maximum solid.

Note: The sampling method jumps instead to the other side of the point cloud, where points are to be found (to be seen above as artefacts in the green minimum solid). To correct this, for creating the minimum solid the parameter "maximum point distance" should be set to the distance of the window pane to the kubit plane. This method has been used in the corrected error figure below.



Equally, it is possible to analyse a wall from both sides with the "**Average**" **sampling method** and a previously measured "maximum point distance" to the window pane. The distance can be measured with the aid of the function → **Modeling/Construction – Plane – Perpendicular projection of points onto a plane** (KPPERPRETURN). The wall is then created from these solids using the boolean operations in ("Modify" → "Solid Editing"). This wall is, because of the sampling method used, a truer solid and more error tolerant.






In the above figures you can see from left to right:

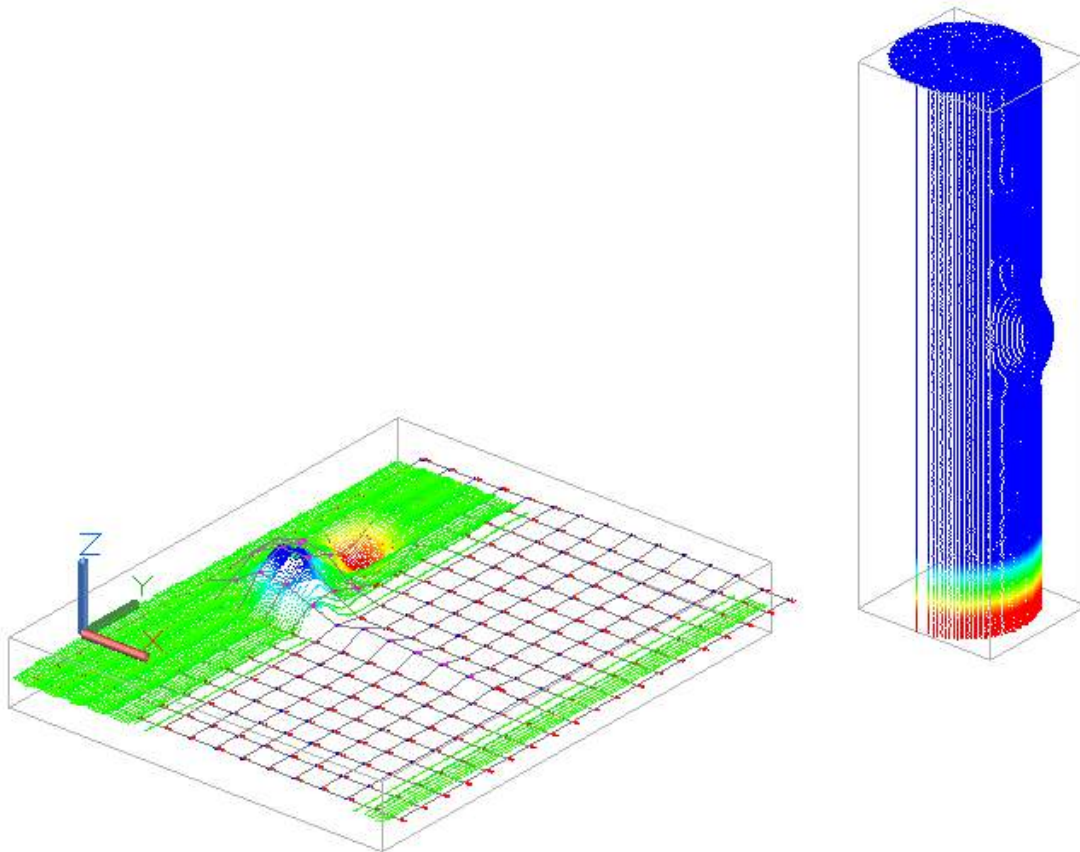
- the point cloud with its two kubit planes (beige),
- blue - the average solid from the outside,
- green - the average solid from the inside and
- red - solid of the wall, obtained by using the boolean subtraction (AutoCAD command "SUBTRACT") on both of the average resulting solids from a base cuboid. The base cuboid is formed using the outermost corner points of both of the average solids.

Analysis – Cylinder – Analysis of cylindrical shapes in a point cloud

	Command Name:	PCCYLINDERANALYSIS
	Ribbon Tab:	As-Built Analysis

Cylindrical shaped regions of a point cloud (e.g. a tank or a well) can be analysed in various ways with this feature. Deformations can be determined and visually displayed, **volume computations** can be made based on the real shape (including built up deposits: so called 'deadwood'). It is also easy to create an AutoCAD solid of the variances to the ideal cylinder as a separate object and get a visual display of it.

This command has many options and they can impact on each other. We therefore recommend that you read through the whole of this section before you start working with this powerful feature.

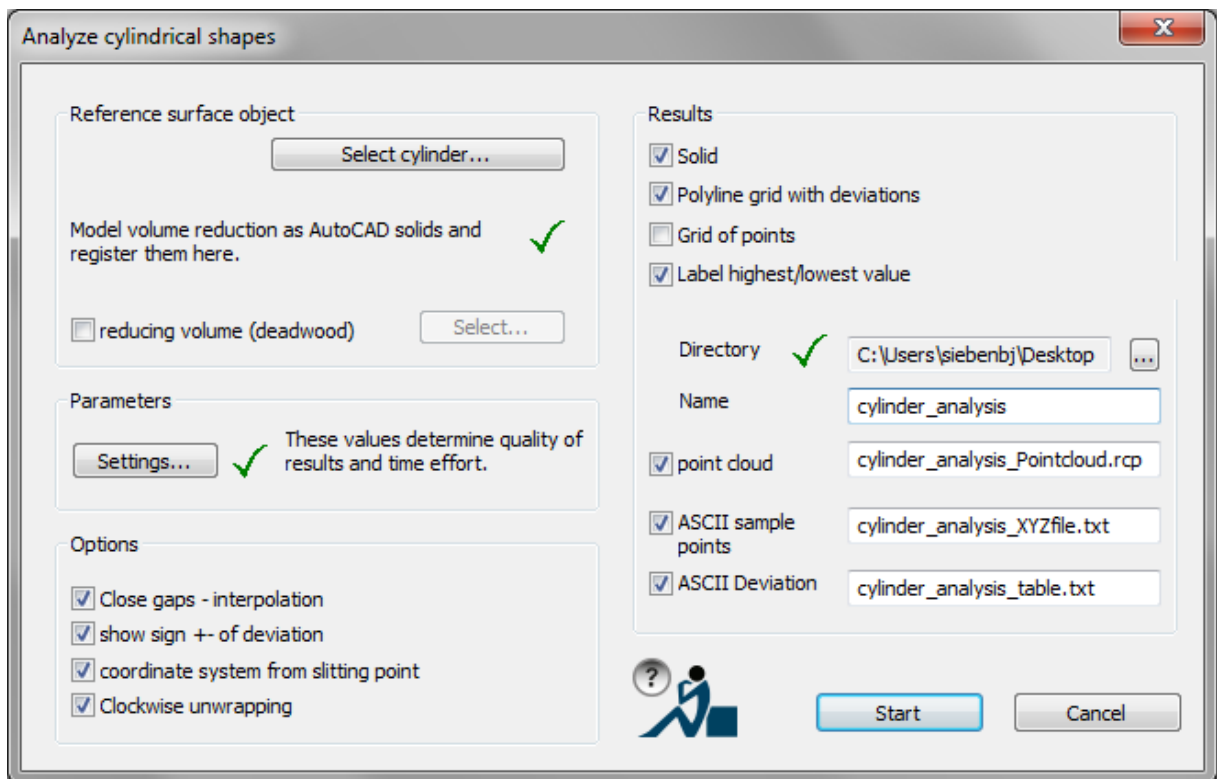


Initially open a drawing with an AutoCAD point cloud in it and fit a kubit cylinder, by using, for example, the command →**Modeling/Construction - Cylinder - Fit cylinder using visible points (PCFITCYL2)**.

The command PCCYLINDERANALYSIS opens a dialogue box where you can enter the basic settings to be used in the cylinder analysis.

Buttons with a question mark behind them, must first be clicked so that the button [Start analysis] can be enabled and the cylinder analysis can now start.

Here you can see a fully completed dialogue box:



In the "**Reference surface object**" pane, a click on the button [**Select cylinder...**] starts the most important input: The kubit cylinder, one of the theoretical shapes, should be selected and the break point should be defined. The break point is required for unwrapping the cylindrical point cloud onto the plane. It can later be used as the coordinate origin in the unwrapped analysis results.

You can, when computing volumes, optionally take account of a "**volume deduction (dead wood)**" in the form of an AutoCAD solid.

In the "**Parameters**" pane, the second most important dialogue box lies behind the [Settings...] button.

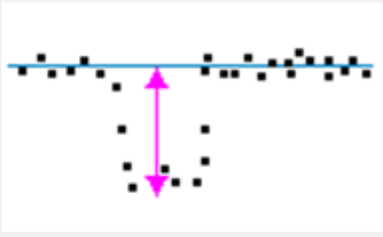
Here the parameters are set that will have a significant influence on the desired results of the analysis and modelling.

Analysis Parameters

Maximum point distance from the reference surface

This value indicates to which distance from the reference surface, a scan point is considered for the analysis.

Distance value (> 0): m



Exaggerated scale

The values of deviation are multiplied with this factor. Thus an exaggerated scale can be achieved.

Factor (>0):

Wall thickness

With positive values, inner volumes can be determined with scans from the outside.

Value: m

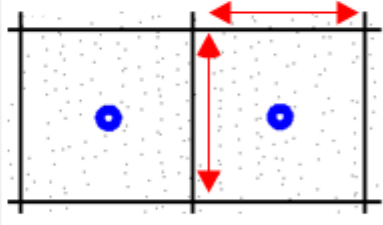
Grid size

During the analysis, the point cloud is scanned in a raster. These values define the size of a grid cell.

square

X-direction: m


Y-direction: m



1555 x 658 = 1023190 Cells

Sampling method

Use average deviation of all points in area of sample point (recommended for volume calculation).



How these parameters influence the results will be explained below.

In the "**Options**" pane you can decide if

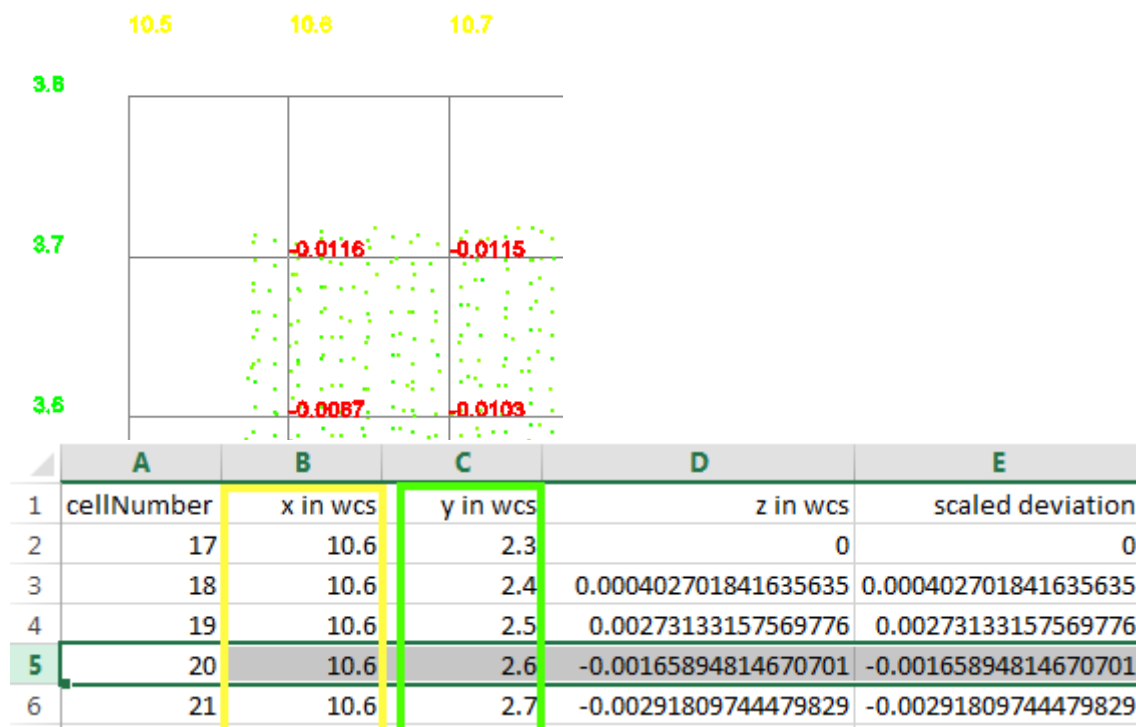
- holes in the raster grid should be interpolated or not
- the variances from the theoretical cylinder should have a prefix
- the coordinate system for the unwrapped results should be based on the break point
- it should be unwrapped in a clockwise direction

When you have set all of the parameters and selected the results that you want, you can start the analysis and modelling by clicking on the [**Start analysis**] button in the main dialogue box.

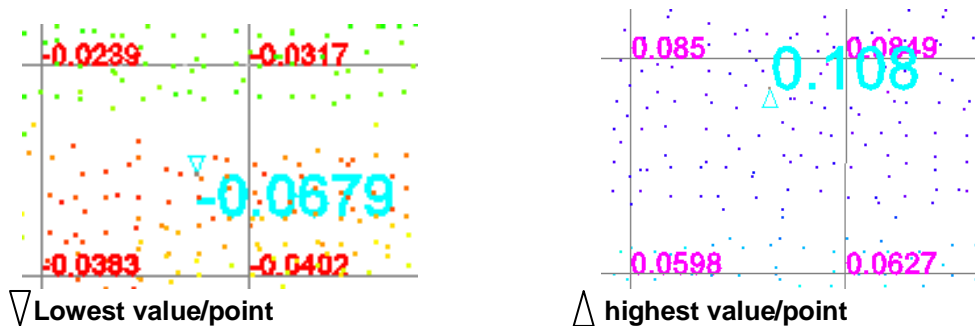
What results are possible?

In the "Results" pane you can choose depending on what task you are doing: deformation analysis, volume computation or modelling the results that you want:

- **Solid:** Select this option if you want a solid that is similar to the theoretical cylinder but generated from the point cloud as a deformed model, e.g. for wells, towers or similar.
- **Polyline grid with deviations:** Creates 3D polylines, which form a grid that precisely nestles onto the point cloud, with the variance values shown as text on the vertices of the grid. Additionally the outermost grid lines are annotated with the x/y values, which are to be found in the ASCII table of the cell points. Thereby the deviation values from the ASCII table are to be found again in the drawing. The nomenclature is on a separate layer and can be hidden when they are not needed.

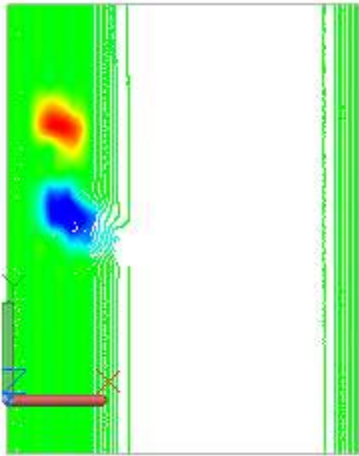


- **Grid of points:** A thinned out point raster is created, in comparison with the point cloud. How new points are defined in respect to the variance between the point cloud and the reference plane is explained below in a diagram for the sampling methods. By exporting these points as a DXF or DWG file they can then be further modelled in other systems. **Warning:** Please only use this option for smaller numbers of points, less than 10,000 points, otherwise AutoCAD will have problems dealing with such a large number of objects.
- **Label highest/lowest value:** This option creates an annotation, there in the point cloud where both of the absolute extreme points lay, in relation to the reference surface:



The tip of the small triangle marks the precise point. The value corresponds to the respective deviation from the reference surface.

- **Point cloud:** Creates an unwrapped planar point cloud file in E57 format, in which the deviations of the points from the reference plane are stored as intensity values. This point cloud is designed exclusively for data storage. It is used to create a colorized AutoCAD point cloud (RCP) - like the one below - with the help of the PCDISTCOLEEDIT command (→**Colorize an analyzed point cloud**), which is automatically started in the next step.



The **mean** plane is the unwrapped ideal cylinder, the points in the new cloud have the same distance to this plane, as previously to the skin of the reference cylinder. The origin of the new point cloud is precisely that of the **break point**. All of the points in the unwrapped cloud are defined in terms of the cylinder's coordinate system. Y is the elevation parallel to the cylinder axis. X is the arc length along the perimeter of the theoretical cylinder. Z is the variance to the theoretical cylinder.

- **ASCII sample points:** Creates an ASCII file (.txt) with all of the information of the polyline grid. They can be imported into other CAD systems and statistical software:

	A	B	C	D
1	1	55	-405,428406	-7,38408467
2	2	165	-405,428406	-7,37332778
3	3	275	-405,428406	-6,86403126
4	4	385	-405,428406	-6,3066185
5	5	495	-405,428406	-5,63825647
6	6	605	-405,428406	-4,85020989
7	7	715	-405,428406	-13,9978199
8	8	825	-405,428406	-20,1475717
9	9	935	-405,428406	-23,5950657
10	10	1045	-405,428406	-24,2106004
11	11	1155	-405,428406	-23,5579374
12	12	1265	-405,428406	-20,3703656
13	13	1375	-405,428406	-14,666373
14	14	1485	-405,428406	-6,09109434
15	15	1595	-405,428406	-6,8218704
16	16	55	-295,428406	-7,29281788
17	17	165	-295,428406	-7,23945586
18	18	275	-295,428406	-6,91977892
19	19	385	-295,428406	-6,38091893

- **ASCII deviation:** This option is a very powerful tool for **analysing deformations**. When imported into software such as Excel it looks like this:

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	orige	12.93°	17.82°	85.03°	360.3°								
2	arc length/height	33	183	270	1480	1230	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume	
3		-184.2	-1.3940467	-1.17131778	-8.8640112	-1.89424	-8.8208704	1478.631174	9402545.045	9401545.045	550000	8852545.045	8852545.045
4		-74.29	-1.29111788	-1.16645585	-8.8187385	-1.87494	-8.78020857	1478.591179	18775649.88	18177598.63	12815534.121	17029480.58	35281915.7
5		36.77	-1.17038181	-1.14015882	-8.8088875	-1.82494	-8.81142188	1479.721787	18788888.89	18888484.11	1812742.31	17896110.67	31988101.37
6		145.8	-1.11889318	-1.13542954	-7.8058891	-1.61777	-8.89855398	1472.94485	18638851.89	1853208.65	521474.698	18113183.85	6309335.18
7		255.8	-1.17347215	-1.18870542	-7.1233862	-1.49277	-8.5252417	1471.17918	18577301.65	84080312.84	101252.4004	18383811.18	86417144.34
8		365.8	-1.40328861	-1.45764878	-7.8947364	-1.48818	-8.92727824	1466.832438	18481291.63	182548411.7	1194446.733	18260749.28	96717811.63
9		475.8	-1.38298457	-1.41775577	-7.1587597	-1.61725	-8.8816418	1463.116362	18381671.11	120809388	2812598.683	18518111.63	112669385.8
10		585.8	-1.39832393	-8.24939314	-8.3378938	-1.88238	-8.88164181	1459.890257	18214094.5	133123418.1	812983.837	17318894.43	130069385.7
11		695.8	-1.34133393	26.8844849	45.190333	-1.77917	-7.11679324	1461.81438	1322813.67	12831179.13	0	15228110.67	14538708.8
12		805.8	-1.41889215	51.5554738	80.681148	-1.60031	-7.03012608	1544.128841	38889407.12	17998184.7	0	26889407.12	188897118.1
13		915.8	-1.51881882	51.808566	80.458882	-1.49514	-7.0465001	1557.884304	26812558.57	189491524.8	0	26812558.57	180128478.1
14		1025.8	-1.71229488	31.8481818	26.631888	-1.67887	-7.18939889	1618.928887	18188888.88	21798811.8	0	18188888.88	28812988.8
15		1135.8	-1.91888818	55.181542	55.184257	-1.86952	-7.13964598	1427.813848	18473441.14	174481854.8	0	18473441.14	225495498.1
16		1245.8	-1.88848872	59.884287	58.798338	-1.83319	-7.12782388	1371.420825	15493256.76	24952811.4	0	13493136.78	24835568.8
17		1355.8	-1.55093042	-41.484044	-85.584844	-1.64187	-7.18489985	1256.787775	15167648.12	26528682.1	0	15167648.12	256127408.5
18		1465.8	-1.88821256	-1.75288888	-7.8787311	-1.57882	-7.28189788	1352.782610	15488884.26	188681884.8	0	15488884.26	271586487.8
19		1575.8	-1.78877724	-3.818788	-7.4491888	-1.88341	-7.19429842	1312.58888	1898818.47	23398813.8	0	1498818.47	28838887.8
20		1685.8	-1.72122497	-7.7441888	-7.4548888	-1.88832	-7.13881498	1318.828889	13818847.8	16812881.4	0	13818847.8	16812881.4
21		1795.8	-1.98788178	-197.883287	-198.44878	-1.88825	-1.98188274	1351.888887	18188889.885	111818273.4	0	18188889.885	18188889.885

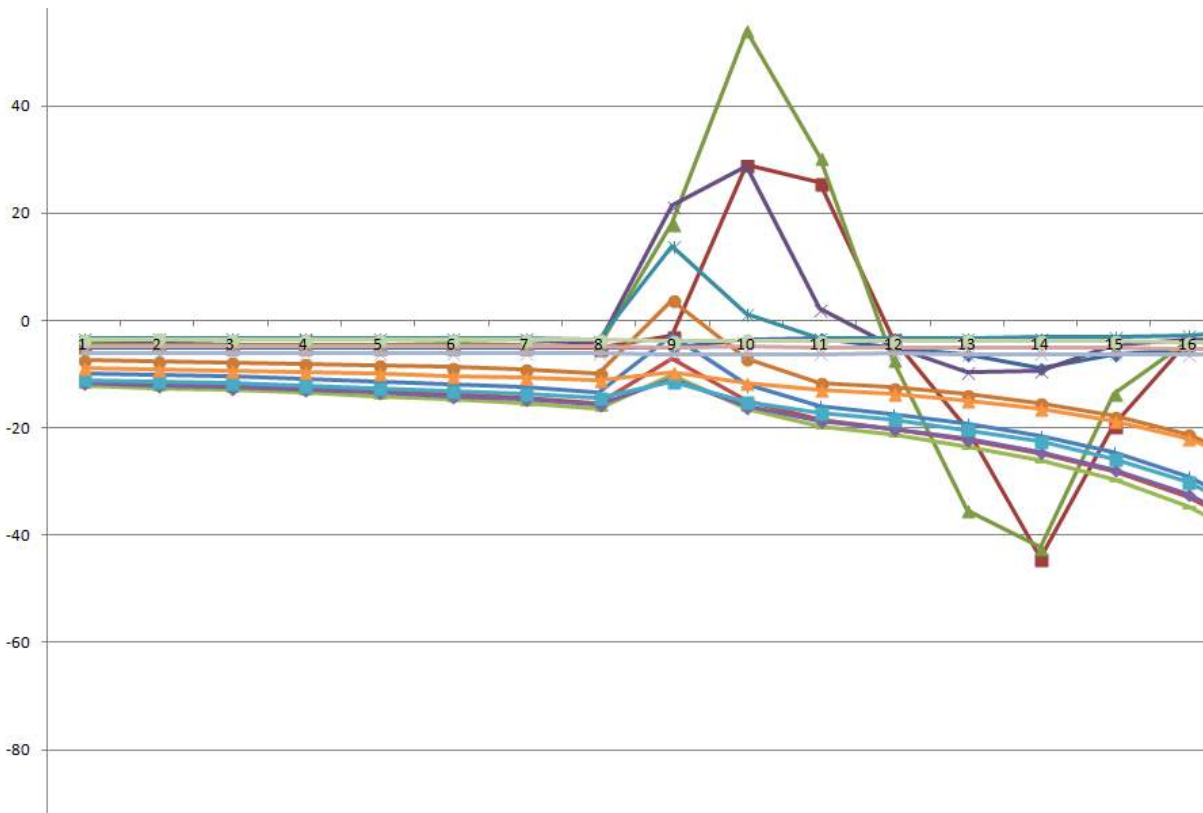
From the third line, the table contains the variance values of the grid beginning at the bottom edge of the cylinder.

The last line belongs to the top slice of the theoretical cylinder. Especially interesting are the last six columns. They contain the volumes for every line (that means every slice of the object) and the total volume from the base of the cylinder to the slice. If, in the dialogue box, Volume deduction (deadwood) was selected, then not only would the deducted volumes be shown but also the totals too. In the column after the variances the circumferences of the individual slices are also shown.

Note: The accuracy of all of the stated volumes and circumferences are defined by the parameters "Grid size" and "Sampling method. For computing more

precise volumes we recommend that a smaller grid size and the "Average" sampling method be used.

From the table an Excel diagram that looks like this can also be generated:

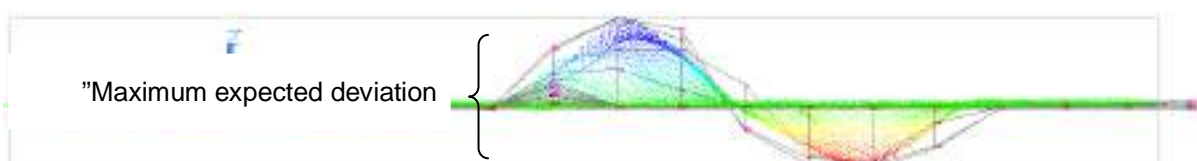


With "Directory" [...] and "Name" you define the file name and the path for the generated results. The individual composite file names can be likewise edited, they are updated as soon as the main name is changed in the text box "Name".

All of these new results (solid objects, the deviation grid with its annotations, ...) that have been directly created in the drawing are automatically placed on a new layer, so that you can show and hide them separately. You can change the name of the default layer by using the As-Built command → **Settings** → Layer → Analysis Tools .

How do the parameters influence the results?

The parameter "**Maximum point distance from the reference surface**" of the Analysis Parameters dialogue box determines which points in the original point cloud should be used in the calculation. All of the points inside of and outside of the theoretical cylinder with a distance smaller than the value entered here will be used. If you first clip out the cylinder, then you can enter here a relatively large number. *Only visible points* are taken into consideration.



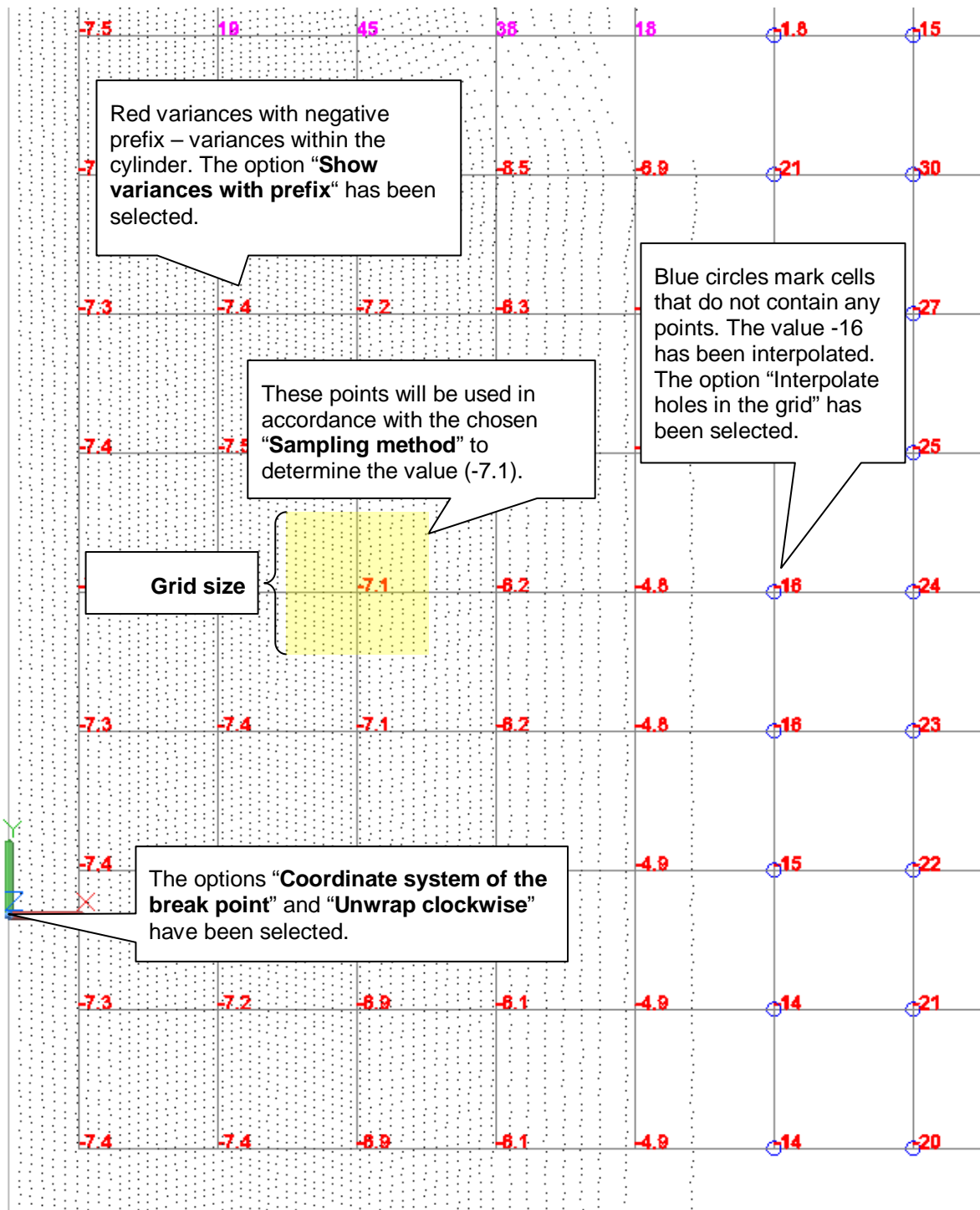
“Exaggerated scale”: A value of 2 or 3 for the exaggerated scale significantly increases the displayed "hills and "valleys" of the solids and polyline grids and make the variances much easier to see. The actual variance is simply multiplied by this factor. A factor of 1 gives the original variance. **Warning**: If you are modelling to scale and want to determine non-exaggerated volumes, then set this factor to a value of 1.

The **grid size** in the X and Y directions determines how many grid cells will be used in the calculation. Both of the directions represent the arc length and the elevation of the cylinder grid cells. If you set values that will require a very long time to compute (over 10000 cells), then when you close the parameter dialog box with [OK] a warning message will be displayed. **Note**: The grid size influences the accuracy of the volumes, as does the resolution when modelling and the deformations analysis. *For computing more precise volumes we recommend that a smaller grid size and the "Average" sampling method be used.*



The **wall thickness** defines a value that will be subtracted from all of the calculated cell values. Positive values, for external scans, will allow internal volumes to be determined (e.g. for tanks). In the opposite way, negative values for internal scans will similarly produce external volumes.


To explain the usual settings we shall take a look at the unwrapped point cloud together with the generated polyline grid and the variance values. You should see something similar to this:



There are four sampling methods::

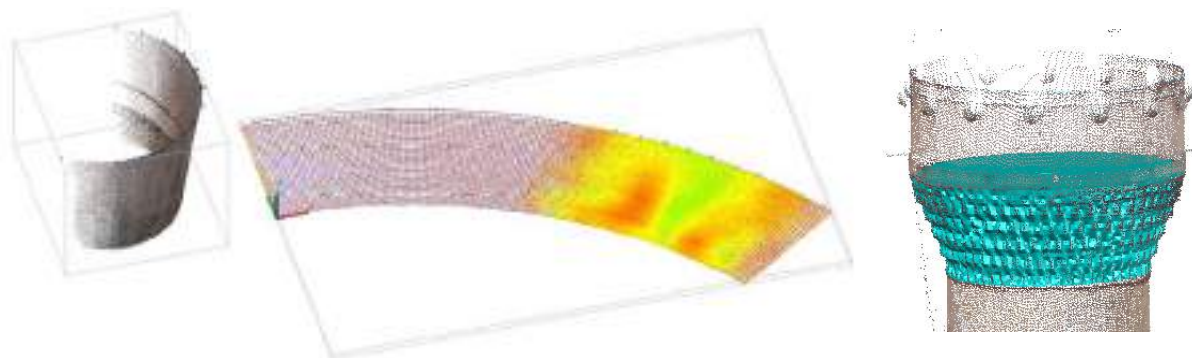
- **Average:** Averages all points within the area defined by the grid size around the grid position (recommended for volume computations)
- **Minimum:** The smallest absolute variance in the area around the grid position.
- **Maximum:** The largest absolute variance in the area around the grid position.
- **Nearest point:** The value will be used of the nearest point to the grid position.

Analysis – Analyze frustum of a cone in a point cloud

	Command name:	PCFRUSTOCONICALANALYSIS
	Ribbon Tab:	As-Built Analysis

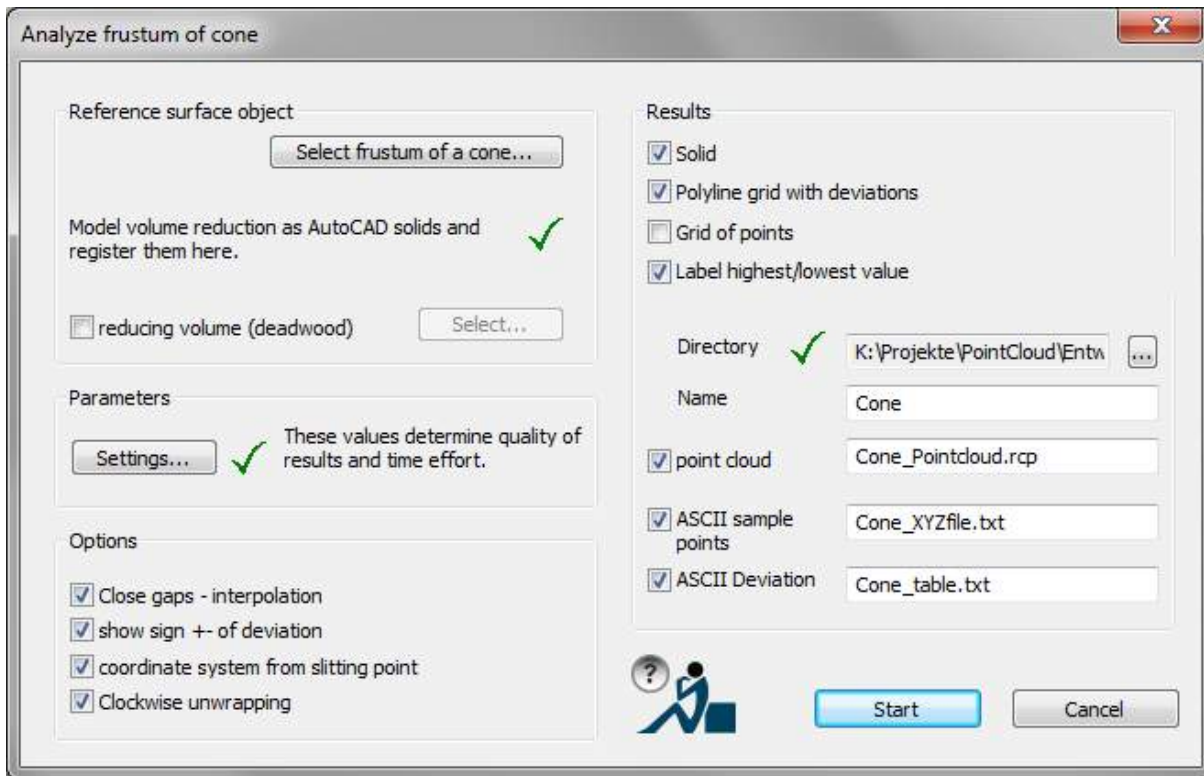
Conical shaped regions of a point cloud (e.g. a tank or a tower) can be analysed in various ways with this feature. **Deformations** can be determined and displayed. **Volume computations** can be made based on the real shape (including deformations and built up deposits: so called 'deadwood'). It is also easy to create an AutoCAD solid of the variances to the ideal truncated cone as a separate object and get a visual display of it.

This command has many options and they can impact on each other. We therefore recommend that you read through the whole of this section before you start working with this powerful feature.



Initially open a drawing with an AutoCAD point cloud in it and fit a kubit truncated cone, by using, for example, the command →**Modeling/Construction - Truncated cones - Fit elliptical truncated cone** (FCCREATEELLIPSIS).

The command PCFRUSTOCONICALANALYSIS opens a dialogue box where you can enter the basic settings to be used in the truncated cone analysis. Buttons with a question mark behind them, must first be clicked so that the button [Start analysis] can be enabled and the cone analysis can now start. Here you can see a fully completed dialogue box:



In the "**Reference surface object**" pane, a click on the button **[Select cone...]** starts the most important input: For the kubit truncated cone, one of the theoretical shapes should be selected and the break point should be defined. The break point is required for unwrapping the conical point cloud onto the plane. It can later be used as the coordinate origin in the unwrapped analysis results.

You can, when computing volumes, optionally take account of a "**volume deduction (dead wood)**" in the form of an AutoCAD solid.

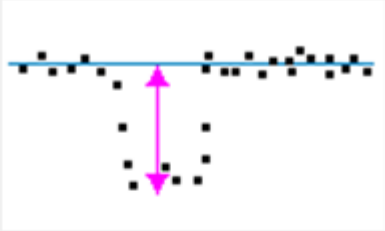
In the "**Parameter**" pane, the second most important dialogue box lies behind the **[Settings...]** button. Here the parameters are set that will have a significant influence on the desired results of the analysis and modelling.

Analysis Parameters X

Maximum point distance from the reference surface

This value indicates to which distance from the reference surface, a scan point is considered for the analysis.

Distance value (> 0): m



Exaggerated scale

The values of deviation are multiplied with this factor. Thus an exaggerated scale can be achieved.

Factor (>0):

Wall thickness

With positive values, inner volumes can be determined with scans from the outside.

Value: m

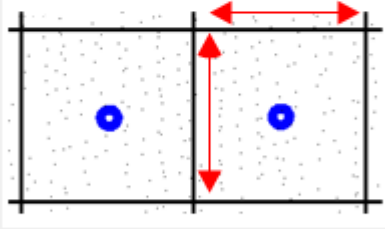
Grid size

During the analysis, the point cloud is scanned in a raster. These values define the size of a grid cell.

square

X-direction: approx. m


Y-direction: approx. m



1976 x 248 = 490048 Cells

Sampling method

Use average deviation of all points in area of sample point (recommended for volume calculation).



How these parameters influence the results will be explained below.

In the "**Options**" pane you can decide if

- holes in the raster grid should be interpolated or not
- the variances from the theoretical cylinder should have a prefix
- the coordinate system for the unwrapped results should be based on the break point
- it should be unwrapped in a clockwise direction

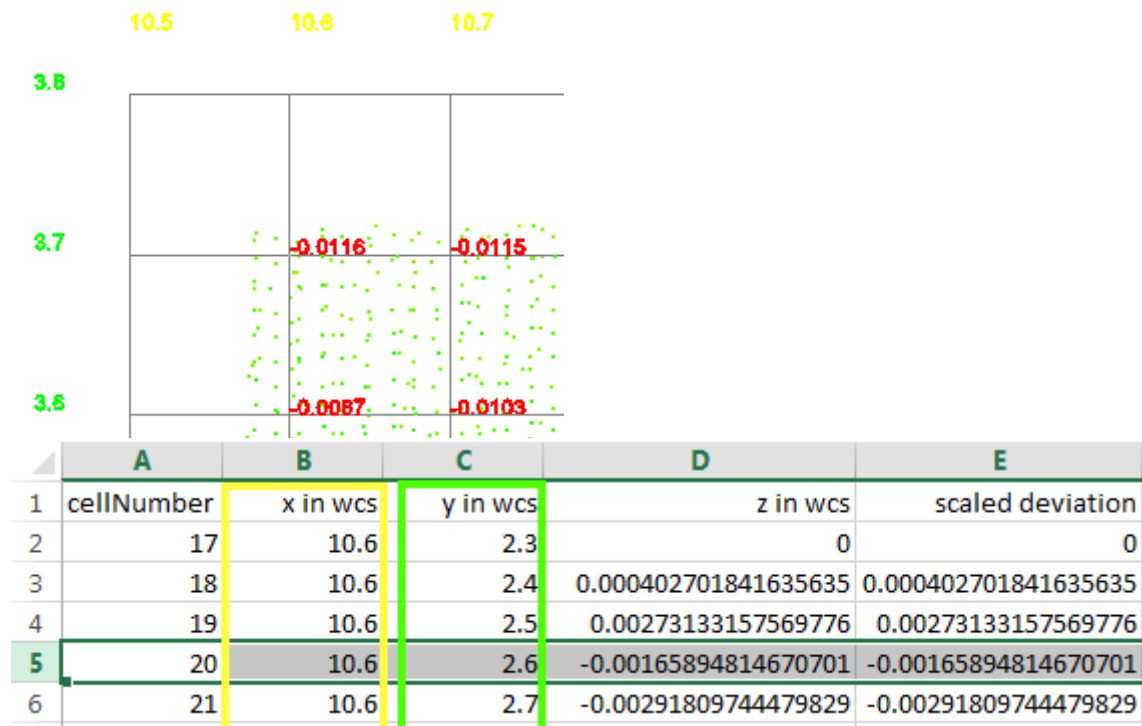
When you have set all of the parameters and selected the results that you want, you can start the analysis and modelling by clicking on the **[Start analysis]** button in the main dialogue box.

What results are possible?

In the "Results" pane you can choose depending on what task you are doing:

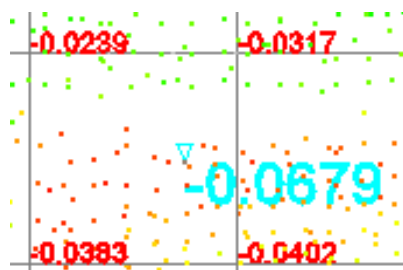
Deformation analysis, volume computation or modelling and what results you want:

- **Solid:** Select this option if you want a solid that is similar to the theoretical truncated cone but generated from the point cloud as a deformed model, e.g. for fountains, towers or similar.
- **Polyline grid with deviations:** Creates 3D polylines, which form a grid that precisely nestles onto the point cloud, with the variance values shown as text on the vertices of the grid. Additionally the outermost grid lines are annotated with the x/y values, which are to be found in the ASCII table of the cell points. Thereby the deviation values from the ASCII table are to be found again in the drawing. The nomenclature is on a separate layer and can be hidden when they are not needed.

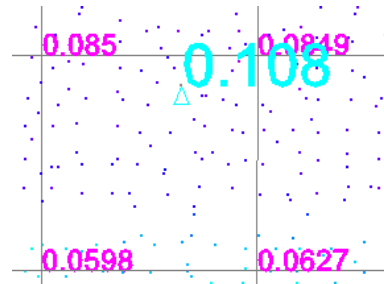


- **Grid of points:** Creates AutoCAD points on the vertices of the grid. A thinned out point raster is created, in comparison with the point cloud. How new points are defined in respect to the variance between the point cloud and the reference plane is explained below in a diagram for the sampling methods. By exporting these points as a DXF or DWG file they can then be further modelled in other systems. **Warning:** Please only use this option for smaller numbers of points, less than 10,000 points, otherwise AutoCAD will have problems dealing with such a large number of objects.

- **Label highest/lowest value:** This option creates an annotation, there in the point cloud where both of the absolute extreme points lay, in relation to the reference surface:



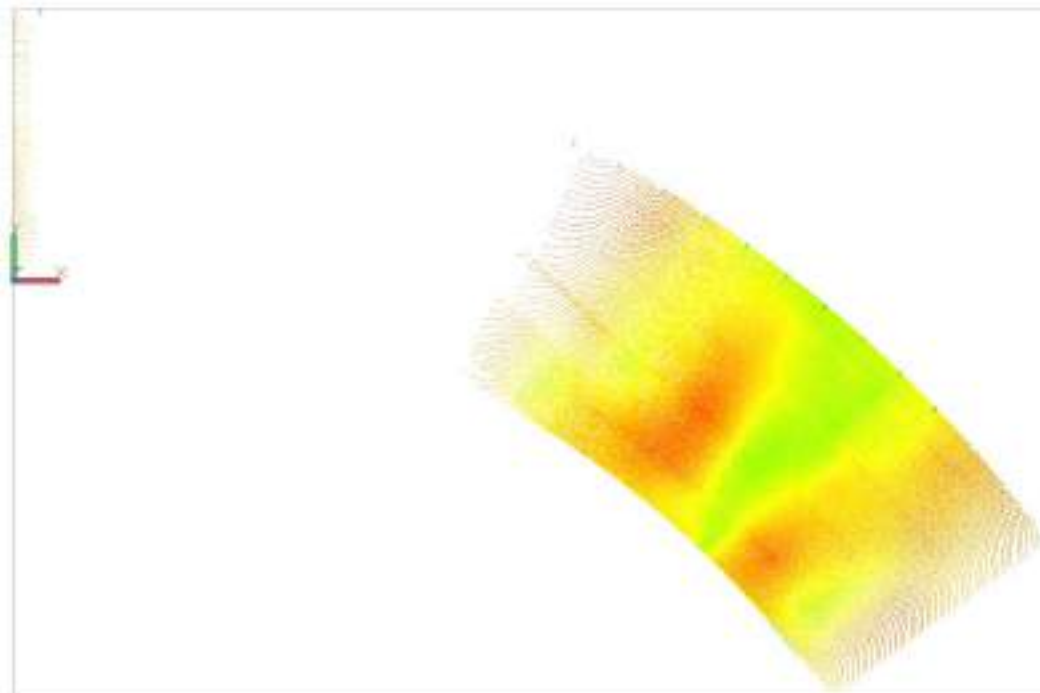
▽ Lowest value/point



△ highest value/point

The tip of the small triangle marks the precise point. The value corresponds to the respective deviation from the reference surface.

- **Point cloud:** Creates a new planar point cloud file in E57 format, in which the deviations of the points from the reference plane are stored as intensity values. This point cloud is designed exclusively for data storage. It is used to create a colorized AutoCAD point cloud (RCP) - like the one below - with the help of the PCDISTCOLEEDIT command (→**Colorize an analyzed point cloud**), which is automatically started in the next step.



The **mean** plane is the unwrapped ideal truncated cone, the points in the new cloud have the same distance to this plane, as previously to the skin of the reference cone. The origin of the new point cloud is precisely that of the **break point**. All of the points in the unwrapped cloud are defined in terms of the cone's coordinate system. Y is the elevation parallel to the cone axis. X is the arc length along the perimeter of the theoretical cone. Z is the variance to the theoretical cone.

- ASCII sample points:** Creates an ASCII file (.txt) with all of the information of the polyline grid. They can be imported into other CAD systems and statistical software:

	A	B	C	D
1	1	55	-405,428406	-7,38408467
2	2	165	-405,428406	-7,37332778
3	3	275	-405,428406	-6,86403126
4	4	385	-405,428406	-6,3066185
5	5	495	-405,428406	-5,63825647
6	6	605	-405,428406	-4,85020989
7	7	715	-405,428406	-13,9978199
8	8	825	-405,428406	-20,1475717
9	9	935	-405,428406	-23,5950657
10	10	1045	-405,428406	-24,2106004
11	11	1155	-405,428406	-23,5579374
12	12	1265	-405,428406	-20,3703656
13	13	1375	-405,428406	-14,666373
14	14	1485	-405,428406	-6,09109434
15	15	1595	-405,428406	-6,8218704
16	16	55	-295,428406	-7,29281788
17	17	165	-295,428406	-7,23945586
18	18	275	-295,428406	-6,91977892
19	19	385	-295,428406	-6,38091893

- ASCII deviation:** This option is a very powerful tool for **analysing deformations**. When imported into software such as Excel it looks like this:

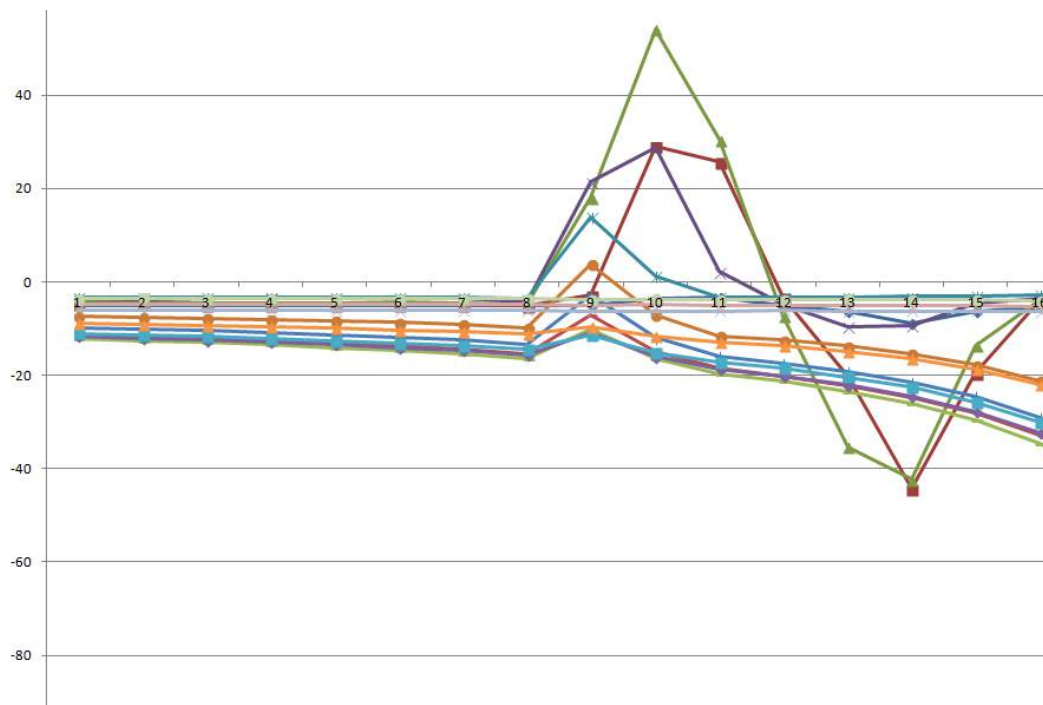
A	B	C	D	E	F	G	H	I	J	K	L	M
1	1	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
2	2	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
3	3	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
4	4	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
5	5	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
6	6	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
7	7	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
8	8	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
9	9	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
10	10	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
11	11	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
12	12	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
13	13	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
14	14	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
15	15	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
16	16	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
17	17	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
18	18	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
19	19	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
20	20	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume
21	21	1241'	1758'	8500'	1480	1390	circumference	slice volume	total volume	deadwood slice volume	reduced slice volume	reduced total volume

From the third line, the table contains the variance values of the grid beginning at the bottom edge of the truncated cone.

The last line belongs to the top slice of the theoretical truncated cone. Especially interesting are the last six columns. They contain the volumes for every line (that means every slice of the object) and the total volume from the base of the cylinder to the slice. If, in the dialogue box, Volume deduction (deadwood) was selected, then not only would the deducted volume be shown but also the totals too. In the column after the variances the circumferences of the individual slices are also shown.

Note: The accuracy of all of the stated volumes and circumferences are defined by the parameters "Grid size" and "Sampling method". For computing more precise volumes we recommend that a smaller grid size and the "Average" sampling method be used.

From the table an Excel diagram that looks like this can also be generated:



With "Directory" [...] and "Name" you define the file name and the path for the generated results. The individual composite file names can be likewise edited, they are updated as soon as the main name is changed in the text box "Name".

All of these new results (solid objects, the deviation grid with its annotations, ...) that have been directly created in the drawing are automatically placed on a new layer, so that you can show and hide them separately. You can change the name of the default layer by using the As-Built command → **Settings** → Layer → Analysis Tools.

How do the parameters influence the results?

The parameter "**Maximum point distance from the reference surface**" of the Analysis Parameters dialogue box determines which points in the original point cloud should be used in the calculation. All of the points inside of and outside of the theoretical cone with a distance smaller than the value entered here will be used. If you first clip out the cone, then you can enter here a relatively large number. *Only visible points* are taken into consideration.



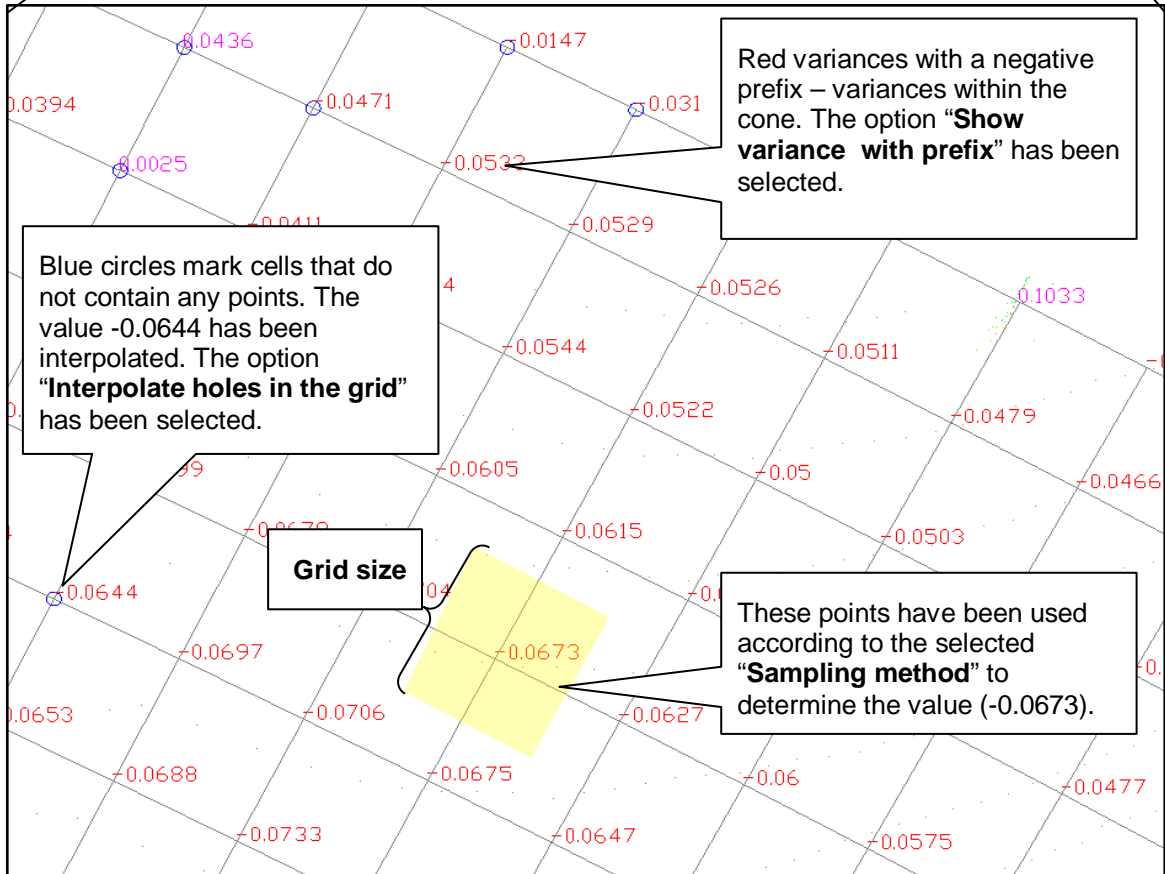
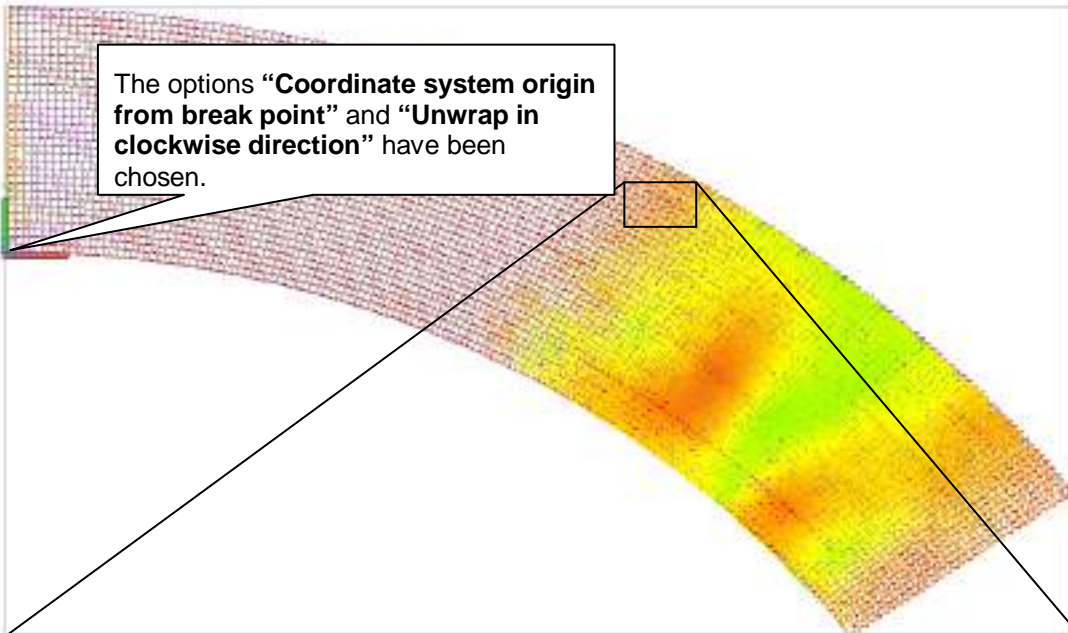
"Exaggerated scale": A value of 2 or 3 for the exaggerated scale significantly increases the displayed "hills and "valleys" of the solids and polyline grids and make the variances much easier to see. The actual variance is simply multiplied by this factor. A factor of 1 gives the original variance. **Warning:** If you are modelling to scale and want to determine non-exaggerated volumes, then set this factor to a value of 1.

The **grid size** in the X and Y directions determines how many grid cells will be used in the calculation. Both of these directions represent the either the upper or the lower arc length and the elevation of the truncated cone grid cells. It will be the longer arc length (upper or lower) of the truncated cone that will be used as the reference distance. The grid size will be fitted in both directions on the cone to obtain an equally spaced grid. If you set values that will require a very long time to compute (over 10000 cells), then when you close the parameter dialog box with [OK] a warning message will be displayed. **Note:** The grid size influences the accuracy of the volumes, as does the resolution when modelling and the deformations analysis. *For computing more precise volumes we recommend that a smaller grid size and the "Average" sampling method be used.*



The **wall thickness** defines a value that will be subtracted from all of the calculated cell values. Positive values, for external scans, will allow internal volumes to be determined (e.g. for tanks).

In the opposite way, negative values for internal scans will similarly produce external volumes. To explain the usual settings we shall take a look at the unwrapped point cloud together with the generated polyline grid and the variance values. You should see something similar to this:




There are four sampling methods:

- **Average:** Averages all points within the area defined by the grid size around the grid position (recommended for volume computations, see below)
- **Minimum:** The smallest absolute variance in the area around the grid position.
- **Maximum:** The largest absolute variance in the area around the grid position.
- **Nearest point:** The value will be used of the nearest point to the grid position.

10.3 As-Built Plant Tools

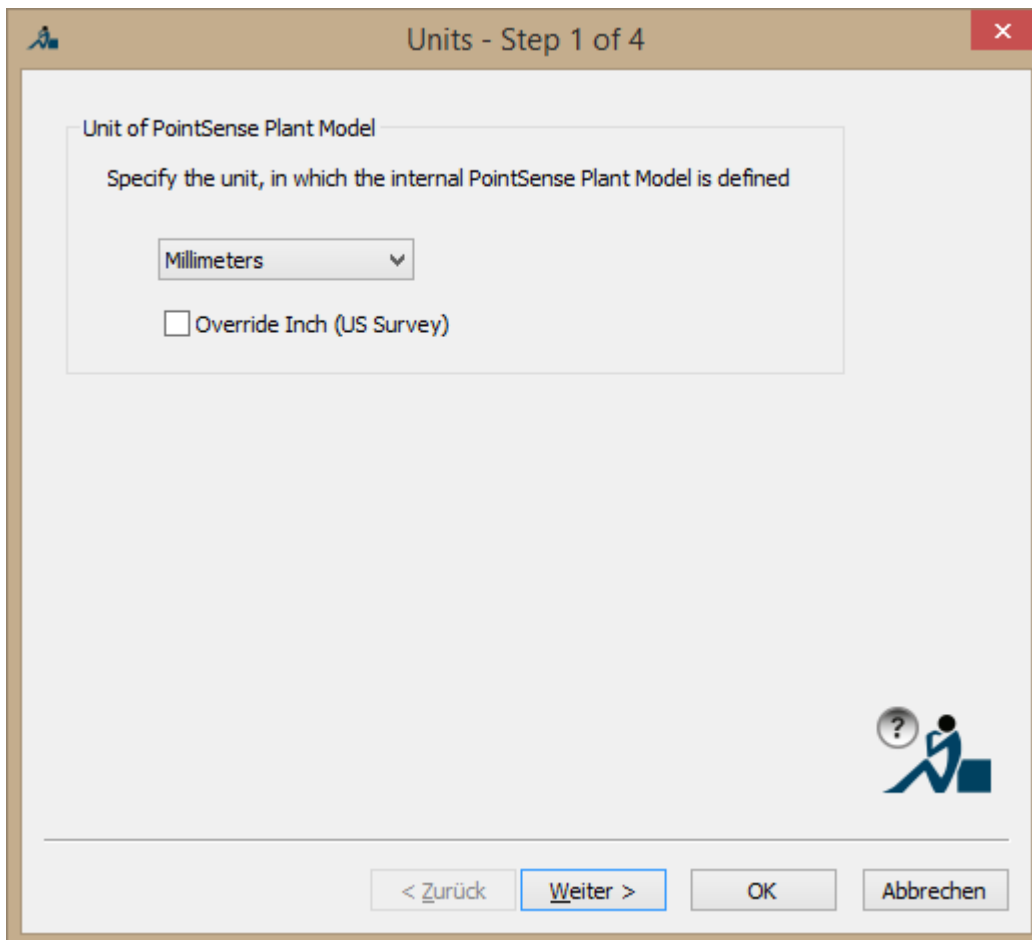
Prepare – Settings

	Command Name:	PCCFG
	Ribbon Tab:	As-Built Plant

With the help of a wizard you configure all the settings you need for tracing the pipe run (→Pipes – Walk The Run, Walk the run), for applying constraints (→Pipes – ApplyConstraints) and for the export (→Export).

All settings are save in the Plant Model (also refer to →Prepare – Edit).

Step 1 of 4 – Units



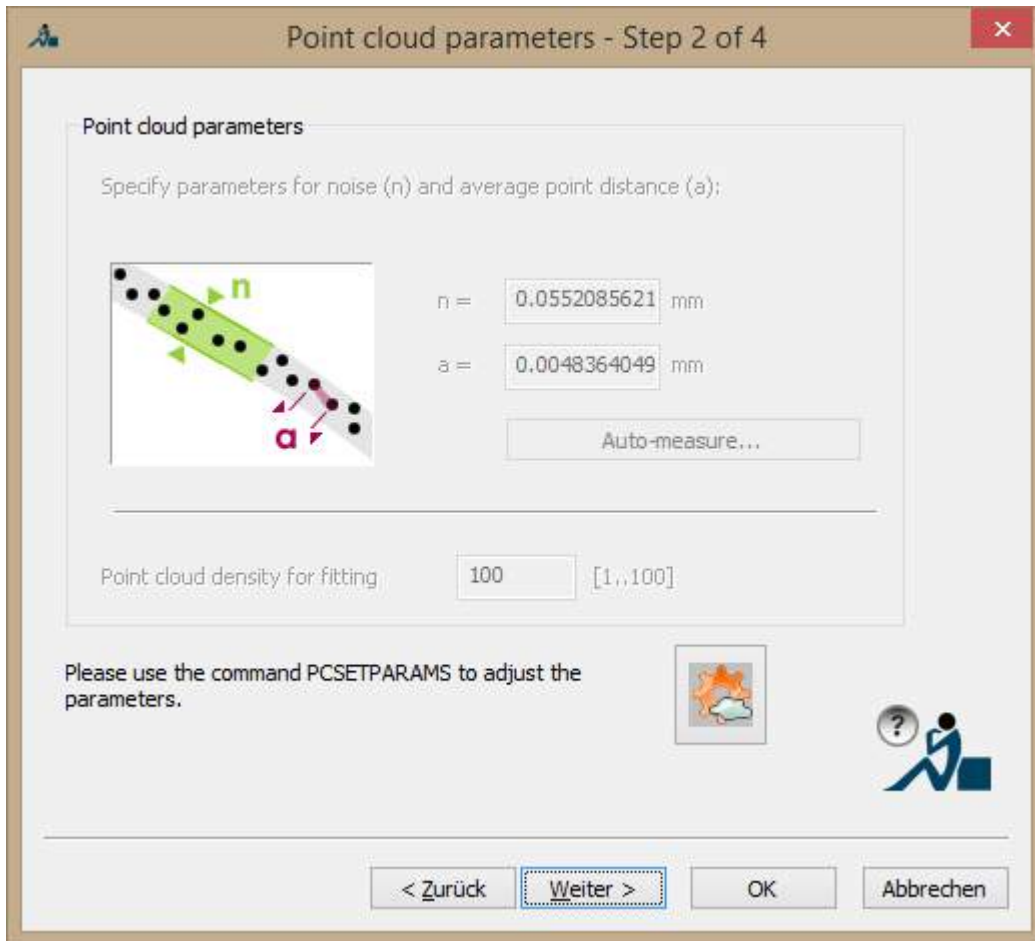
Here you set the units of the length unit of the Plant Model (PSPM). The length unit of the current drawing with its point cloud you can set in KUBT_PC_CONFIG. This separation allows you - without any further scaling - to work with a point cloud that has meters as unit while your plant model is defined in inch.

When you want to create models for AutoCAD Plant 3D, “Override Inch (US Survey)” should be ticked. This setting will only be evaluated, when the drawing’s linear unit is set to **Inch (US Survey)**. If this is so and if “Override Inch (US Survey)” has been


ticked, then the components such as flanges and valves will be drawn in **Inch (International)** instead of Inch (US Survey). The background for this option, is that in the USA it is usual to work in Inch (US Survey), however AutoCAD Plant 3D supports exclusively Inch (International) or millimetres. Although the difference is very small (1 ft (survey) = 1.0000019685039 ft), it leads to errors in Plant 3D Model (Drops).

“Override US Inch” has nothing to do with the plant model linear units rather only with the drawing’s linear units!

Step 2 of 4 – Point cloud parameters



The fitting algorithms used for ‘walking the run’ need an estimation of the point cloud parameters for the noise (n) and the average point distance (a). The current setting

are displayed here. If you want to change them, use the  button. Everything else is described in this command reference under →**Point clouds – Set point cloud parameters**.

Step 3 of 4 – Fitting parameters

Fitting parameters (Expert level) - Step 3 of 4

Manual pipe

Search region size mm

Auto-pipe

Radius deviation threshold %

Direction deviation threshold degrees

Position deviation threshold mm

Quality threshold (amount of points belonging to the pipe) %

Initial length of search region mm

RANSAC (Do not change!)

Outlier percentage %

Target probability %

< Zurück Weiter > OK Abbrechen

Normally these settings do not have to be changed. At the end of this chapter you find a brief description of the individual parameters.

Step 4 of 4 – Layers and tolerance parameters

Layers and tolerance parameters (Expert level) - Step 4 of 4

Layers

Layer for temporary auxiliary: kubitPointSensePlantTemp

Layer for axis polygon: kubitPointSensePlantAxis

Layer for inscriptions: kubitPointSensePlantInscription

Layer for 3D solids (beams): kubitPointSensePlant3DSolid

Tolerance parameters

Angle threshold for pipe extension by point: 2.0 degrees

ApplyConstraints: Tolerance for moved pipes: 200 mm

< Zurück Weiter > OK Abbrechen


These settings normally do not have to be changed. At the end of this chapter you find a brief description of the individual parameters.

Description of the parameters

Name of the parameter	Description
Manual pipe	
Search region size	Defines an initial size for the search region for points around the two points that have been clicked for measuring the pipe

Auto-pipe	
Radius deviation threshold	Defines how much the auto-fit pipe is allowed to differ from the nominal radius.
Direction deviation threshold	Defines how much the auto-fit pipe is allowed to differ from the nominal axis.
Position deviation threshold	Defines how much the auto-fit pipe is allowed to differ from the nominal insertion position.
Quality threshold (amount of points belonging to the pipe)	Defines the minimum percentage of points that belong to the pipe and do lie within the search region for the auto-fit pipe.
Initial length of search region	Defines the initial length of the search region for the automated pipe fitting.
RANSAC	
<i>These parameters must not be changed by the user!</i>	
Layers	
Layer for temporary auxiliary	Name of the layer for temporary auxiliary graphics (e.g. when measuring geometry in →Prepare – Edit)
Layer for axis polygon	Name of the layer for the axis polyline being created during export (see also →As-Built – Settings)
Layer for inscriptions	Name of the layer for the annotations (inscriptions) being created during export (see also →As-Built – Settings).
Layer for 3D solids (beams)	Name of the layer for beams that have been converted to solids.
Tolerance parameters	
Angle threshold for pipe extension by point	Defines the maximum deviation angle between the axis of the extended pipe and the original pipe axis.
ApplyConstraints: Threshold when moving pipe axes	(see also →Pipes – ApplyConstraints). If a pipe axis is moved for a distance greater than specified here, the situation will be marked as a “problematic spot” (if the highlighting of violated constraints has been activated).

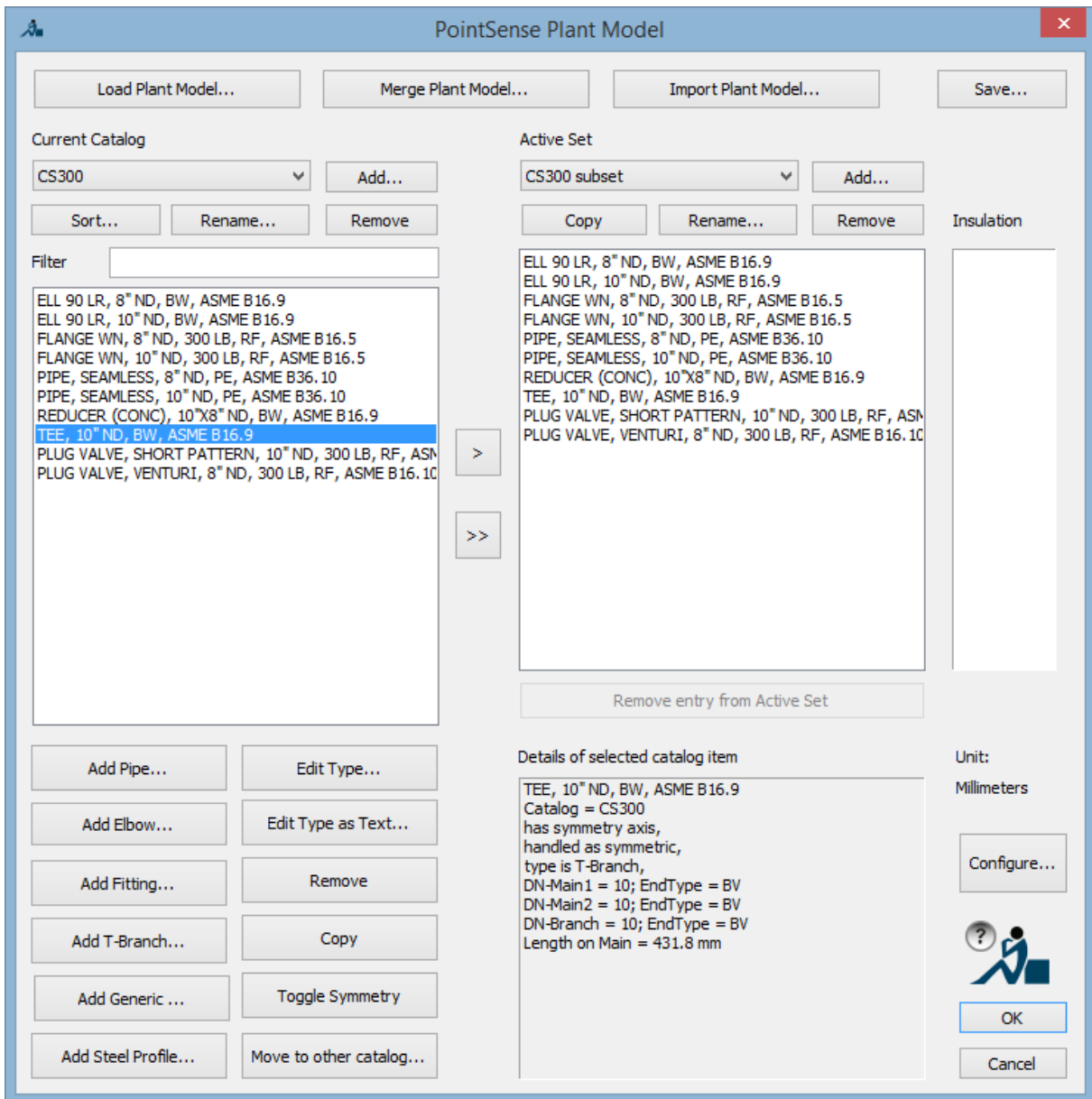
Prepare – Edit Plant Model

	Command Name:	PCEDITSPM
	Ribbon Tab:	As-Built Plant

The command for fitting pipes PCWTR, which is explained under →Pipes – Walk The Run and the command for fitting steel PCFITBEAM (→Beams – Fit beam) require one or more catalogs for the types (standard pipes with nominal radius, standard elbows, beam profiles, etc) that do exist within the plant. These catalogs can be managed and even extended using the **PCEDITSPM** command.

All settings for Walk The Run and especially the catalogs of the plant types are called **Plant Model (PSPM)**. The plant model is stored completely inside the AutoCAD drawing. But you may also export it to a text file and import it into another drawing. This is also done with the PCEDITSPM command.

When running the command following window opens:



The dialog consists of four parts:

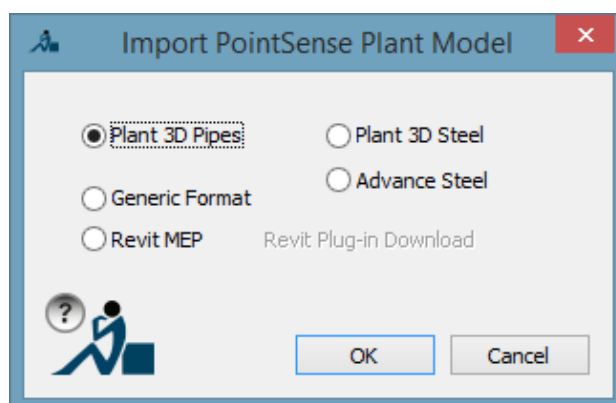
1. **At the top** are the two buttons for loading and saving the Plant Model from the drawing. We prepared some Plant Models for you. You find them in the installation folder of the As-Built in the subdirectory \PSPM:
 - kubtAmericas201x.txt is derived from catalogs with American standards.
 - kubtISO201x.txt is derived from ISO catalogs and it is in millimeters.
 - StructuralPSPM.txt is the catalog for steel beams.
 Try which Plant Model meets your project requirements and adjust it further if needed.

The button [Import Plant Model ...], allows the import of multiple models. The plant model can be imported from AutoCAD Plant 3D catalogues and pipe classes for piping systems and steelwork, from a Revit MEP export file and from special generic file formats.
2. The biggest part are the control elements for the manipulation of the **Current Catalog** on the left. The Plant Model may contain multiple catalogs. A catalog is a number of types like pipes, elbows, t-branches etc, which are combined to a complete pipe run during Walk The Run (PCWTR).
3. In the middle you find the **Active Set**. The Active Set are those plant types which are offered during Walk The Run. The active set can comprise types from different catalogs. While a catalog may contain dozens or hundreds of types the Active Set should not be too large. This way clearness and speed of Walk The Run are improved. This means the active set should only contain types that actually exist or are expected in the current point cloud.
4. On the right you find the list of **Insulations**. These are the insulation sizes (in units of the Plant Model), which are used during the pattern recognition during Walk The Run.

The most important buttons and functions of the dialog are explained here:

[Import Plant Model...]

The Import function offers the possibility to use the types and pipe classes supplied with Plant 3D, or from separately available catalogues, in Plant Model (PSPM). Furthermore a file with pipe classes from Revit MEP can be imported, when such a file was previously exported by As-Built Revit. In addition, it is planned for the compatibility with other applications also to be able to import using a special generic file format.



Notes on importing from Plant 3D:

The location of the catalogues and pipe classes can be determined from the system variable PLANTCONTENTFOLDER. Moreover the project related pipe classes are saved in the project folder. When you intend to later export as Plant 3D objects

(→Pipes and beams - Export), you must definitely select for the import the appropriate pipe class from the project folder. The current drawing (even if it's empty) should be part of the target project.

Due to the numerous formats of the fittings descriptions it is however only possible to make a complete import if the current drawing belongs to a Plant 3D project. Otherwise it is only possible to do a reduced import based on types from the selected catalogue and pipe classes.

The types that were not imported will be listed in a report window.

Analogous to the concept in Plant 3D complex types (e.g. valves with multiple connections or complicated structural shapes) will be principally represented by block definitions. The block definitions used by PLANT MODEL are characterised by the prefix "kubt_".

The profiles that are needed for beam recognition can be imported directly from the AdvanceSteel profile database. Thereby after modelling there is a trouble free transformation of the As-Built beams into AdvanceSteel beams (→Export – Export S. 340).

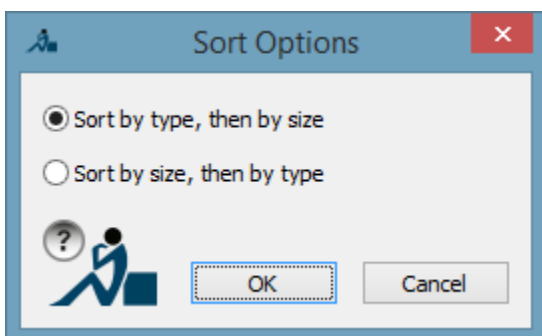
After selecting the import option for AdvanceSteel, the desired profile database for AdvanceSteel must be entered. It is first searched for in the default folder of the database. Should the database have been saved to another location, please navigate to it there.

After selecting the database you can select the desired profile ranges in the following dialogue box. You can select the profile ranges from different types or catalogues. At the moment only standard profiles are supported.

The import is carried out in the As-Built Plant "AdvanceSteel" catalogue.

[Sort...]

This button allows sorting the types of the current catalog.



Sort by type means grouping using generic terms like pipe, t-branch, elbow, inline fitting. Sort by size means sorting by the nominal size (diameter).

Filter

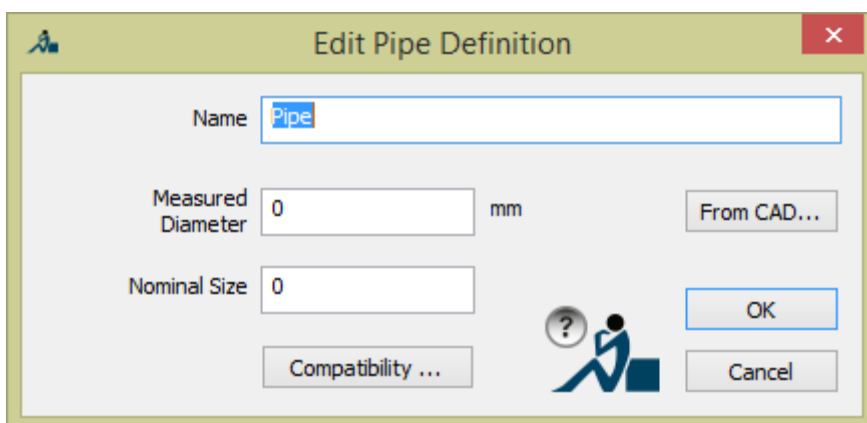
Items in the catalog list can be found via full-text filter. The object which is the first that contains the string entered here will be selected. The list is reduced to all entries that contain the entered string. The filter is not case sensitive.

[Rename...], [Add...], [Remove]

In the section 'Current Catalog' allow the according changes.

[Add Pipe...]

This button allows creating a new pipe type and adds it to the catalog. Following dialog opens:



You may enter the geometrical data as well as the name of the pipe. This name will be displayed during Walk The Run.

The Measured Diameter is the one actually expected as outer diameter in the cloud. By clicking **[From CAD...]** you may determine it directly from the point cloud. The dialog disappears and you are prompted to click two points on the surface of a pipe. The Measured Diameter is displayed in the current unit of the drawing or the point cloud.

The Nominal Size (diameter) usually is a positive integer, which has to correspond exactly with all other types (e.g. elbow) that can be linked with the pipe.

[Add Elbow...]

This button allows creating a new elbow type and adds it to the catalog. Following dialog opens:

The screenshot shows the 'Edit Elbow Definition' dialog box. It features a title bar with a close button (X) and a help icon. The main area is divided into two columns for arm (1) and arm (2). Each column has input fields for 'Center to End Length' (1 mm), 'Nominal Size' (dropdown), 'Matching pipe outside diameter' (1 mm), and 'Port outside diameter' (-1 mm). There are buttons for 'Symmetrical elbow from CAD...' and 'Asymmetrical elbow from CAD...'. A 'Compatibility ...' button is also present. The 'Angle' is set to 90 degrees. At the bottom, there is a 'Block definition for depiction' dropdown, a help icon with a question mark, and 'OK' and 'Cancel' buttons.

You may enter the geometrical data as well as the name.

The Nominal Size of both arms (1) and (2) of the elbow are not entered here but selected from a list. The list consists of all nominal sizes of all previously entered pipes (by [Add Pipe...]). The Diameters are the actually expected pipe diameters inside the point cloud. The fuzziness is a measure which defines how far a pipe grows into an elbow during pattern recognition. If you are not sure what to enter here just keep the default 0.

When clicking the buttons [A/symmetrical elbow from CAD...] the dialog disappears and you can pick the real geometry from the point cloud. For symmetrical elbows the average is calculated from the picked radiuses of both arms (1) and (2).

[Add Fitting...]

This button allows creating a new type of inline fitting and adds it to the catalog. Following dialog opens:

Edit Inline Fitting Definition

Name:

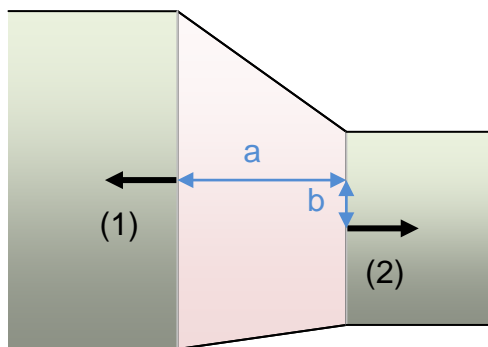
(1) Nominal Size: mm
 Matching pipe outside diameter: mm
 Port outside diameter: mm

(2) Nominal Size: mm
 Matching pipe outside diameter: mm
 Port outside diameter: mm

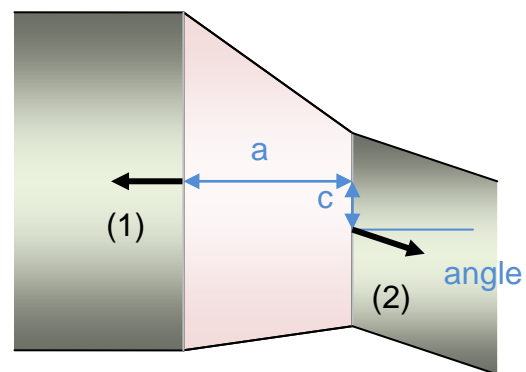
Distance between the planes: mm
 Length (distance in direction (1)): mm

Eccentric (distance perpendicular to direction (1)): mm
 Angle: °

Following figure explains the parameters. Coordinate system is defined by the axis of pipe (1) and the plane determined by both direction vectors:



Side view: Planes containing pipe axis are perpendicular to the paper plane.



Top view: Planes containing pipe axis are parallel to the paper plane.

Legend

a = Length
b = Distance between the planes
c = Eccentric
angle = Angle

For coaxial fittings like flanges, valves or reducers only the *Length* parameter will differ from 0. For regular eccentric reducers *Eccenter* and *Angle* will be 0.

The fields are filled in the same way as when adding an elbow ([Add elbow]).

The parameters can be derived from the point cloud. Usually the command line option [Fitting with parallel axis from CAD...] will be applied. For coaxial fittings one has to set *Distance between planes* to 0.

[Add T-Branch...]

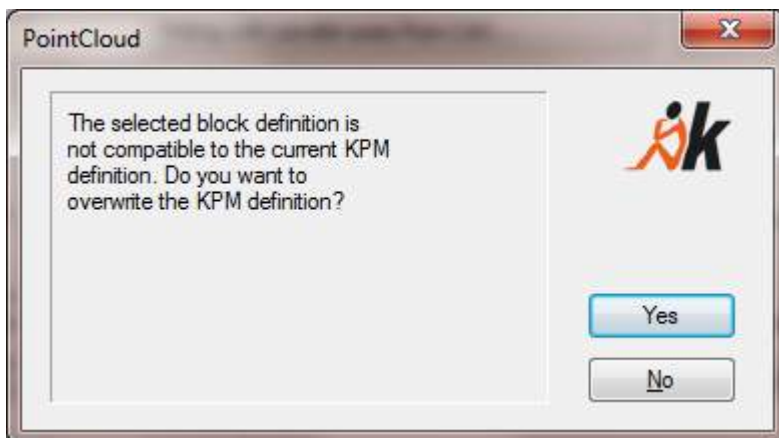
This button allows creating a new type of t-branch and adds it to the catalog. Following dialog opens:

The dialog box is titled "Edit T-Branch Definition". It features a "Name" field with the text "T-Branch" and an "All from CAD..." button. The "Main" section is divided into two columns for ports (1) and (2). Each column includes: "Center To End Length" (1 mm), "Nominal Size" (dropdown), "Matching pipe outside diameter" (1 mm), and "Port outside diameter" (-1 mm). The "Branch" section includes: "Branch Length" (1 mm), "Nominal Size" (dropdown), "Matching pipe outside diameter" (1 mm), "Port outside diameter" (-1 mm), and "Branch Angle" (90 degrees). A "Compatibility ..." button is located to the right of the "Branch Length" field. Below it is a "Block definition for depiction" dropdown menu. At the bottom right, there are "OK" and "Cancel" buttons, along with a help icon.

The fields are filled in the same way as when adding an elbow [Add elbow].

In the edit dialogs for elbow, Tee and fitting you can select a block definition for a more detailed representation. Previously the block definition has to be created or modified using the PCBLOCKCONVERT command.

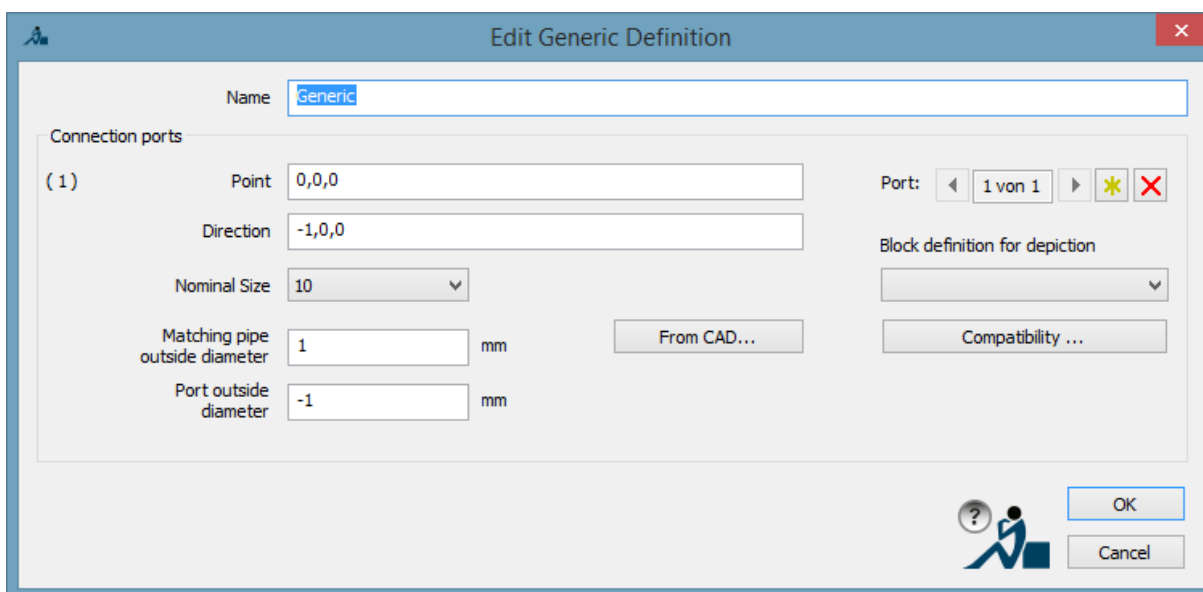
If the geometry of the selected block definition does not fit the PLANT MODEL definition following dialog is shown:




Choosing [Yes] the geometry of the block definition overrides the Plant Model (PSPM) definition. Selecting [No] keeps the PLANT MODEL definition and discards the block.

[New Generic ...]

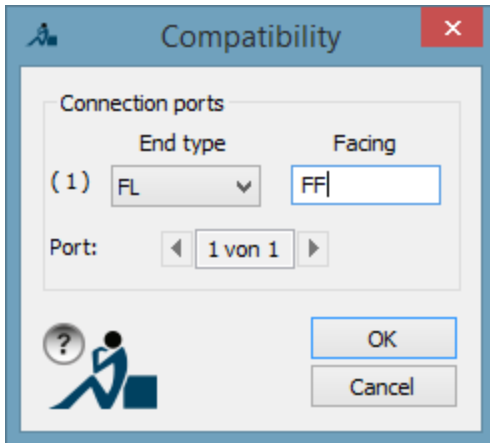
Generic types are types (like crosses) which do not belong to other type by reason of their geometry or function. Following dialog opens:



At first only one connection port is defined. That is valid for caps or blind flanges. More connection ports can add by . Their geometrical definition is described in the fields point and direction. The AutoCAD syntax for the input of 3D coordinates is valid.

Compatibility

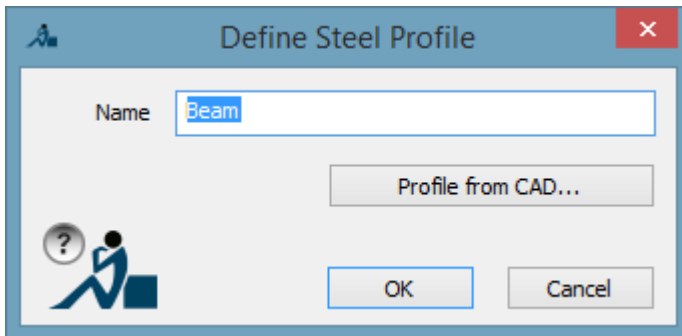
All dialogs for editing of type definitions contain the button [Compatibility ...]. There you can define the parameter for end type and facing of the ports. These parameters are used in WalkTheRun for filtering the matching types. (To be able to do this the box "Strict compatibility check" must be ticked).



A port without end type and facing is compatible to all other ports with the same nominal size.

[Add Steel Profile...]

This button allows the definition of a new type of beam for the later use in PCFITBEAM (→Beams – Fit beam).



With [Profile from CAD...] you are prompted to select a 2D polyline from the drawing which represents the profile of the beam to be created later.



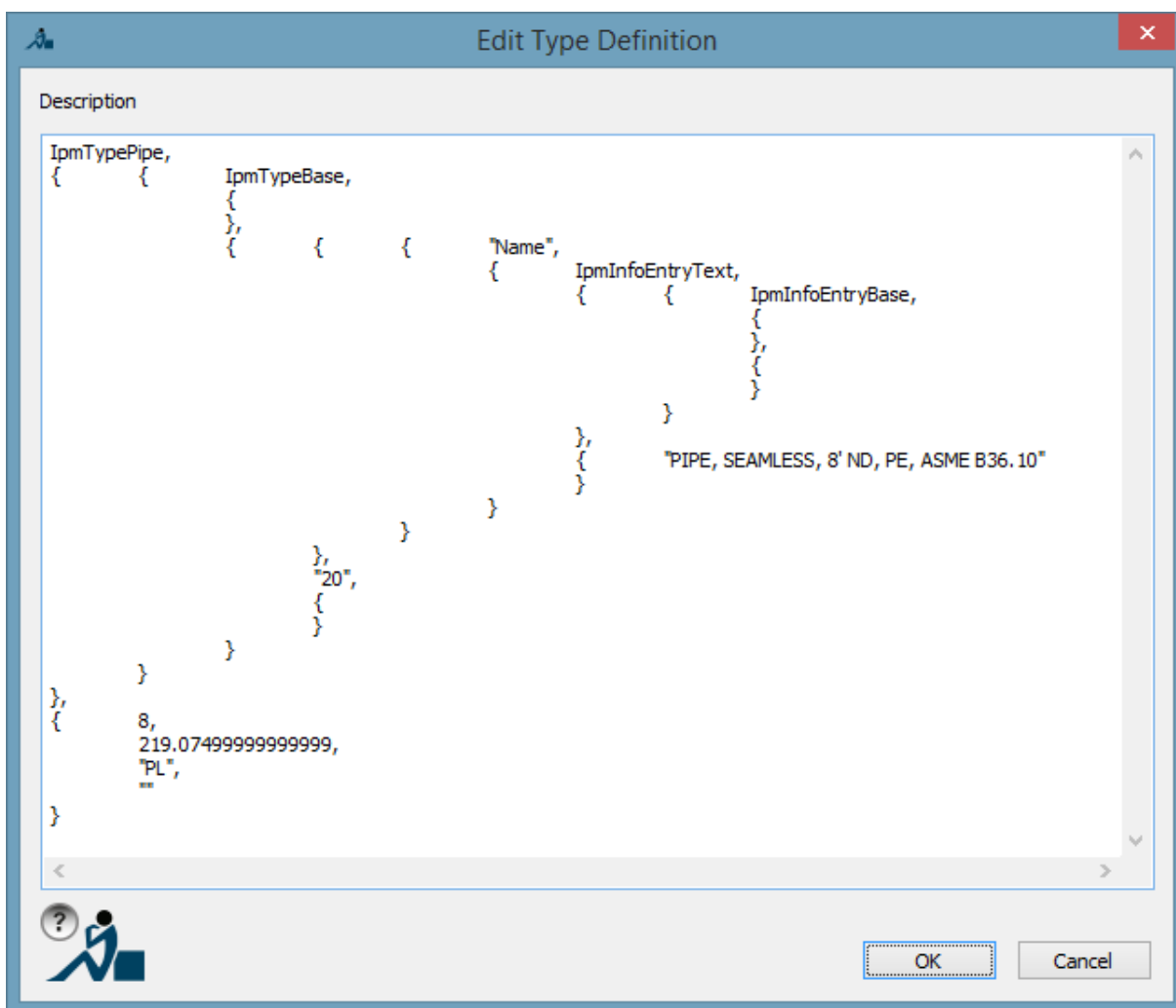
If there are axis points you may also define these, else the balance point of the polygon defines the axis.

[Edit Type...]

The currently highlighted catalog entry is displayed in the according dialog window and its settings can be edited.

[Edit Type as Text ...]

This function will rarely be used. The part of the PSPM file (Plant Model), that belongs to the highlighted type is displayed and can be edited directly. This function should only be used after a training by FARO.



[Toggle Symmetry]

Many plant types have a symmetrical structure. That means their mounting position does not matter during Walk The Run and thus the wizard shall offer only one of both (or more) mounting positions. The status field 'Details of selected catalog item' displays if – due to its geometry – the object can be recognized as symmetric (has symmetry axis). In this case you may decide if the object shall be 'handled as symmetric' during Walk The Run. If you are not sure we recommend to toggle the symmetry so 'handled as symmetric' is displayed as status.

[>]

This button adds the currently selected catalog items to the Active Set. Only the types of the Active Set are used when walking the run (→Pipes – Walk The Run) or when fitting beams (→Beams – Fit beam).

[>>]


This button inserts all of the types in the current catalogue of the "Active selection". Even those, which because of the current filter settings, are not visible. These types

are used in “Walk The Run” (→Pipes – Walk The Run) and PCFITBEAM (→Beams – Fit beam).

[Rename...], [Add...], [Remove]

In the section ‘Active Set’ allow the according changes.

Prepare – Define a block for a plant object

	Command name:	PCBLOCKCONVERT
	Ribbon Tab:	As-Built Plant

For a detailed display of the plant objects you can use AutoCAD block definitions. But before using them you have to define the position and direction of the connection points.

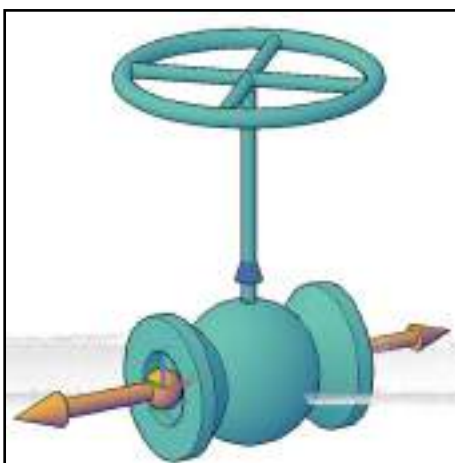
Prior using the PCBLOCKCONVERT command you have to insert a suitable block reference.

After running the command and selecting the block reference you get following options in the command line:

Select an option [Add/Delete/Move/diRection/eXit]:

- Add:** Define a new connection point. Choose the first point for position and the second one for the direction. The new connection point is symbolized by an arrow.
- Delete:** Select the according arrow to delete the connection point.
- Move:** Define the new position of the selected connection point.
- diRection:** Change the direction of the selected connection point.
- eXit:** Exit the command by this option to save the changes.

If the defined connection points are collinear you get a further option <rOtation>. With this option you can define the general rotation of the component and define it's z-axis. The defined rotation is symbolized by a blue arrow.




Please note the changes will only be saved by using the <eXit> option.

After that you can erase the block reference.

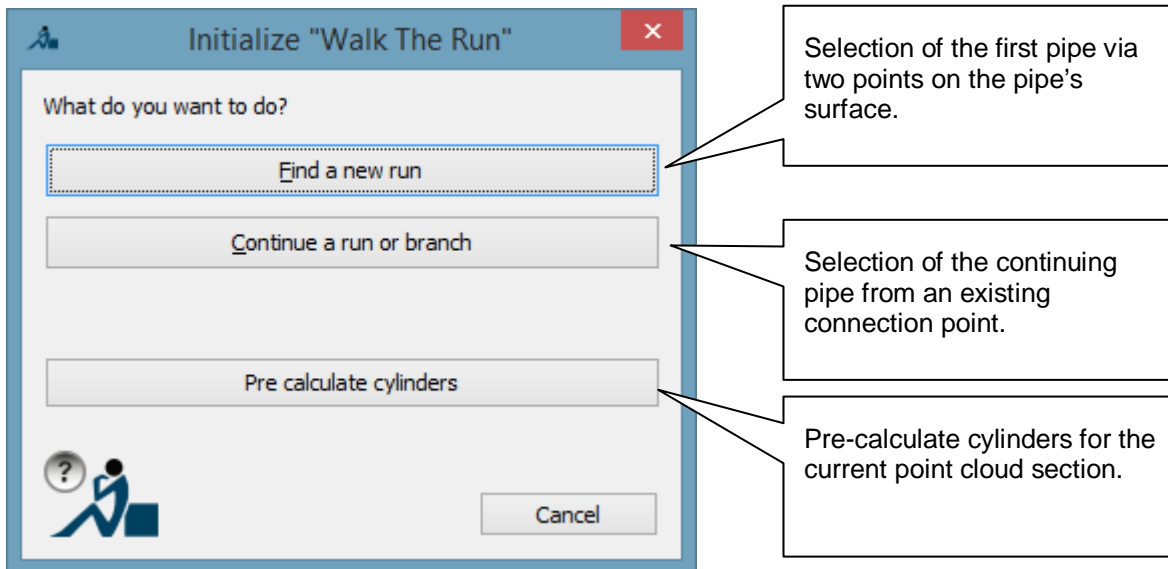
To use the block based representation for a plant object, you have to add it to the PLANT MODEL. To do so run the command PCEDITPSM (→Prepare – Edit Plant Model)

Pipes – Walk The Run

	Command Name:	PCWTR
	Ribbon Tab:	As-Built Plant

This command allows semi-automatic tracing and recognizing of a pipe run including **existing** components (Walk The Run - WTR).

Following start window opens:



Find a new run:

The pipe is recognized after you clicked two points on the pipe's surface. After the cylinder fit was successful the suitable pipe type for this diameter is searched from the PLANT MODEL.

Select two points on a pipe

First point or [change Insulation/select Object/parameters]: i

The ends of the pipe are not limited to the extends of the clicked points. The pipe is extended as far as possible. Changes of direction or parts that tangent the pipe stop the pipe extension.

If the algorithm finds more than one suitable pipe type, it offers them for selection.

When the first picked point lies on the surface of a previously computed cylinder, this will be displayed and can be immediately used, by either pressing ENTER (or option "b"). The pipe run tracer will then be continued from the picked point in the direction of the farthest pipe end, which will also be indicated by an arrow.

Should the previously computed cylinder not be used for the first pipe, then the pipe recognition will be started by the picking of the second point.

Besides the input of the pipe points, you may also change the insulation that shall be considered. To do so simply use the command line option 'i'.

With the “select Object” option, a Plant pipe or even a Kubit cylinder can be used as the start object.

The “paraMeters” option opens the settings dialogue box for the point cloud parameters.

Continue a run or branch:

To continue a pipe run you may also select the free end of any pipe run component. If the selected component has exactly one free connection point the continuing pipe is searched from that point. If however there are multiple free connection points you have to define where to continue by clicking close to the desired connection point. Please use the object snap ‘node’.

Select a plant object or [changeInsulation]:

Specify point close to the desired connection point by Osnap or [changeInsulation]:

After initializing Walk The Run the control center opens. Here you control all further action and get feedback on the current results. The basic workflow is described in the following.

Pattern recognition will only be carried out for pipes (cylinders). Here there are three methods which will be used:

1. Local temporary computation of all cylinders around the current position of the pipe run tracer
2. Use of the pre-computed cylinder or the already existing pipes
3. Selection of a specific pipe type as the continuation using the 2 point method

The button “Pre-compute cylinders” starts the command →Pipes – WTR – Pre-compute Cylinders. All further information is to be found in the appropriate section.

If a pipe was detected by one of these methods the algorithm automatically starts searching for a suitable continuation. It checks for all compliant fittings and components (nominal size) of the Active Set if - when inserting them - a continuation can be found in the point cloud at their free pipe end.

Detected components with valid continuations are immediately displayed in the control center and sorted by their quality. A color scheme provides fast overview. And you get a preview in the drawing for the selected list entry.

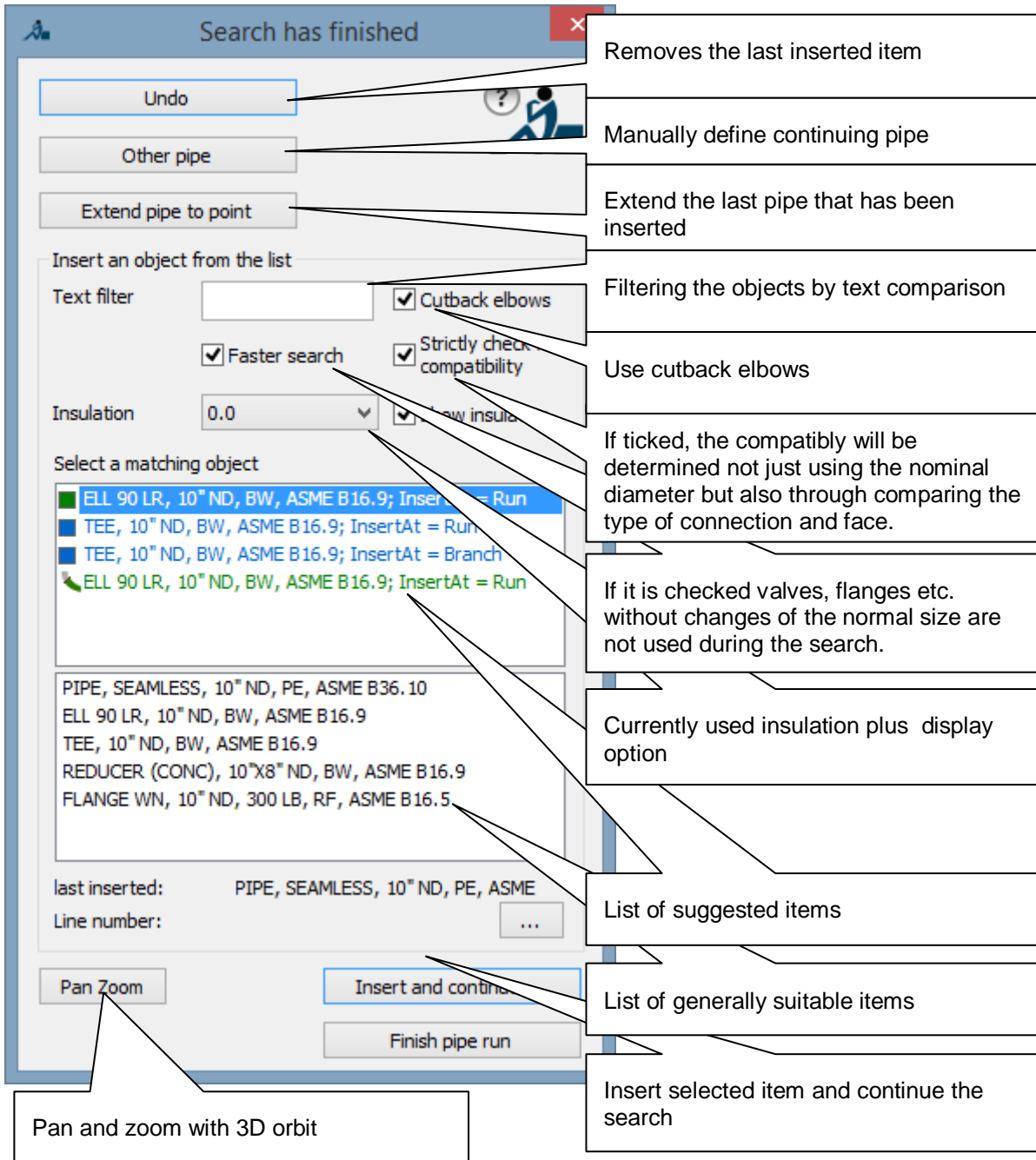
■ Elbows

■ Tees

■ Inline Fittings

If the connecting component was selected and inserted the continuing pipe is automatically extended as far as possible. The new pipe end is the new start point for the next search for suitable components.

For supporting the pipe tracing the continuing pipe can also be defined manually by clicking two points on its surface ([Other pipe]).



[Undo] removes the last component inserted and pans the current view to the new pipe end.

The definition of **another pipe** speeds up the search or allows the continuation if the automatic search fails to find the next pipe. At components with more than one connection point (like t-branches) you may also select the branching pipe. However, the run will be continued in the main direction.

When the continuation pipe has the same diameter, then there will be only a few proposed fittings that can be used. It is different when the diameter varies and/or it is also possible that several other pipe types would fit.

In the list of proposals there are also components which could fit when used together with the use of a (concentric) reducer. So by changing the nominal size from DN 200 to DN 250 it could be that both an elbow of DN 200 nominal size with a following extension and also one of DN 250 with a leading reducer would be compatible. These combinations are marked in the list with an icon.

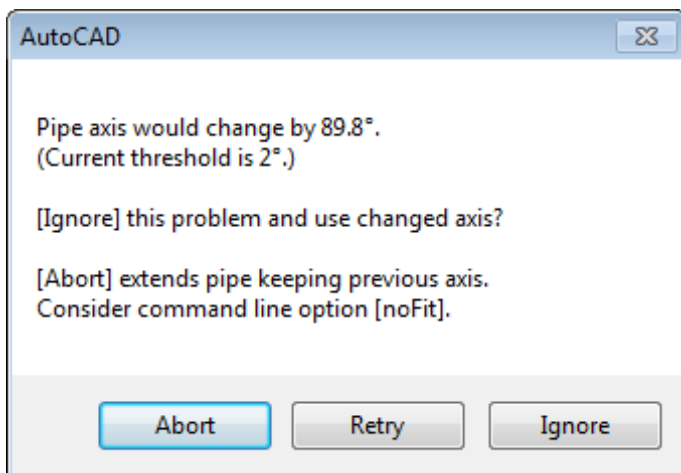
To be able to use this nominal size adaptation the appropriate reducer components must be available in the current selection.

In addition to these variants of the nominal size adaptations, components with the same nominal size will always be available, that means when the other pipe, for example, is DN 300, even though the current pipe run end has a nominal size of only DN 150, then elbows or tees with a nominal size of DN 150 will also be listed.

If Walk The Run does not detect the correct length of a pipe you can extend (or shorten) it manually ([**Extend pipe to point**]). The extension can be made by keeping the existing axis or by fitting the pipe anew (getting a more accurate alignment). Additionally you have to option to let the pipe grow.

Select point up to which to extend the pipe or [noGrow/noFit]:

If the extension results in an axis deviation that is larger than the tolerance defined with PCCFG (→Prepare – Settings), following message opens:



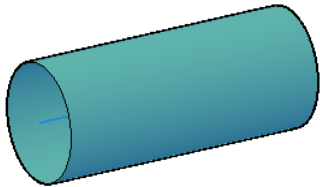
Filtering the types can speed up the search, because only the types which correspond to the entered text will be considered. And the search result will be more clearly.

The tick in "**Faster search**" does not take into account during the search any fittings which have only collinear connection points and a consistent nominal diameter, such as valves and flanges etc. Accordingly, such fittings are no longer to be found in the coloured list of suggested fittings, but rather they must always be inserted manually from the black part of the list.

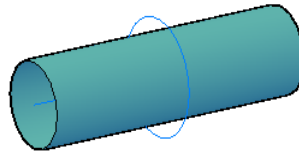
The tick in "**Strictly check for compatibility**" further restricts the fittings used in the search, in that additional information to the type of connection and the face will also be included. Thereby after a flange only flanged-fittings are available for selection.

The connection types and the faces are coded using abbreviations, according to the origin of their data. By default the abbreviations used in the compatibility matrix in Autodesk Plant 3D are also used here. Should this matrix not be valid because of other abbreviations or compatibility requirements, then it can be customised by editing the text file `PlantEndTypes.txt` to be found in the installation folder of As-Built. The changes will then apply to all newly created PLANT MODEL.

The **insulation** thickness can be changed during the search. The search is then repeated with the changed pipe diameter. The option for the display of the insulation affects the visualization of a plant object like shown in following sketch:



Show insulation

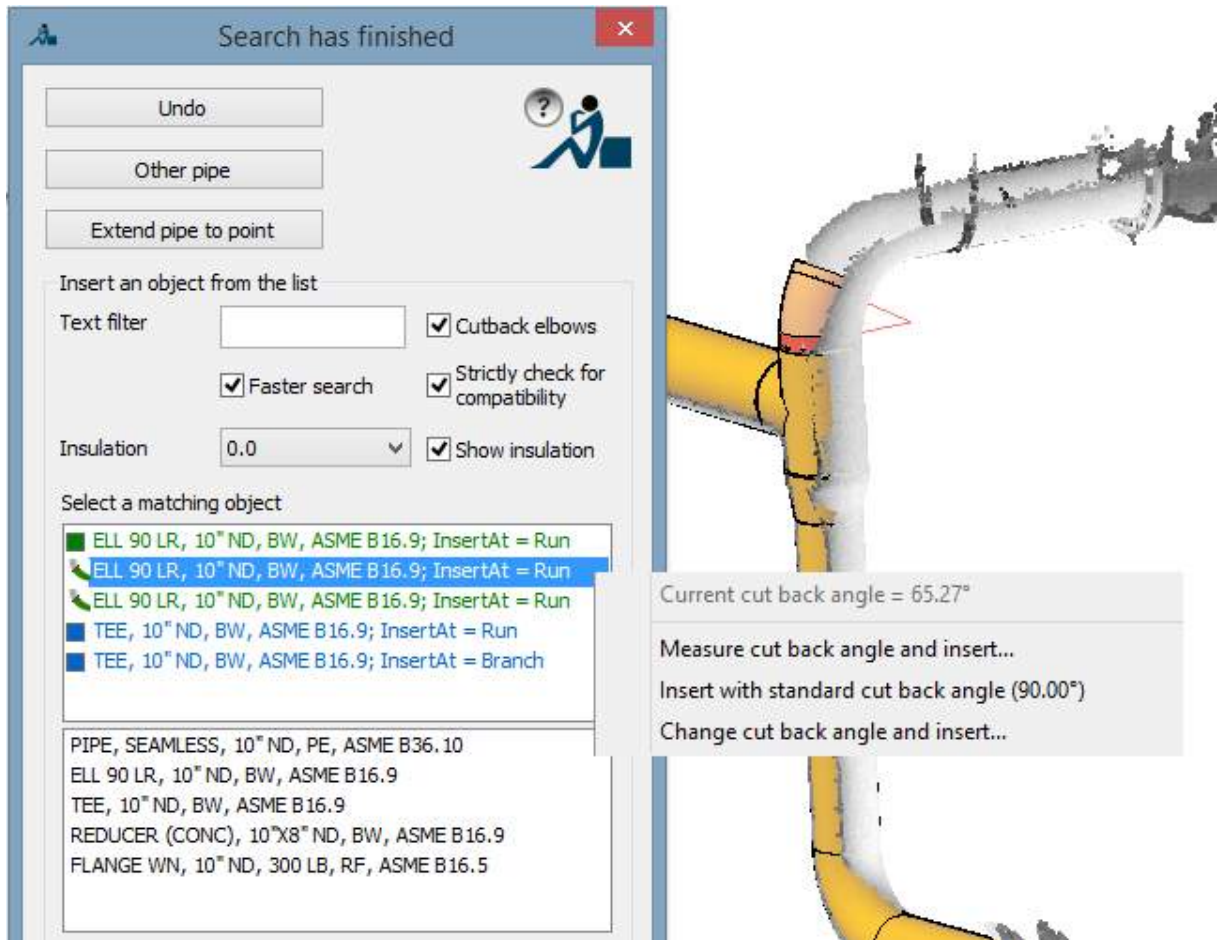


Do not show insulation (the insulation is indicated by a circle)

The list of suitable components shows in black all components that would generally fit at the currently free connection point. As the search is in progress colored items are automatically added in the upper list, that represent those items for which a continuation was found in the point cloud.

If the correct component is present and highlighted you can confirm the selection by double clicking it or by pressing ENTER or clicking [Insert and continue ...] without needing to wait until the automatic search is finished.

If you have activated the **cutback elbow mode**, then cutback elbows can also appear in the list, as well as cut off pipe bends. These are also marked by a separate symbol:



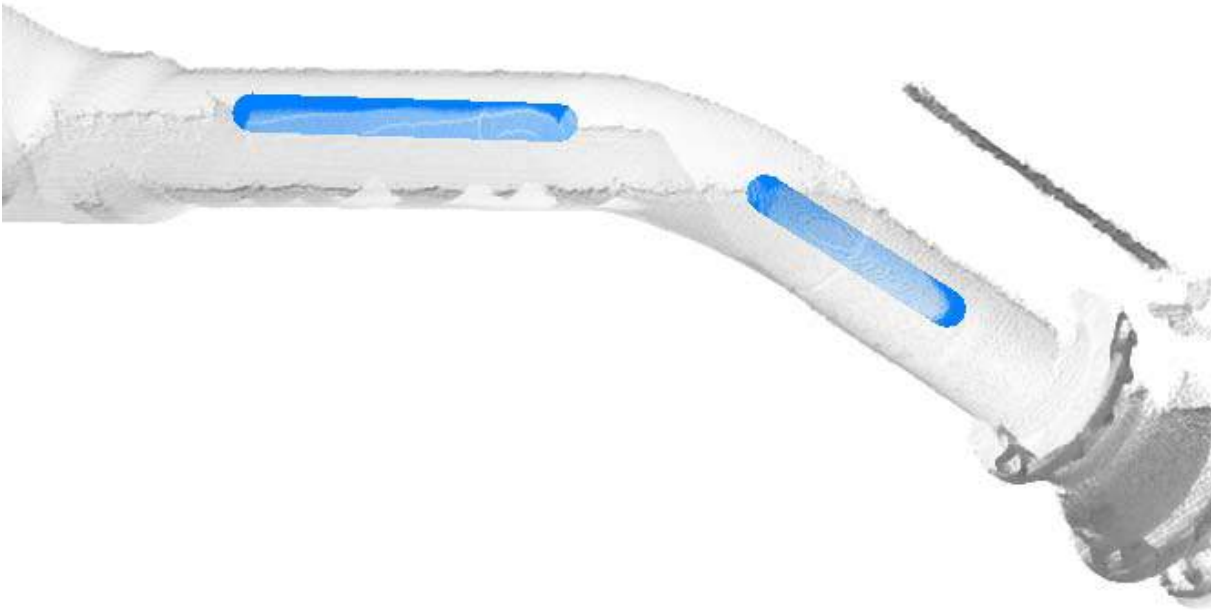
You can see what the current cutback angle is by right clicking this entry. Furthermore, you have available options to manually change or measure this angle:

Measure cutback angle and insert

There are two options to measure the cutback angle:

Choose measurement option [Angle/Circle]:

With the option **Angle** you will be prompted for two tangential lines on the legs of the cutback elbow:



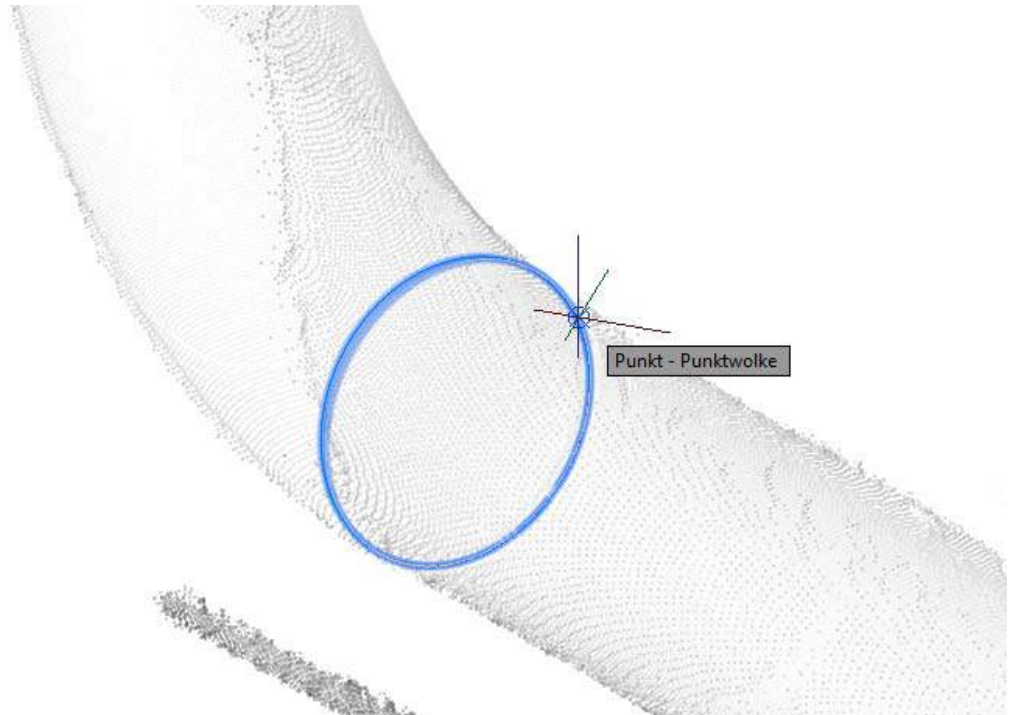
First point for first tangent:

Second point for first tangent:

First point for second tangent:

Second point for second tangent:

The option **Circle** on the other hand will prompt you for points on the connecting circle of the cutback elbow. In an ideal situation you will be able to identify these connecting circles (circular cross sections) as welded joints in the point cloud.



Circle 1 - Select 1. point on circle:

Circle 1 - Select 2. point on circle or [Undo]:

Circle 1 - Select 3. point on circle or [Undo]:

Circle 1 - Select 4. point on circle or [Undo/Finish] <Finish>:

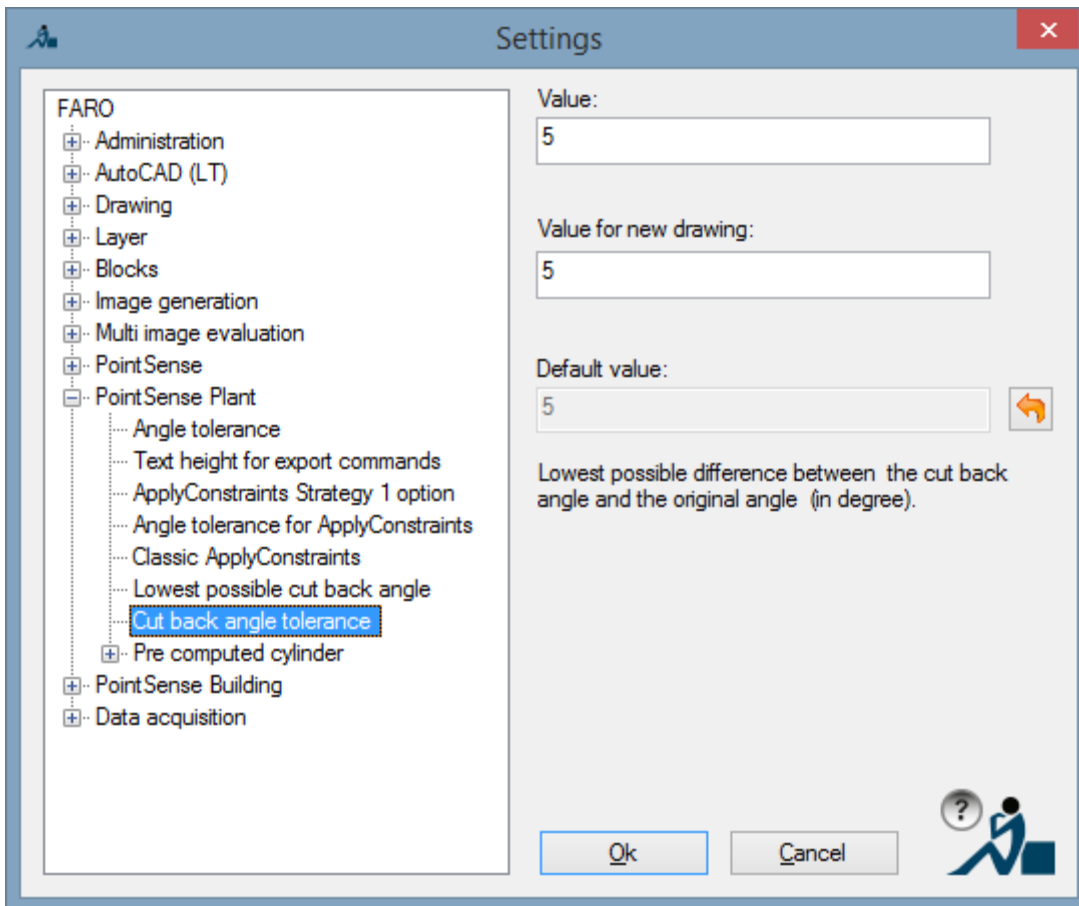
Circle 2 - Select 1. point on circle:

Circle 2 - Select 2. point on circle or [Undo]:

Circle 2 - Select 3. point on circle or [Undo]:

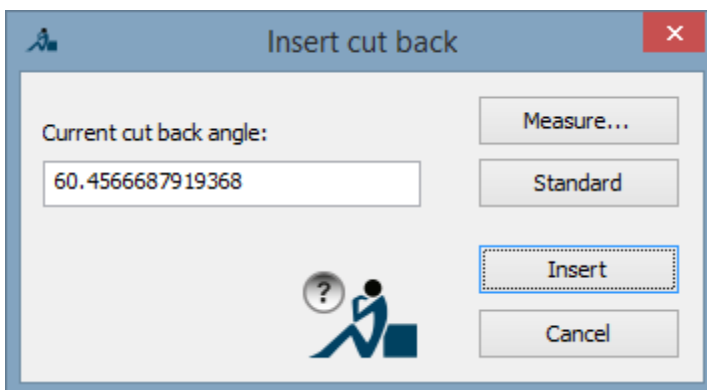
Circle 2 - Select 4. point on circle or [Undo/Finish] <Finish>:

If the new cutback angle derived from both options lies within the tolerance, then the cutback elbow will be inserted and WTR (**W**alk **T**he **R**un) can be continued. You can set the angular tolerance in the As-Built Settings:



Cutback elbows can be inserted with their original angle (from the component definition), (Option Insert with standard cutback angle).

As the third option there is **Change cutback angle and insert:**



Here you can manually enter the cutback angle (decimal degrees) or determine it by measurement (for this see the option **Measure cutback angle and insert**). The cutback elbow is first inserted after you have clicked the button **Insert**.

Except inline fittings all components are automatically placed according to the continuation found. Due to their constant axis inline fittings have to be placed manually on the pipe axis.

Specify insertion point at the start of plant object or [Toggle]:

The option [Toggle] swaps the insertion point from the start to the end or any appropriately relevant intermediate points of the fitting (e.g. rear side of a flange) and conversely, looking in the direction of the run.

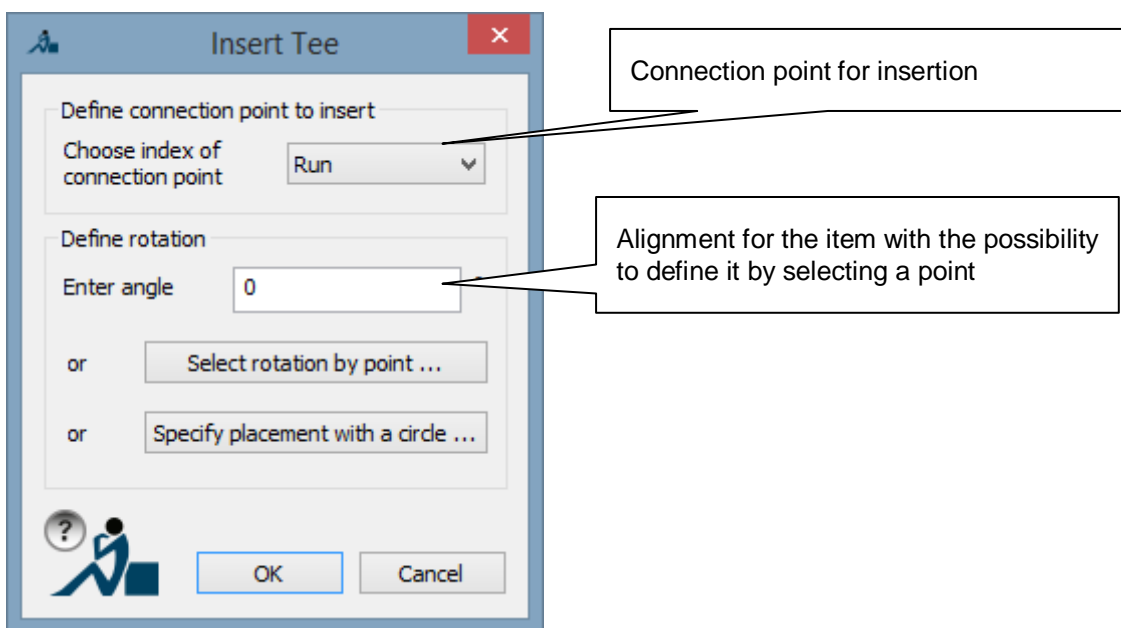


The current insertion point is marked with a red ring (This ring is clearly visible when in AutoCAD *show/hide line widths* is activated).

The finally chosen insertion point will then be orthogonally projected onto the centreline, to precisely define the insertion point of the fitting.

If the automatic search was finished and the suitable component is still not shown as colored entry, you may select a component from the black part of the list and insert it manually. When inserting manually you have to position the component directly at the free end of the run. Only inline fittings can automatically adjust the right position along the axis.

Elbows and t-branches can be rotated into the desired position. If moreover the component is not symmetrical you can also select the relevant connection point.



When defining the alignment by a point, a dynamic preview is displayed. Depending on your direction of view you may

Select rotation of object by point:

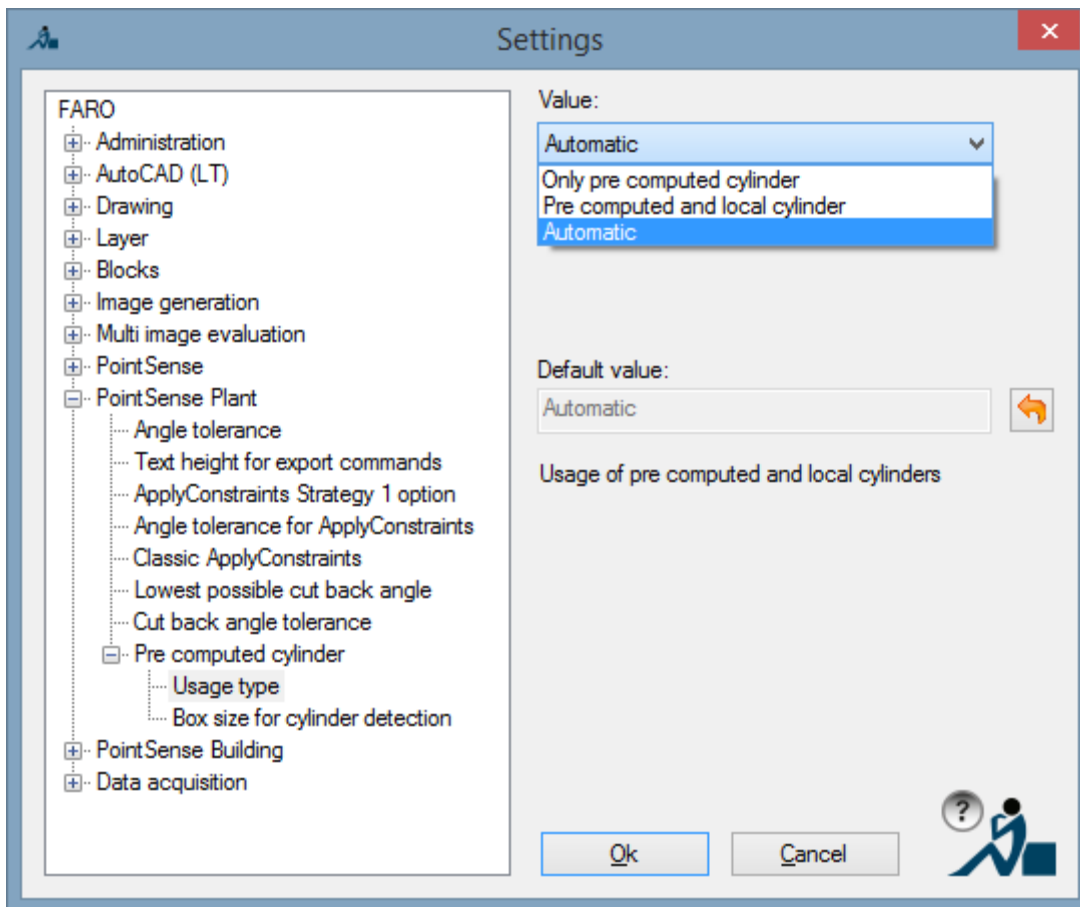
or you

Select rotation of object by direction <Cancel>:

If - from your current direction of view - it is possible to define the alignment by one single point, then you simply press ENTER after having chosen the (first) point and you get back into the dialog. Otherwise click a second point to define the direction.

The second option allows the alignment of the rotatable tie-in point around the centre point of a self defined circle. The definition is done by picking at least three points on the circumference. By selecting more than three points the best possible circle will be used.

As described above, inline fittings can be positioned anywhere on the pipe axis. A dialog window for manual insertion is only shown if the component is not symmetric and thus one has to define the relevant connection point.

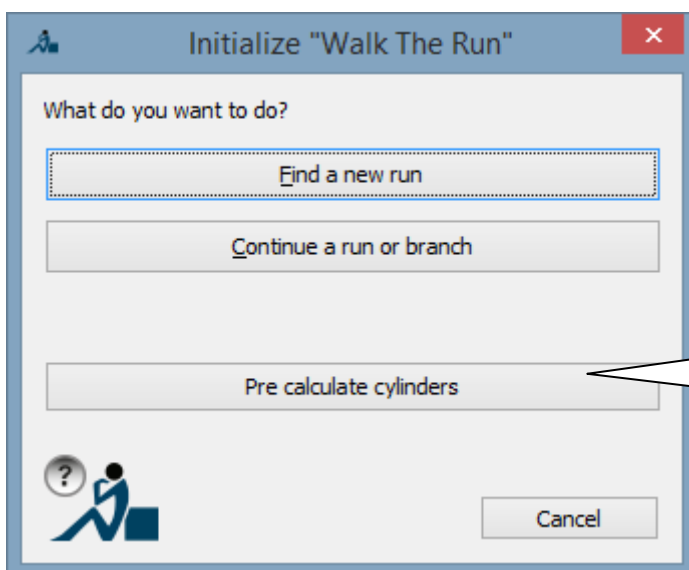


Pipes – WTR – Pre-compute Cylinders

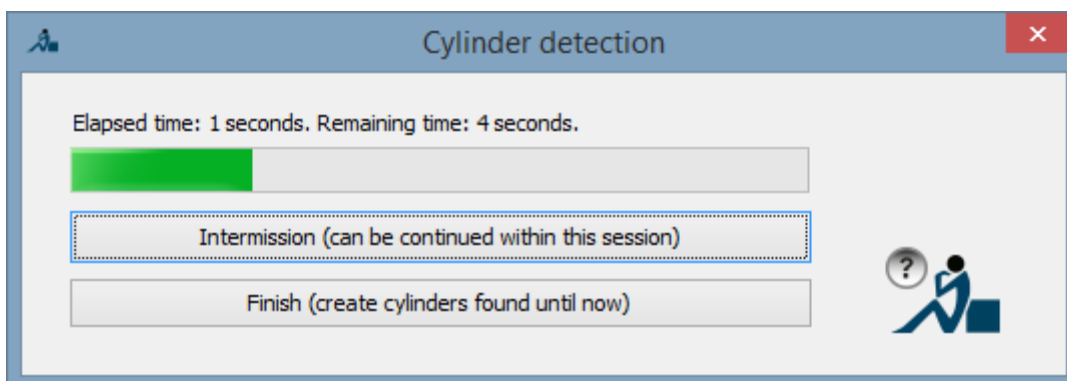
Command Name:	PCPC
Ribbon Tab:	As-Built Plant

“Walk The Run” works significantly faster when the pipes are previously known. For this purpose an algorithm has been developed, with the help of a RANSAC (*random sample consensus*) method, which evaluates the current selected part of a point cloud and attempts to detect all of the cylinders.

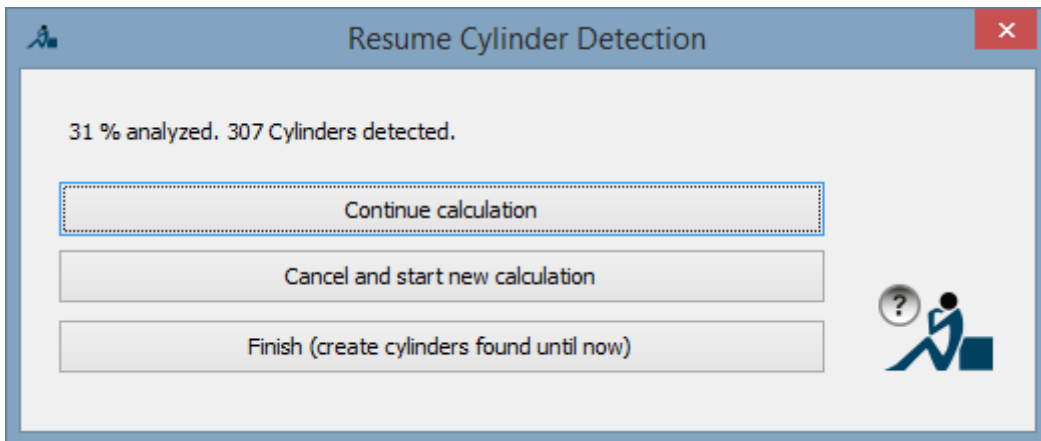
This command can be started by either clicking the “Pre-compute cylinders” button in the WTR dialogue box or with the command PCPC.



When the user starts the cylinder detection the following dialogue box is displayed:

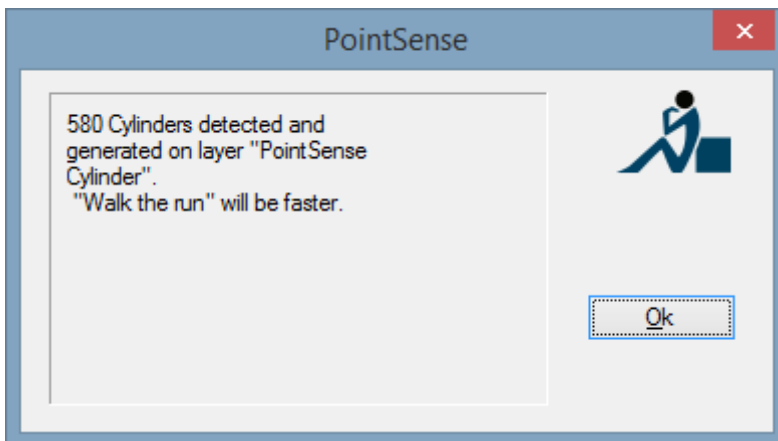


A progress bar shows the actual state of the calculation. The already elapsed time is shown together with the remaining time, which is determined based on the calculation up to now. The user can interrupt the command by clicking the “Intermission” button without losing the cylinders that have already been computed. An interrupted calculation can be resumed within the same session by starting the command again, provided that the selected part of a point cloud has not been changed. Moreover the user has the possibility to completely end the calculation and insert the already detected cylinders as Kubit cylinders into the drawing. If an interrupted calculation is resumed within the same session and starts the pre-calculation of cylinders again, the following window is displayed:



Here there is the possibility to continue the calculation from the point where it was previously interrupted, to start a new calculation or to insert the cylinders that have already been detected into the drawing.

If the pre-calculation of cylinders is allowed to run to the end,



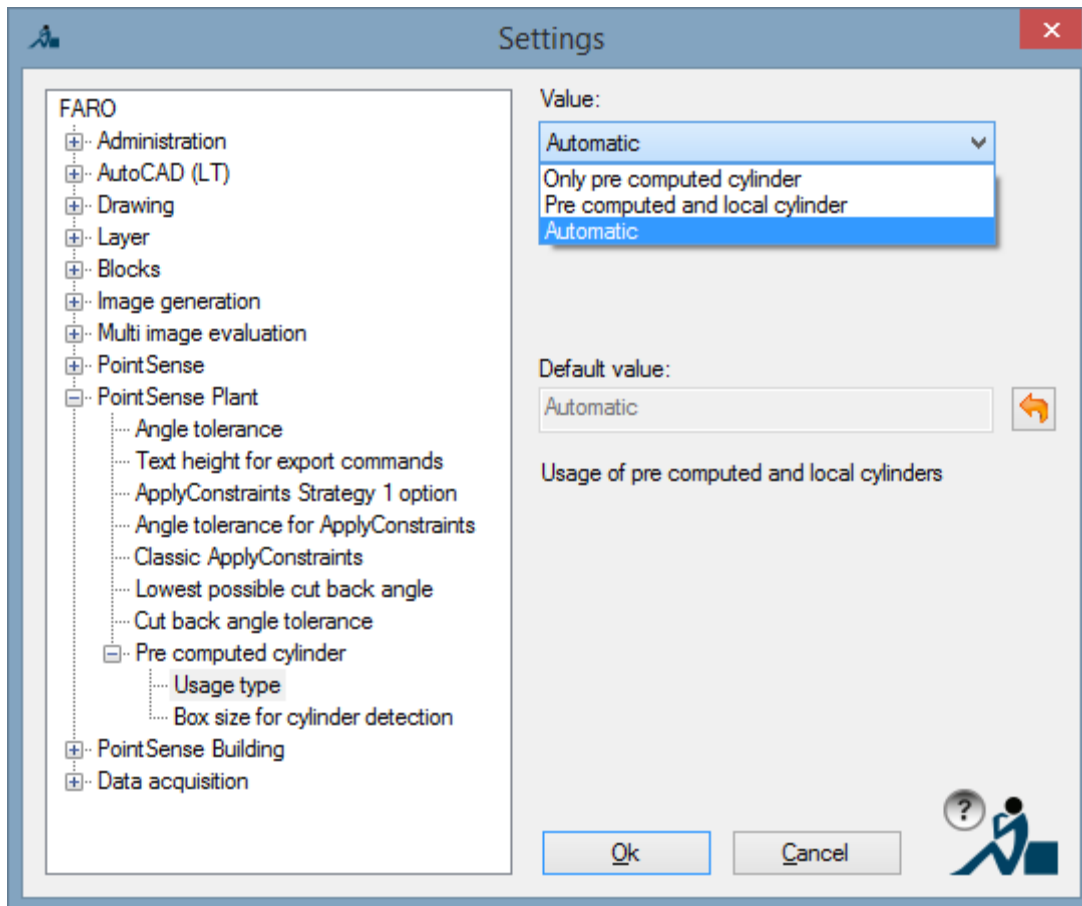
a message box is displayed, that informs the user how many cylinders have been generated and on which layer they have been placed.

Under Settings > Plant > Pre-compute Cylinders there are various parameters that can be set.

Layer > Pre-computed cylinders:

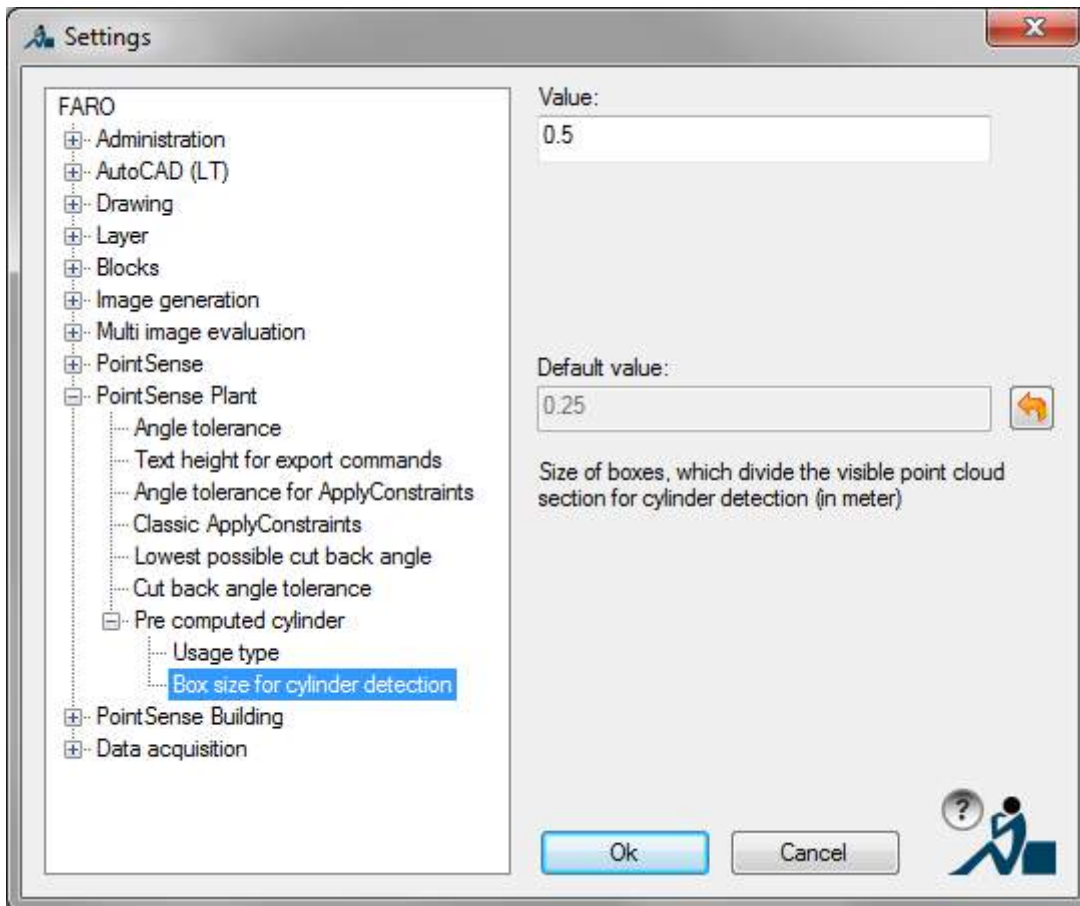
Here enter the name of the layer on which the pre-computed cylinders should be created.

Plant > Pre-computed cylinders > Type of use:



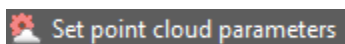
This setting defines that local cylinders can use the pre-computed cylinders that are currently available in the drawing. Local cylinders are temporarily determined during the pipe run tracer in the vicinity of the current position. This computation takes only a few seconds, but can also be switched off with the option “Only pre-computed cylinders”. With the option “Automatic” a sequential creation can be used from the suggestions for the continuation of the run. When the use of the pre-computed cylinders brings no suggestions, it is then that the computation of the local cylinders is started.

Plant > Pre-computed cylinder > Box size for cylinder detection:

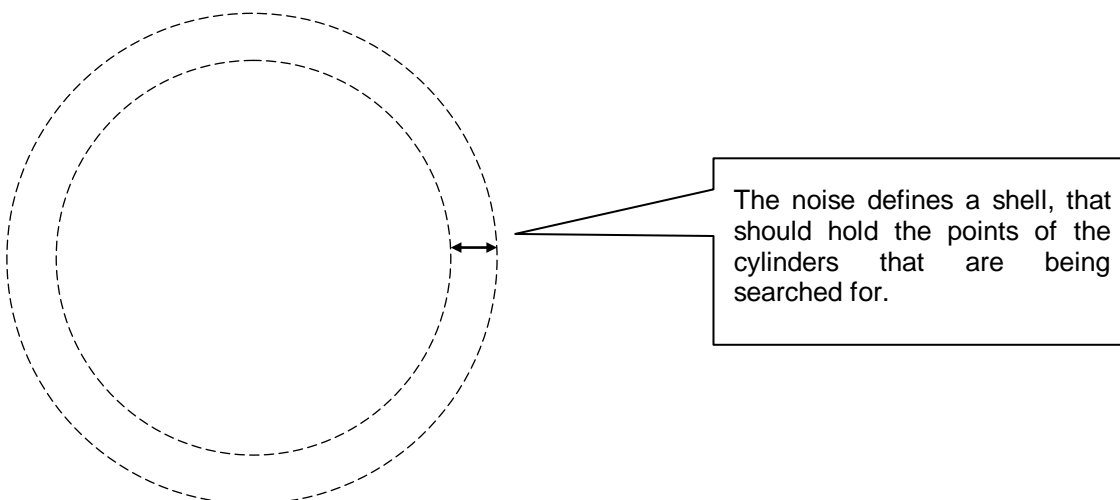


This is the size of the RANSAC boxes that the part of a point cloud will be divided into. Here the user must be aware that in an ideal situation a box is so big that it is only possible for it to hold one cylinder. When cylinders in a plant are computed, that have only large pipe diameters, it is advisable to adjust the size of the boxes accordingly. Thereby fewer boxes will be generated and the computation will be by that accelerated.


A further important parameter for the pre-calculation of the cylinders is the noise value, which can be set under:



A shell model of the cylinder is generated with the help of the noise. The noise defines a shell, that should hold the points of the cylinders that are being searched for.



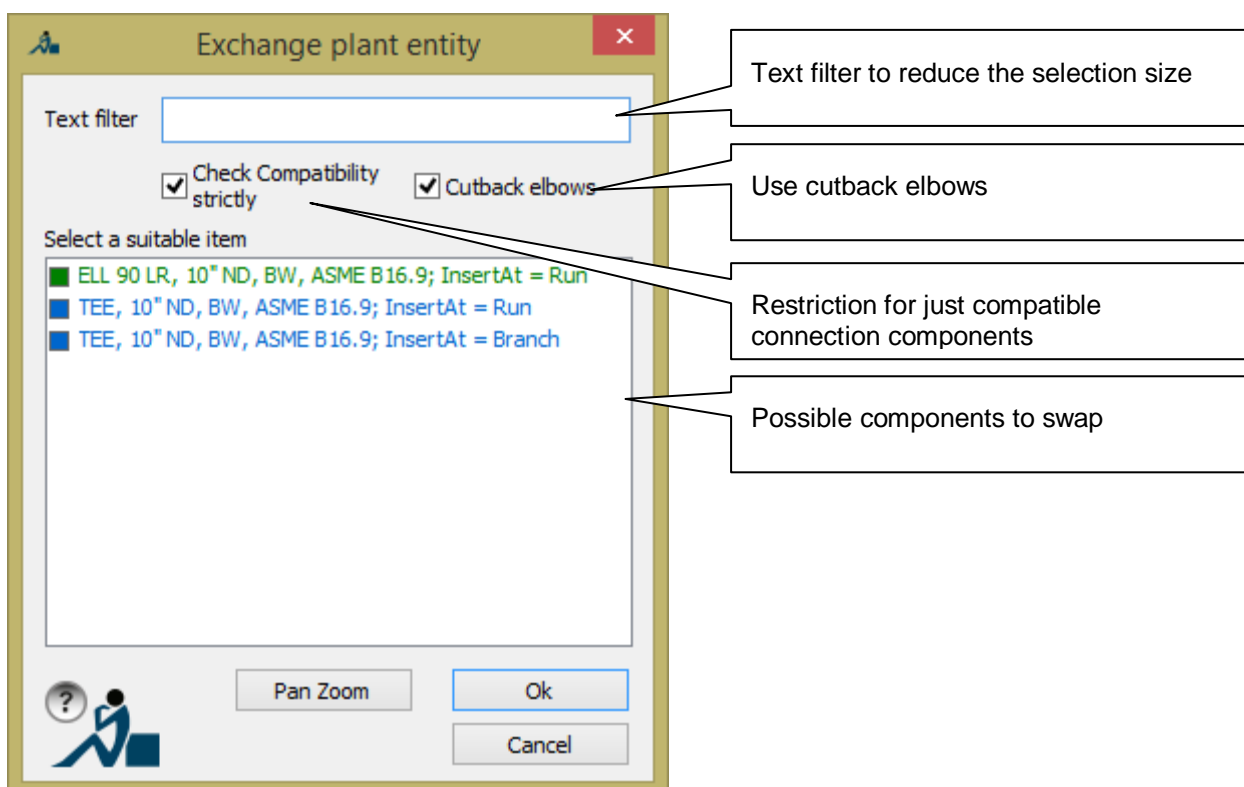
Pipes – Exchange a plant entity

	Command Name:	PCEXCHANGE
	Ribbon Tab:	As-Built Plant

This command allows the exchange of an existing plant entity.

First select the plant entity for exchanging.


In the following dialog, you get all possible types for exchanging:



With [Ok] you can insert the new entity.

When using this command, the component to be swapped must have at least one connection to another component.

Pipes – Set link

	Command Name:	PCSL
	Ribbon Tab:	As-Built Plant

This command allows to explicitly connect plant objects.

You can choose between two methods for selecting the relevant connection points:

- Use the shortest distance between connection points to connect
- Connect the selected points (nearest free connection point to clicked point during object selection)

To toggle the method use the option in the command line:


Current Setting: Use the shortest distance between connection points to connect

Select a plant entity or [setting Toggle] <setting Toggle>:

If the plant objects have not free connection points, you get a message in the command line:

No connection possible because at least one of the selected connection points is already in use.


Pipes – Remove link

	Command Name:	PCRL
	Ribbon Tab:	As-Built Plant

This command allows to explicitly disconnect plant objects.

Only the links between the selected objects are removed.

Pipes – Set new links

	Command Name:	PCRELINK
	Ribbon Tab:	As-Built Plant

This command allows the explicit connection of a large number of Plant objects in a single step.

With connections it is shortest connection between two tie in points that is valid. Thereby, if applicable, any already existing connections will be removed.

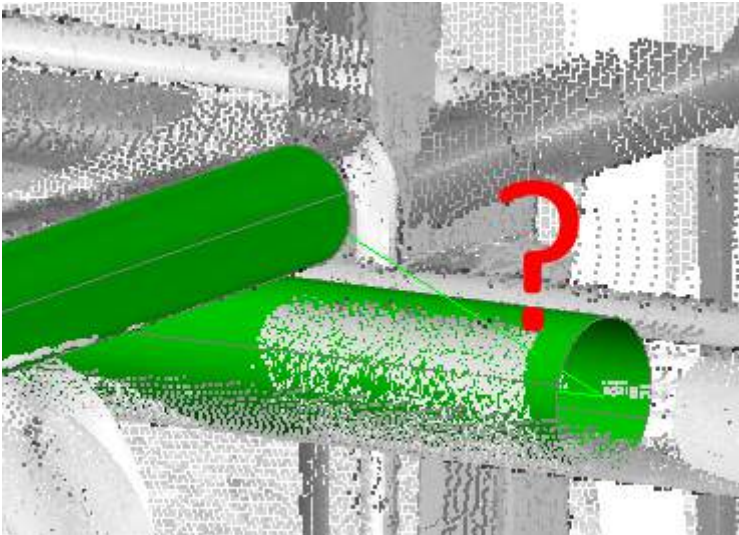
After starting the command all of the objects are to be selected that are to be connected.

Select entities

Thereafter the maximum distance, in drawing units, must be entered, within which two tie in points should be connected.


Specify distance or <100>:

This maximum distance should not be too large, otherwise, for example, the free ends of adjacent pipes could also be connected.



It is not checked at all to see how compatible the connections are to each another.

Pipes – ApplyConstraints

	Command Name:	PCAC
	Ribbon Tab:	As-Built Plant

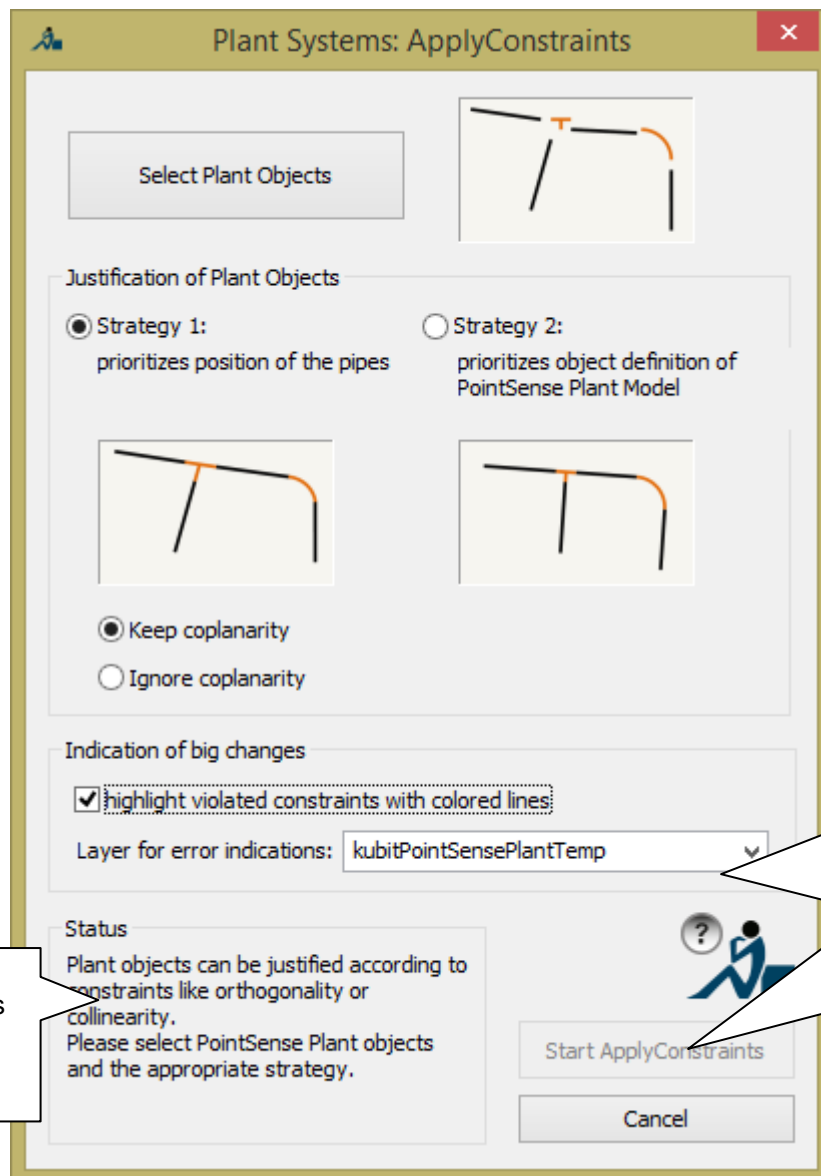
This command is applied at the very end after having used the command → Pipes – Walk The Run (PCWTR) to Walk The Run.

All objects of a pipe run or several pipe runs have been created and do fit – at least locally – optimal to the point cloud. For creating a global ideal and **consistent** model

- Contact/connection of the components (incidence) and
- Collinearity of the axes of all components

have to be generated. This is only possible now, because during tracing the pipe run the position of all components that need to be considered is not yet known. This operation slightly changes all objects and the optimal direction of these changes depends on all connected components (possibly of multiple runs).

Running the command PCAC opens the dialog '*Plant Systems: ApplyConstraints*':



The status section explains the current progress in the process

When opening the dialog the first time, further elements are still disabled.

You work in two steps using this dialog. First click the [Select Plant Objects] button to select at least one component of the plant section to be corrected. The dialog disappears and you are prompted the following in the AutoCAD command line:

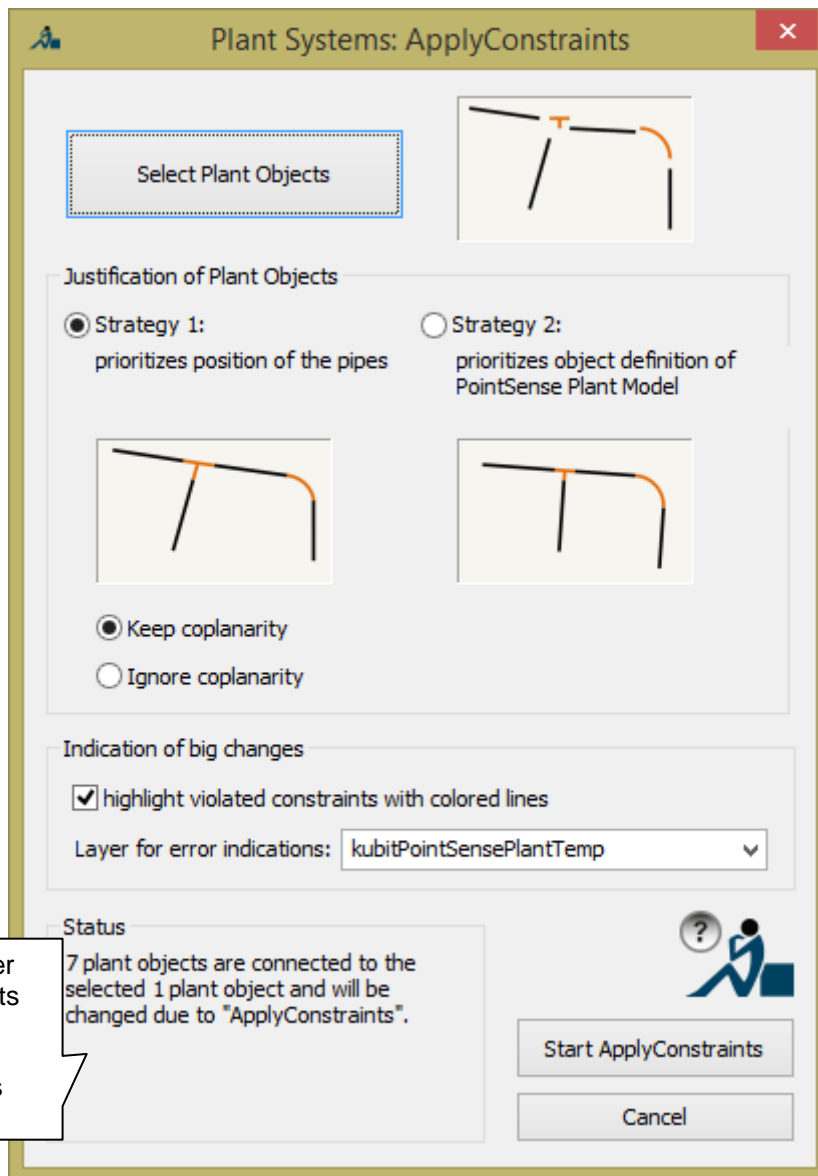
Select plant objects:

Select objects:

It is sufficient to select one single component of the section, since the whole connected section is recognized automatically. If you selected all desired components, just press ENTER and the command line prompts:

Review all connected objects [Continue] <Continue>:

At the same time all Plant objects which are connected with the component you selected are highlighted in green. This way you can visually check if the resulting assemblage is what you want. If that is the case, confirm your selection with ENTER and you get back to the dialog.




Now you can start the automatic correction for the creation of connectivity and collinearity by clicking the [Start ApplyConstraints] button.

Although the process is capable of correctly adjusting most of the pipe runs of practical use, it may happen on rare occasions or with uncommon sequences of plant objects, that the automatic process does not manage to fulfill all of the constraints. For this reason, you have the choice to activate the checkbox „*highlight violated constraints with colored lines*“ in the dialog. Hence, in the results of Apply Constraints, three problem classes are being identified and all of the occurrences are going to be visualized by lines of the colors green, red and blue.

Problem Class	Line Color	Graphical Representation
Significant movement at the end of a pipe axis	green	a single line is drawn perpendicular to the resulting axis for showing the track of the movement
Two connection points are not coincident	red	two lines represent the directional vectors of the connection points, a third one is connecting the coordinates of the connection points
Two connection points are coincident but not collinear	blue	two lines represent the directional vectors of the connection points

Regarding the first problem class, you are able to configure yourself which amount of axis movement you want to be considered as „significant“ change. You may use the command →Prepare – Settings to configure the corresponding value on page 4 of that dialog.


Pipes – Change insulation

	Command Name:	PCSIN
	Ribbon Tab:	As-Built Plant

With this command you may add or remove insulation to/from plant objects.

The insulation values have to be defined for the drawing advance (PCEDITSPM, →Prepare – Edit Plant Model).


Pipes – Select pipe run

	Command Name:	PCSPR
	Ribbon Tab:	As-Built Plant

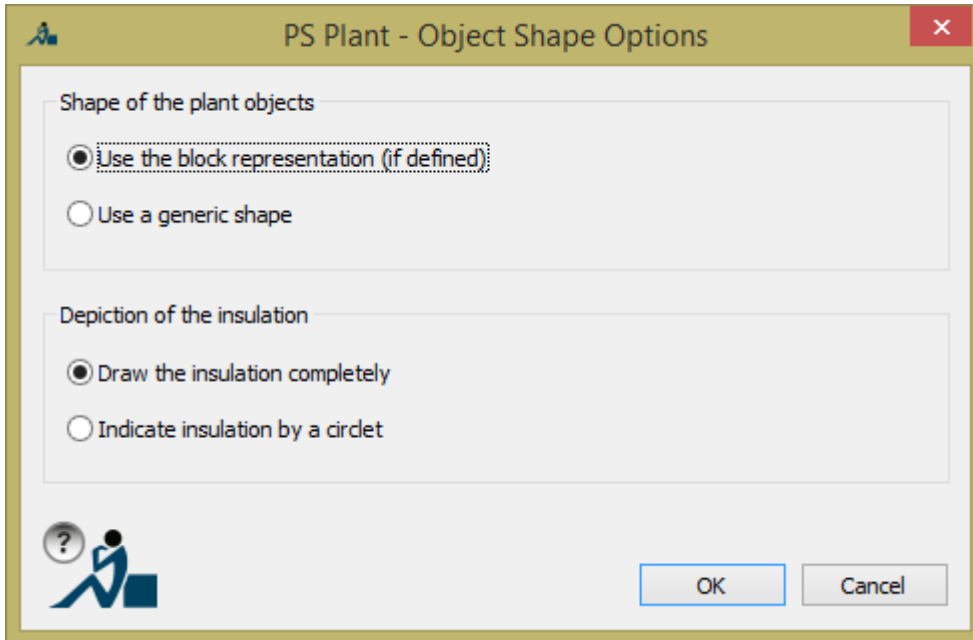
With this command you may add all parts of a pipe run to the current selection set of AutoCAD. You have to select only one part of pipe run.

This is useful if you want to change common properties of all parts of a pipe run simultaneously in the property window.

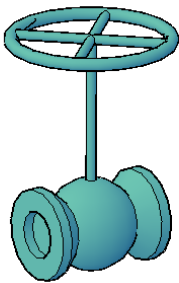
Pipes – Configure plant object's appearance

	Command name:	PCCTRLDISP
	Ribbon Tab:	As-Built Plant

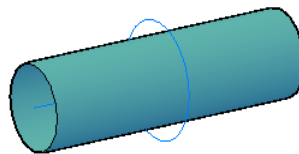
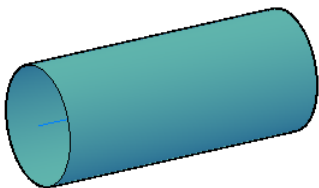
With this command you can change the display of plant objects.




In the left picture you can see the block representation of a fitting. If no block definition is available, you only get the generic shape like the right figure.



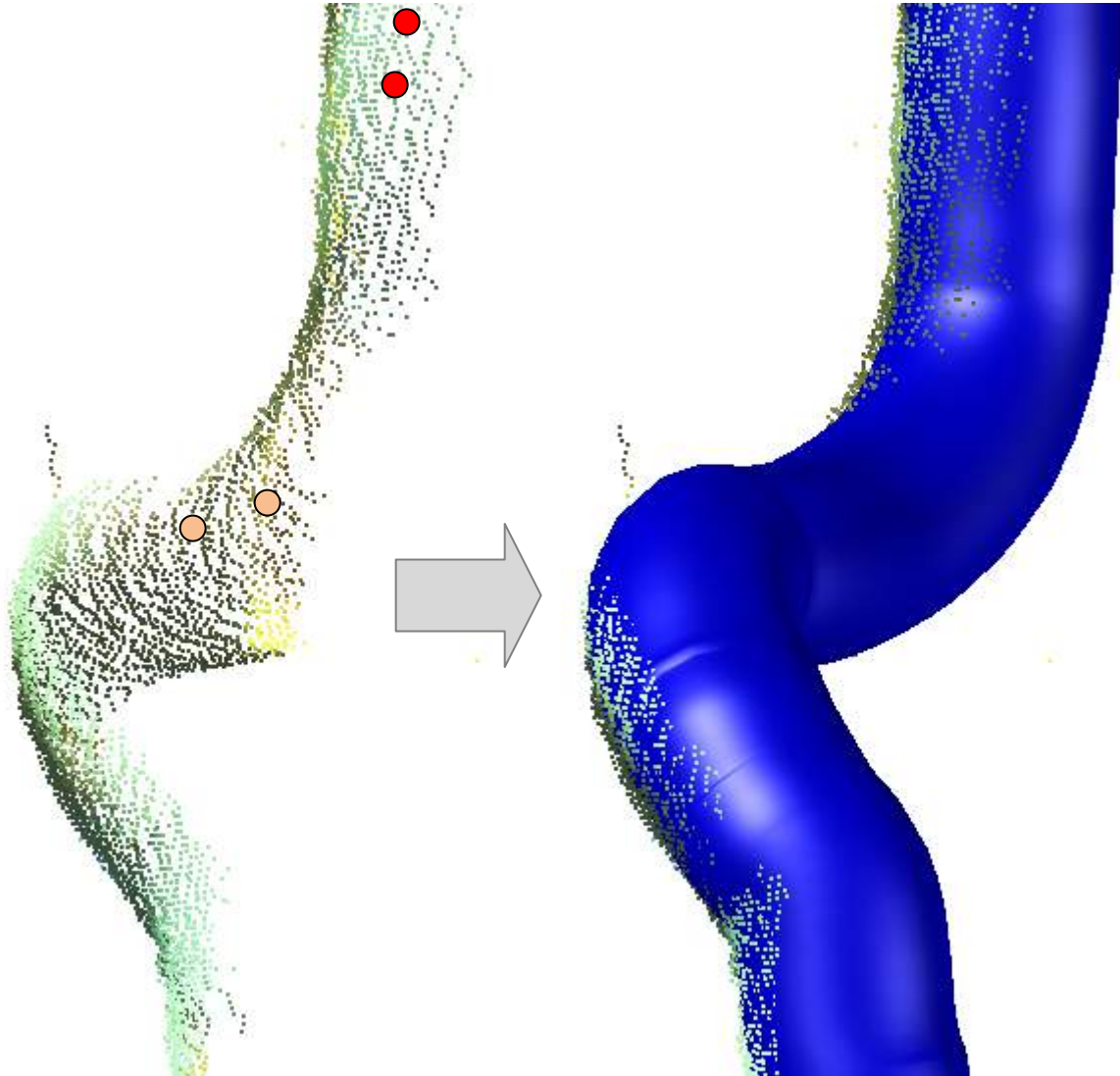
The display of insulation can be complete as in the left picture or only be symbolized by a centric ring like in the right figure.



Pipes – Fit bent pipe

	Command name:	PCFITBENTPIPE
	Ribbon Tab:	As-Built Plant

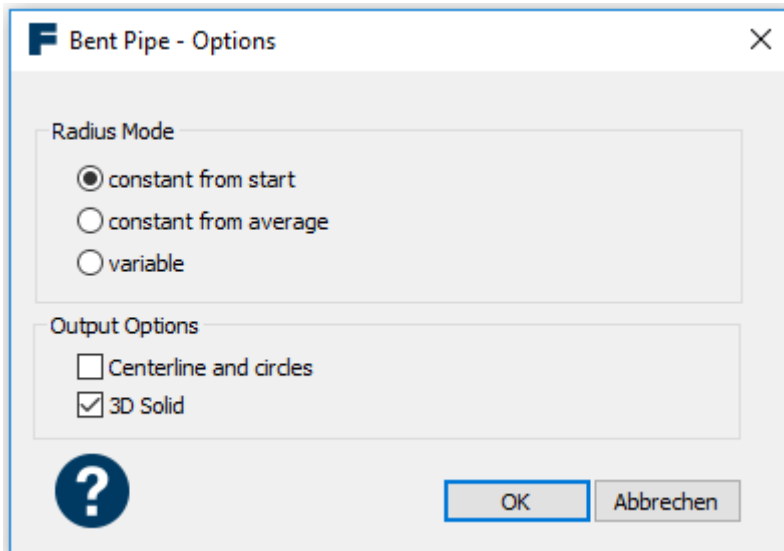
This command allows modeling bent pipes inside a point cloud. The result is a 3D solid and/or a spline for the axis with a set of circles representing pipe cross-sections.



The process is like this. You click two points of the point cloud at the beginning of the pipe/tube approximately in direction of the axis (dark dots in the figure above). The distance between both points tells the software the desired sampling.

If tracing the run stops - e.g. at a tight bend – you may click two further points (light dots in the figure above). The single segments are connected automatically. If AutoCAD fails to create the 3D solid, select the [Undo] option. You may then try to achieve the creation by clicking two slightly different points or you finish the run.

In addition, the [Options] command line option can be used to open the following dialog for configuring pipe tracing and output.



Three different radius modes are available:

- **Constant from start:**
The radius is determined once at the beginning of the pipe tracking. Then the system tries to generate the entire pipe with this radius.
- **Constant from average:**
Each radius of a found pipe section is used to recalculate the average of the radii. The pipe is then produced with the average radius.
- **Variable:**
The radius of each pipe segment is used as calculated.

As a result, the 3D solid and/or the axis line and circles in the drawing can be created.



Figure: point cloud




Centerline with circles

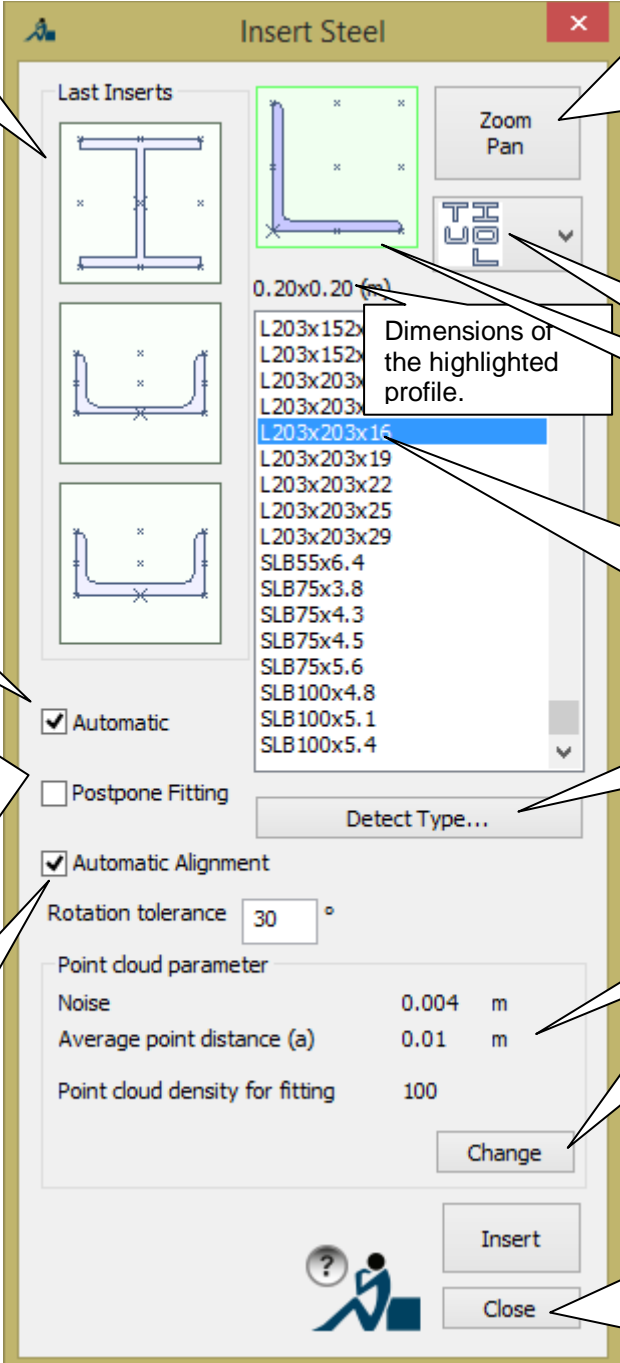


3D solid

Beams – Fit beam

	Command name:	PCFITBEAM
	Ribbon Tab:	As-Built Plant

With this command you may model beams (straight prisms with constant profile)
Following dialog opens:



The last three profiles used

In general the beams are positioned approximately by clicking two points and then fitted automatically. If this option is disabled, the positioning is done manually by clicking up to five points.

Objects are positioned approximately, the fitting is postponed until you click [Close]. Available if "Automatic" is enabled.

Beams are created precisely vertical, should the deviation from the vertical lie within the given tolerance.

Temporarily leave the dialog and navigate to a different view in the drawing.

Filter the list on primitive form.

Dimensions of the highlighted profile.

Profile highlighted in the list below

List of all available profiles according to the Active Set in EDITSPM

Automatic type recognition

Check point cloud parameters and change if necessary

Modeling beams with the selected profile; the dialog closes temporarily.

Close the dialog. If the "Postpone Fitting" option is enabled, the objects will now be fitted.

Insert Steel

Last Inserts

0.20x0.20 (m)

- L203x152x
- L203x152x
- L203x203x
- L203x203x
- L203x203x16
- L203x203x19
- L203x203x22
- L203x203x25
- L203x203x29
- SLB55x6.4
- SLB75x3.8
- SLB75x4.3
- SLB75x4.5
- SLB75x5.6
- SLB100x4.8
- SLB100x5.1
- SLB100x5.4

Automatic

Postpone Fitting

Automatic Alignment

Rotation tolerance 30 °

Point cloud parameter

Noise	0.004	m
Average point distance (a)	0.01	m
Point cloud density for fitting	100	

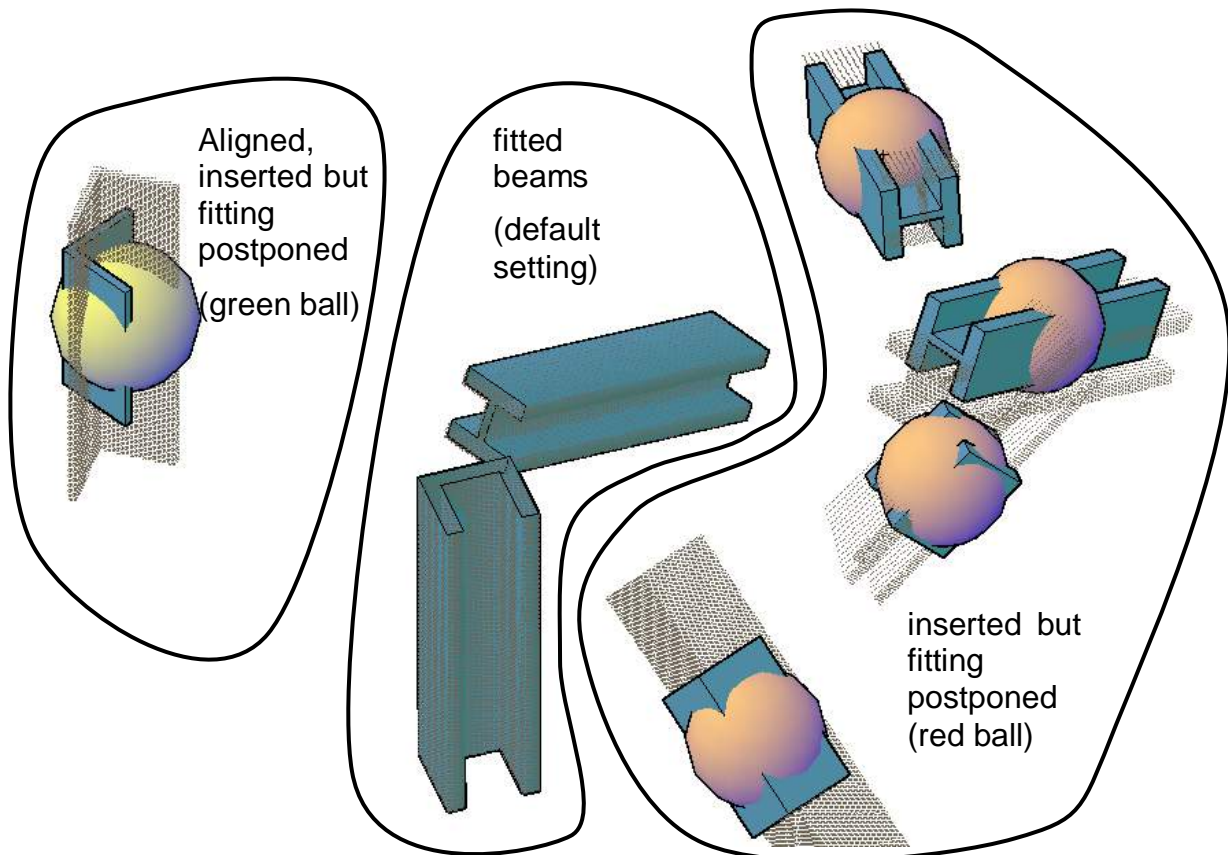
Change

Insert

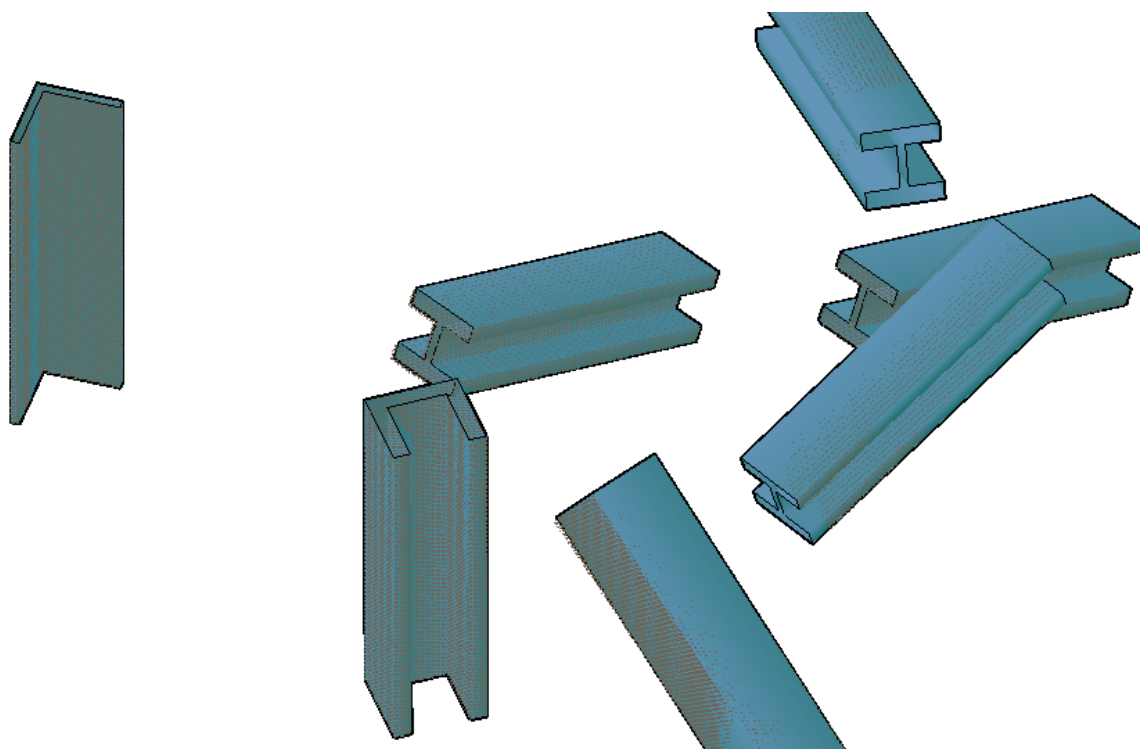
Close

In this dialog you select the type of beam to be modelled as well as the modelling strategy. If you click on [Insert], one of the icons or double click an item in the list, the dialog closes and you can start modelling a number of according beams.

The automatic fitting of the beams (alignment and length) may take several seconds on slow computers. If you want to avoid this delay, you select the "Postpone Fitting" option. The beams are then positioned only approximately but very quick and highlighted with a ball in the middle. When [Closing] the dialog the fitting for all beams is run as a *batch*. In the following figure you see a possible intermediate result before you clicked the [Close] button (the beams have not yet been fitted to the cloud).



After [Close] the batch runs and you get the results:



The batch can be paused by pressing <ESC>. If you later return to **Fit beams** and press [Close] again, the remaining beams will be fitted.

The option "Automatic Alignment" can be configured here:

KUBT_PC_CONFIG → Plant → Angular tolerance

If the centreline of the beam or its profile alignment varies less than the specified angle from the perpendicular, then it will be aligned precisely vertical.

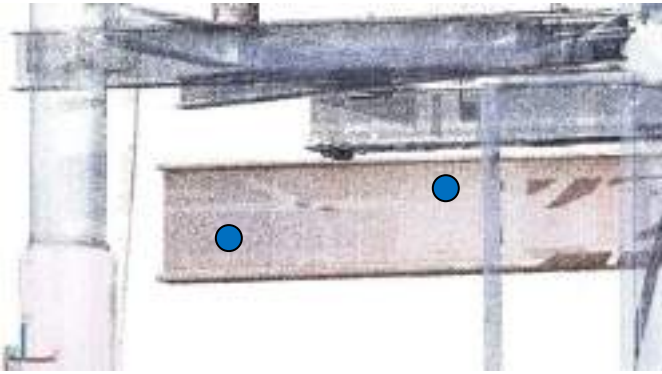
If the profiles have axes, they will be displayed in the dialog. The active axis (large cross X) can be swapped by pressing the middle mouse button. This axis will be used later for export or for alignment of beam to each other. The profiles that appear at the top of the dialogue box originate from the AutoCAD Plant 3D catalog.

C:\AutoCAD Plant 3D 2014 Content\CPak Common\Structural Catalog.acat:

If the type of the beam is unknown then click on the button [Identify type...]. Read about it in the following section:

Automatic type identification

After clicking the button [Identify type...] you will be prompted for two points. These must lay on one face of the 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). Click something like this:



After entering both points then follows the computation of selected face (with a progress bar). Then the following dialogue box appears:

Progress of the ongoing in the background comparison of the "current selection" from the PSPM. The list expands...

Filter the list on primitive form

Temporarily leave the dialog and navigate to a different view in the drawing.

Calculating... Compared: 2822 / 3453 entries (82 %)
✕

Slice position

Shift the candidate point cloud slice along the beam centreline.

Following the shifting of the slice or the removal of points, the computation of the hierarchy should be started again.

Previous face from the two picked points.

Section through the point cloud and in blue the profile of the current selection in the list. The actual noise setting (n) is displayed in white.

Automatically display the most suitable type in the list.

Return to the main dialog

TOOL
Zoom Pan

- W6x25
- HEB-160
- W6x20
- W150x37
- W150x30
- W150x22
- W6x15
- HEA-160
- HEB-180
- S150x26
- W6x16
- HEM-180
- IPE-160
- WT8x20
- UPE-160
- S6x12.5
- INP-160

Select best

?
Use this type

Cancel
Insert beams of this type

Hierarchy of the suitable types. At the top of the list is the best one. The selected profile is depicted on the left.

Return to the main dialogue box. The currently selected type is also selected in the main dialog.

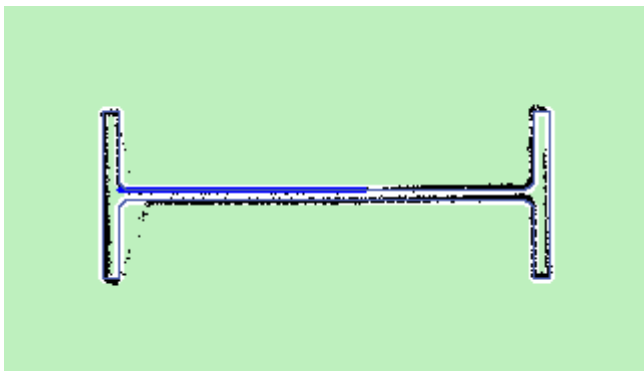
Insert the selected type into the point cloud. All of the normal options can be applied.

With the appearance of the dialogue box there begins in the background a comparison between all of the types of the "current selection" from the PLANT MODEL and the point pattern. All of the types that match the selected face will be displayed in a sorted list in the right hand side of the dialogue box. At the top of the list is always the type that, according to the pattern recognition, best fits. As time progresses the best candidate will change and will be automatically marked. With the button [Insert beam] is the process of identifying the type is ended and simultaneously an object of this type is inserted. When you are satisfied with the identification of the type, you can also return to the main dialogue box using the [Use type] button.

Left - in the green highlighted graphic - the selected candidate is visualised over a slice of the point cloud. As long as the search has not been finished and a manual choice has not been made, the profile is displayed in a light blue colour. If the choice has been decided (by clicking in the list or because the search has finished) the profile is displayed in dark blue.

The face initially selected by picking the two points will be displayed slightly thicker and bright blue. The point cloud noise used by the drawing is displayed white around the object. The noise corridor should be just wide enough so that all significant points lay within it.

The graphic should look something like this:



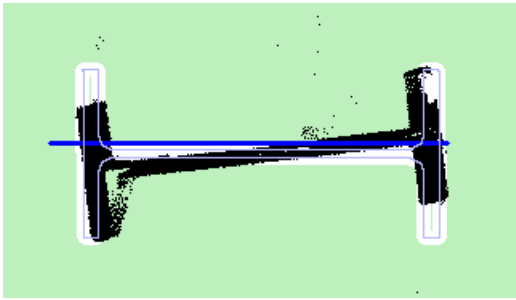
What is the purpose of the option "Select best type"?

Should one of the compared candidates in the list be clicked on with the mouse the automatic change to the currently assumed best one is stopped. Should the best type be automatically selected, then the check box "Select best type" must be ticked.

Solutions for typical problems

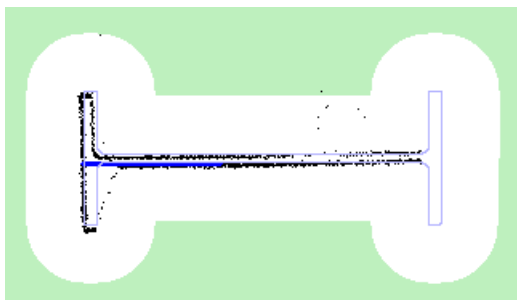
Both of the following effects occur quite commonly:

1. If the two points picked to define the first face inadvertently do not lay on the same face, the point cloud appears to be very wide or skewed in space and often it is not horizontally or vertically aligned:



Exit the type identification [Cancel] and start again selecting better points.

2. If there is too much point cloud noise, points from other objects can be a problem:



Completely exit the command and enter the correct point cloud noise setting (PCCFG, second step of the assistant).

Delete and shift the slice to be analysed

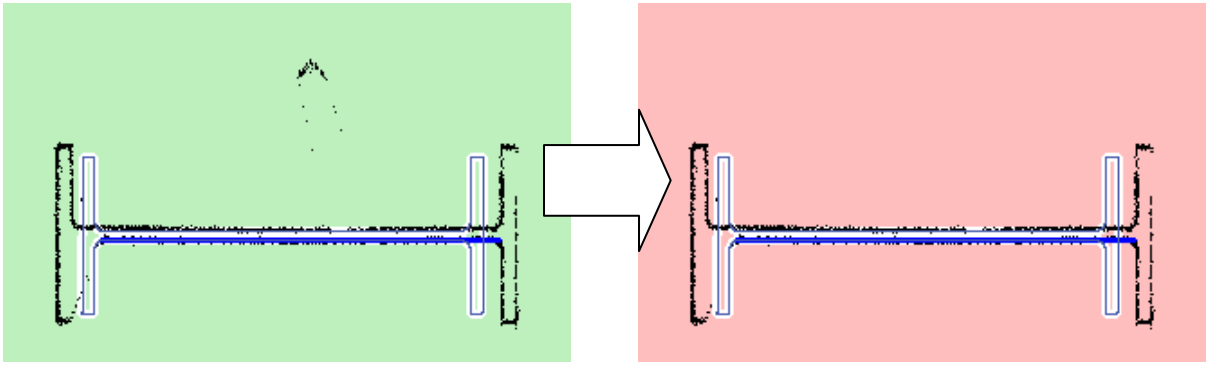
To assist the pattern identification process

- the point cloud slice can be shifted along the axis of the beam and/or erased in the graphic window.
- and/or erased in the graphic.




In both cases the background will be changed to a pink colour. The button [Recalculate] is now enabled as the background pattern identification is no longer being calculated from the current point cloud data. When you are finished with shifting or erasing the slice and the pattern identification has not brought a satisfactory result the calculation must be started again.

The mouse pointer, when it is in the graphic window, will change to a square. To erase the erroneous points, just click on them. The points are not really removed from the point cloud, they are just ignored in the pattern identification process:



Beams – Edit beam

	Command name:	PCEDITBEAM
	Ribbon Tab:	As-Built Plant

With this command you can edit an existing beam (kubitplantbeam).

This can be useful after PCFITBEAM with activated “Postpone Fitting” or if the position or type of a beam seems not to match the point cloud.

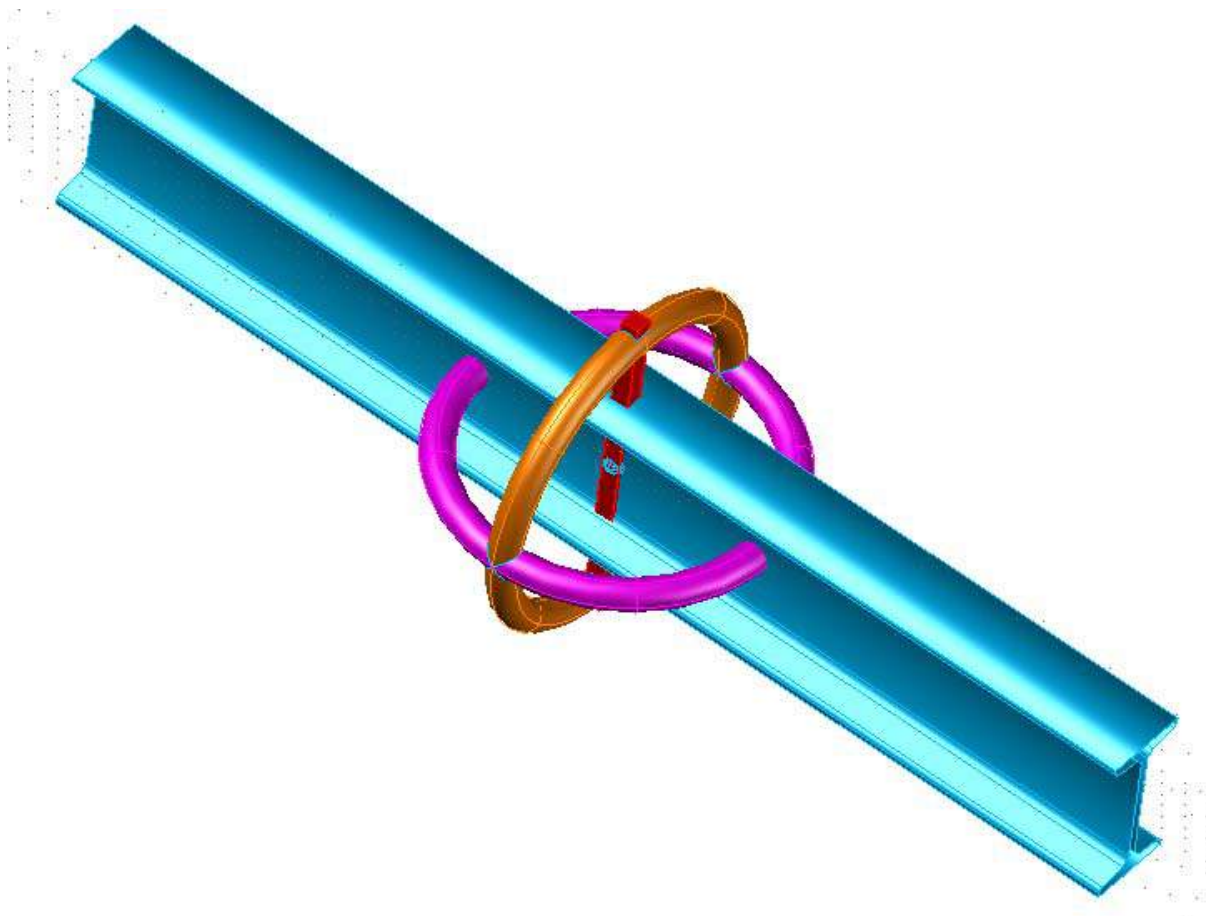
This command will ask for a beam and refit the selected object. Afterwards it offers same options as PCFITBEAM. Especially command line options “Retry” (in case of flipping is needed) or “Swap” (change of type is needed) can help to optimize an already existing beam.

Beams – Show the alignment of the beam:

	Command name:	PCSHOWALIGNSTEEL
	Ribbon Tab:	As-Built Plant

With this command the precise alignment of the beam can be visualised. Especially small deviations from the horizontal or vertical can be recognised without the need to re-measure it. The command toggles between visualisation and off.

A completely skewed beam looks like this when the visualisation is switched on.

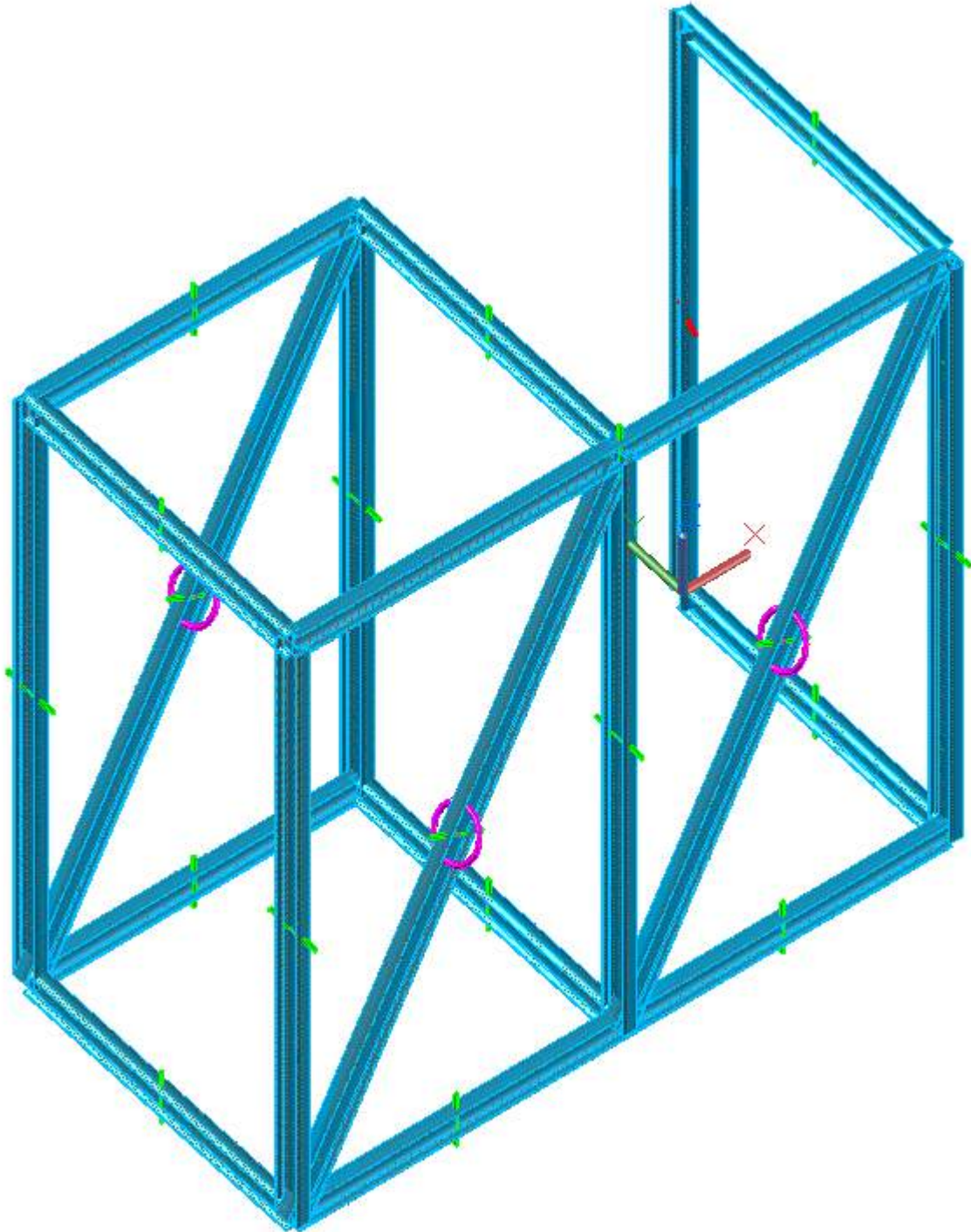


- The **magenta** coloured circle means that the axis is neither horizontal nor vertical.
- The **orange** coloured circle shows a tilting of the profile around its own axis.
- The **red** axis means that the axes of the beam are not parallel to the user coordinate system (UCS).

This marking shows *not* the deviation from the point cloud but deviation from a typical beam construction.


Hint: The command LIST displays the associated angle (deviation from a typical position).

After applying PCACSTEEL (ApplyConstraints for beams) a network of beams usually looks like this:

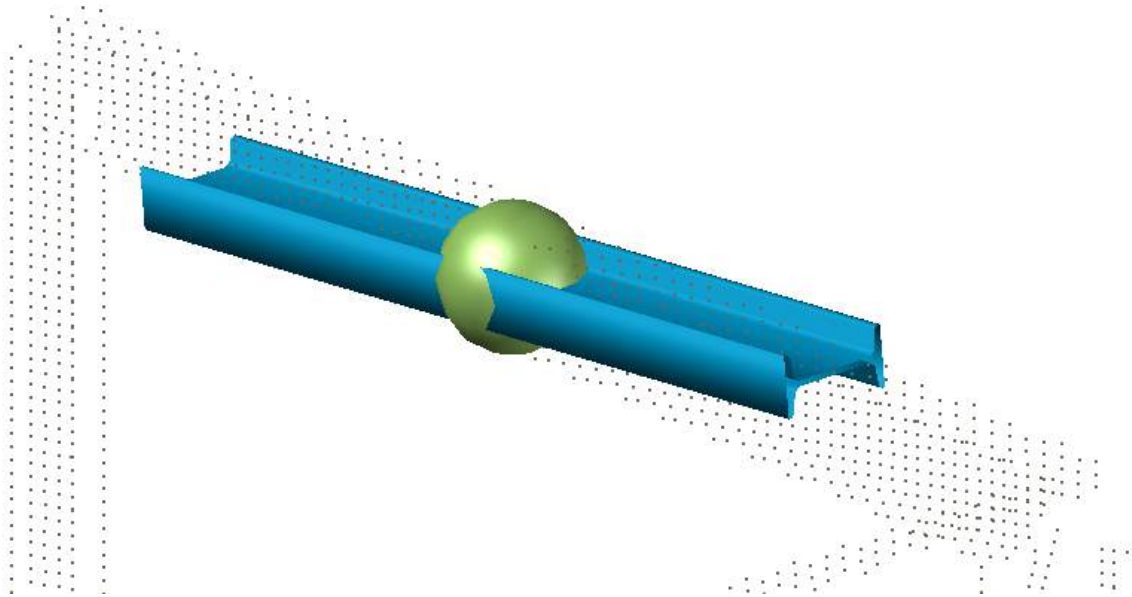


Only the diagonals are marked **magenta**. The axes are green, because they match the UCS. All slight tiltings and slight deviations from the horizontal or the vertical positions are eliminated up to a predefined threshold value. The vertical beam, rear-right, has in this example a large rotation. This stays as it is, as can be seen by the **red** axis.

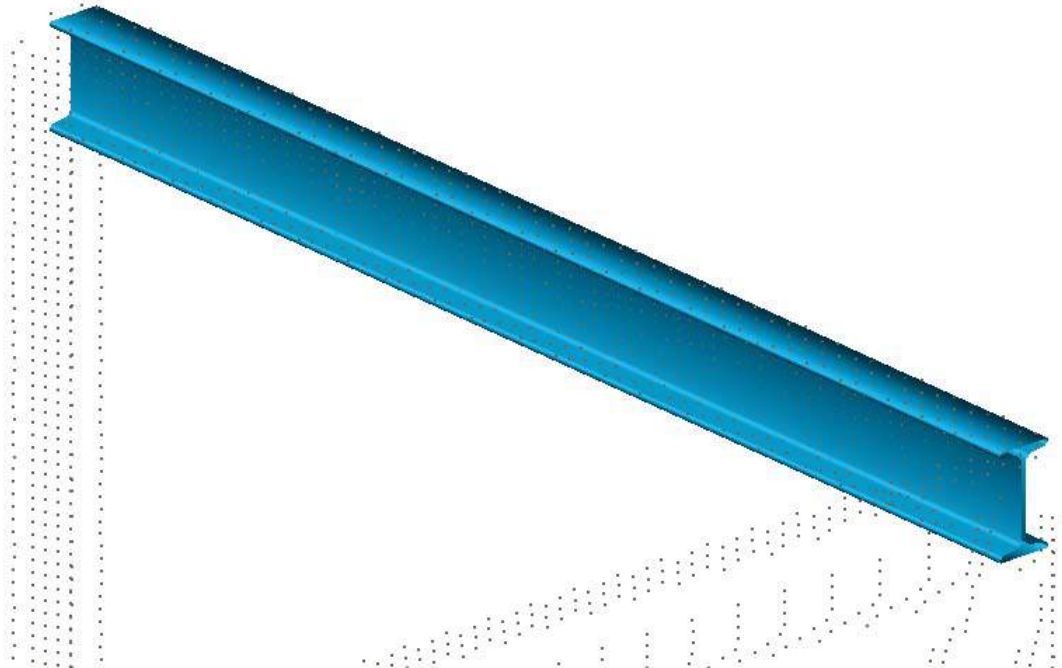
Beams – Switch customisation of the beam

	Command name:	PCTOGGLETOFITSTEEL
	Ribbon Tab:	As-Built Plant

A beam has been marked with a **green** or **red** sphere, when its position has not been optimised with the point cloud. That happens when a beam has been copied or PCFITBEAM (Fit beam) was used with the mode “Deferred fitting”.



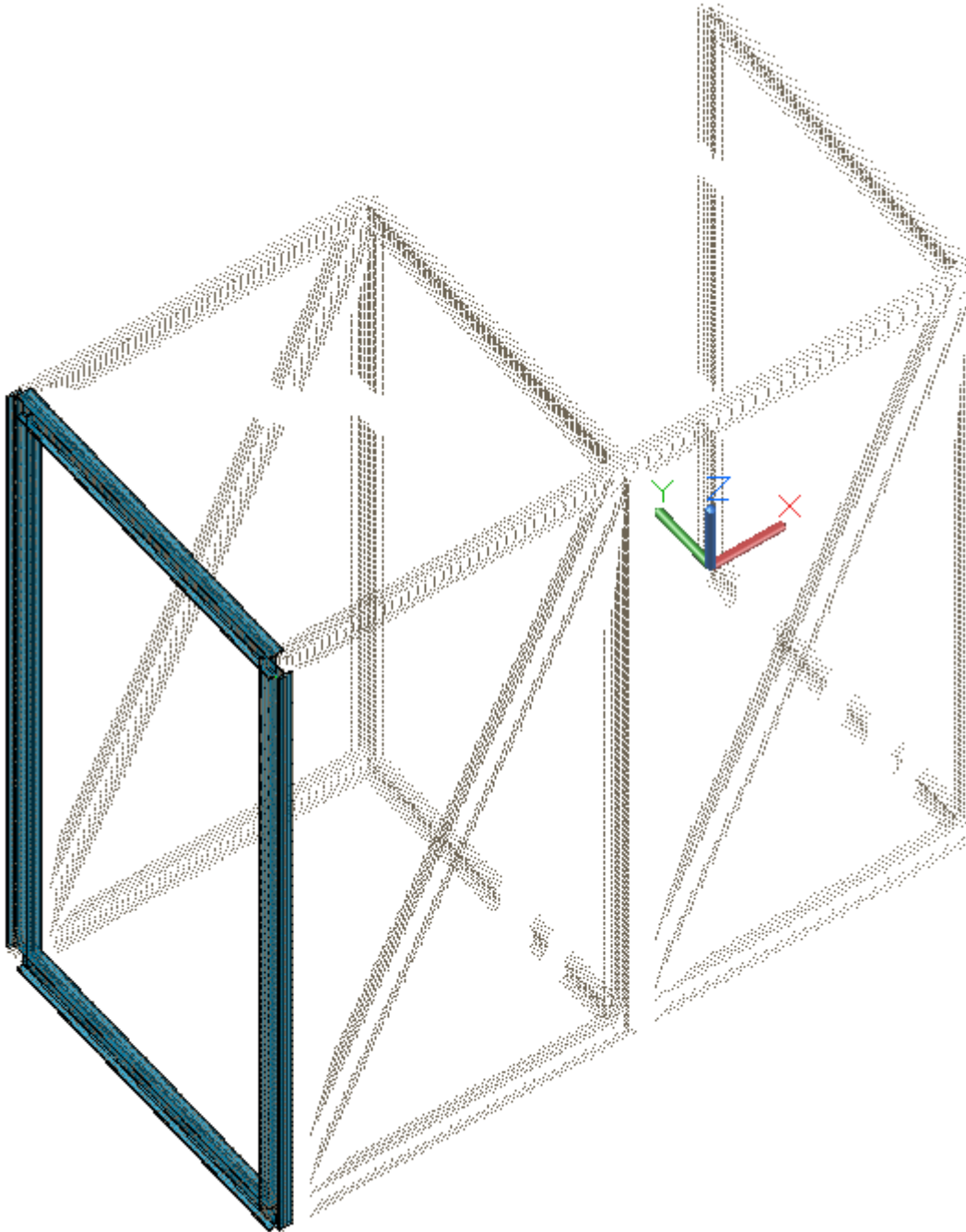
Fitted beam:



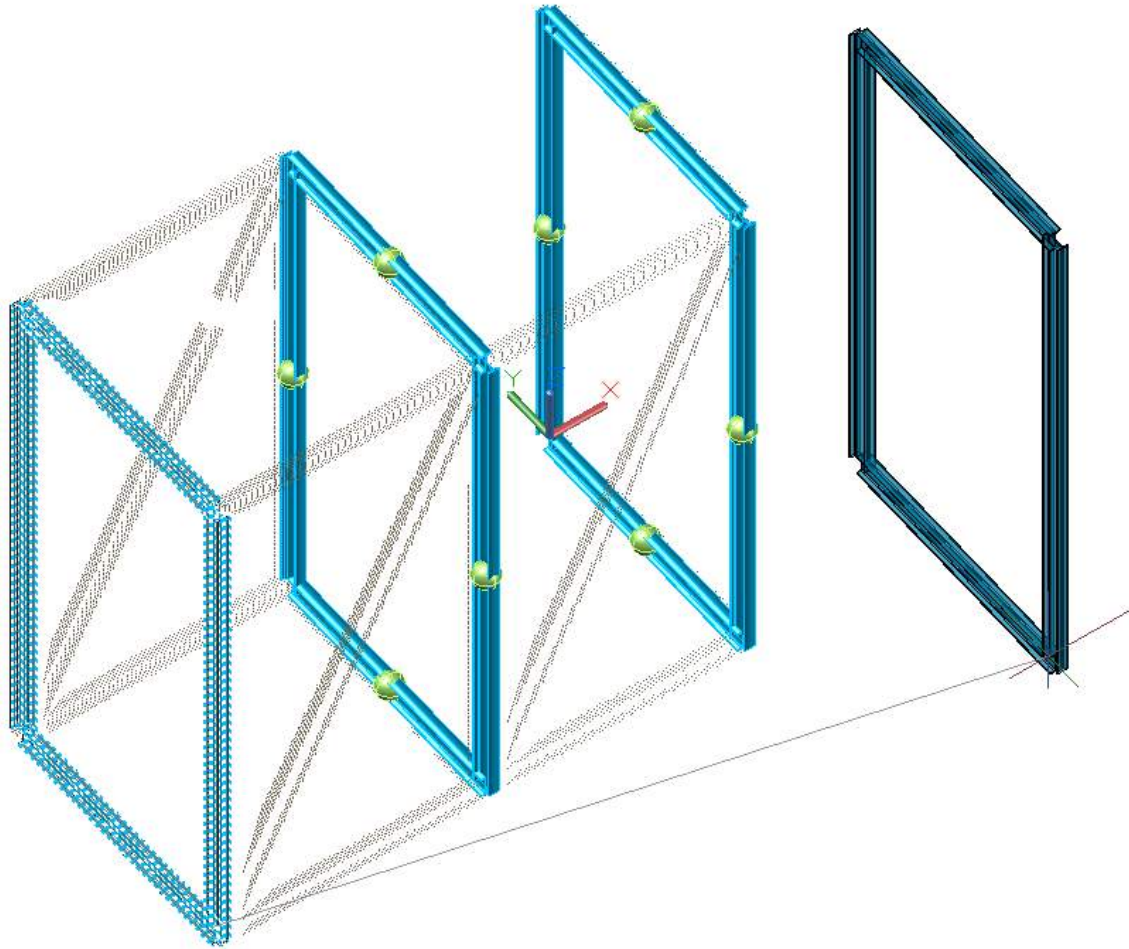
The status of one or more beams can be changed with this command. No fitting of the beam is carried out.

A possible areas of use for the beam is as follows:

A network of beams is to be modelled. Four steel beams will be fitted using PCFITBEAM:

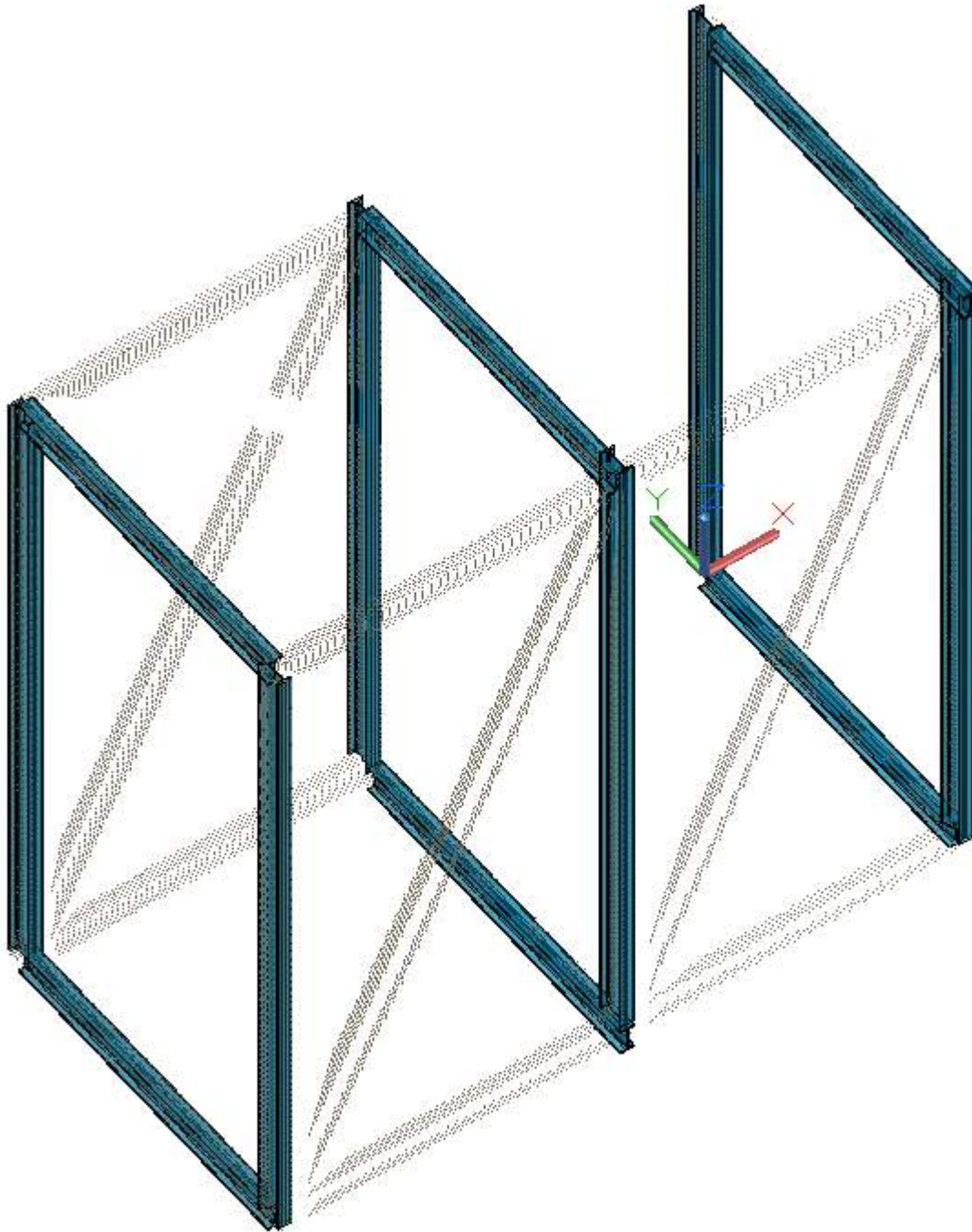


Now these beams will be copied (AutoCAD command `_ARRAY` or `COPY`):



The beams marked with a **green** sphere have not yet been fitted.

Subsequently run PCFITBEAM again and immediately click on [Close]. Now you get the possibility to fit the eight marked beams onto the point cloud.



Then the results are checked. If a steel beam appears to be badly fitted, then it can be marked and fitted again using the command `PCTOGGLETOFITSTEEL`. In many cases it helps.


When the copying has been precisely carried out and steel beams, for example in shadowed areas without any scan points, should not be fitted, then the later fitting of the beams can be prevented with this command. `PCTOGGLETOFITSTEEL` is a switch command to select between three states. The green sphere symbolizes the automatic fitting method which uses the configuration parameters from the “Fit beam” dialog. A fitting without any restrictions is symbolized by the red sphere. Besides automatic fitting and fitting without restriction, i.e. free fitting, the user has the option to disable the postponed fitting.

Moreover, the user can select a set of beams and start the command PCTOGGLETOFITSTEEL with the selected beams. If all beams have the same state (automatic fitting, green sphere for example) the states get switched in a predefined order:

Green → red → off.

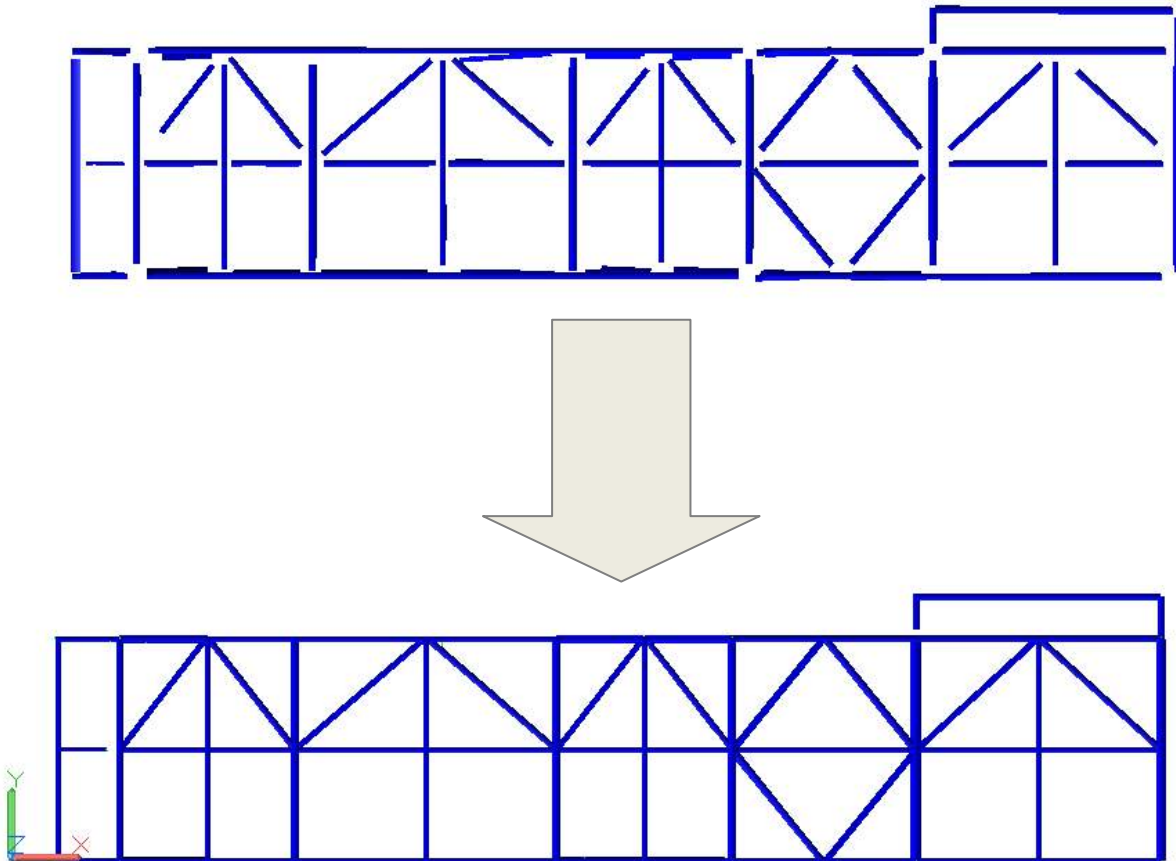
If the user chooses a set of beams with different states, the user will be asked which fitting method shall be set for all selected beams.

Beams – ApplyConstraints for steel

	Command name:	PCACSTEEL
	Ribbon Tab:	As-Built Plant

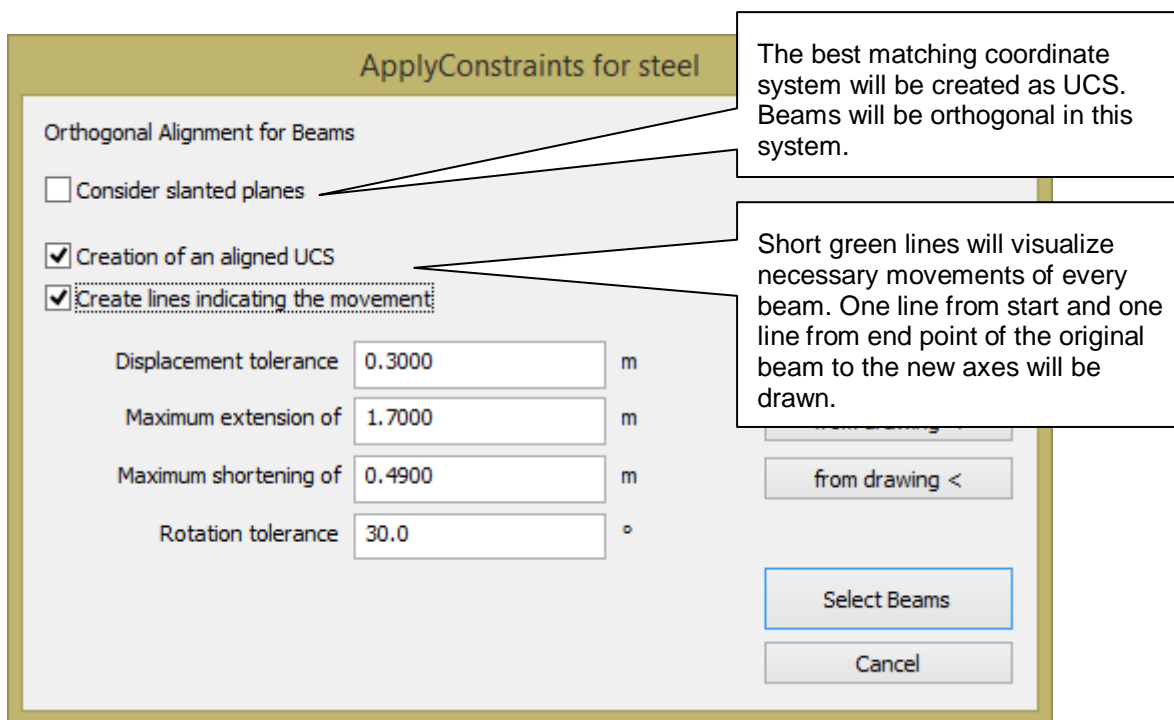
With this command you can align a set of beams (kubitplantbeam) one to each other.

This is nice for further (static) analysis or further design based on the real conditions in other software (E. g. Revit® after export using PCEX). Here are the results of the operation in a top view (actually the whole 3D construction was optimized!):



Beams will become orthogonal and coplanar. Additionally beams will be extended to their next neighbors.

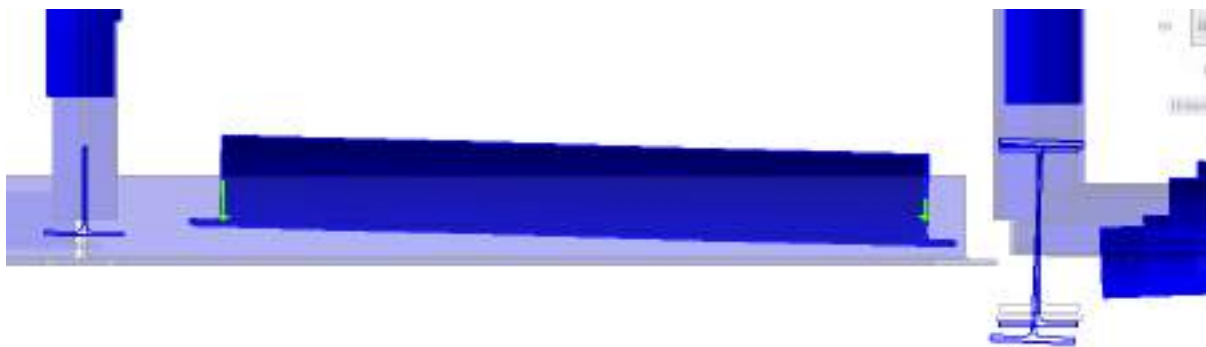
The command starts with a dialog to give you some options:



There is a parameter that limits the tilting of the axis, the *Displacement tolerance*. The *Rotation tolerance* influences the allowable rotation around the axis. *Extension* and *Shortening* influences the intersection.

PCACSTEEL guarantees to move a beam's endpoint not more than *Tolerance* value, except for extending the beam. This means the *Tolerance* has to be significantly smaller than the shortest beam but large enough to compensate the biggest deviation from the perfectly shaped steel network.

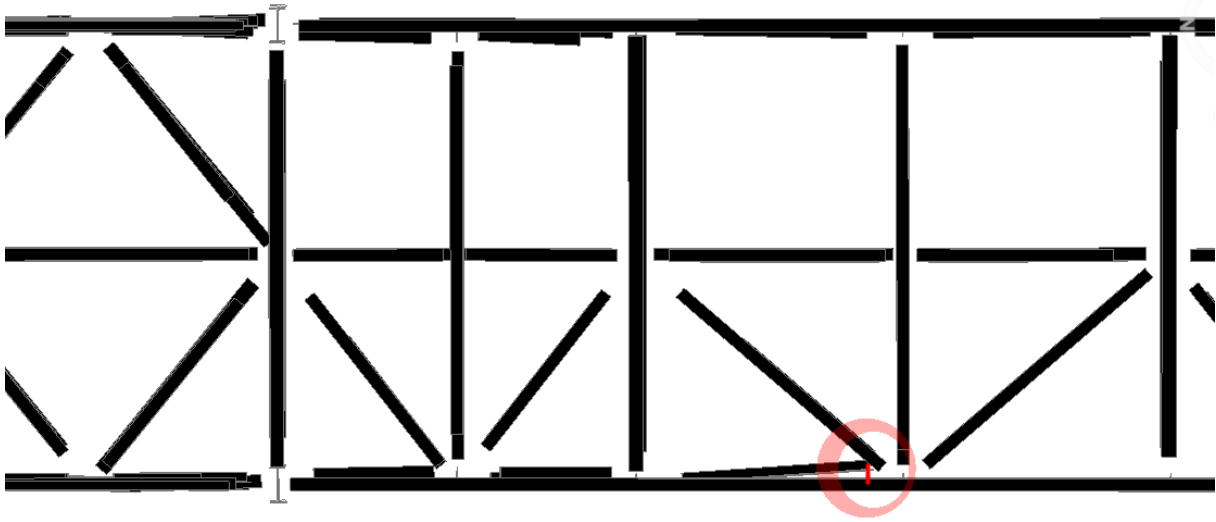
In following figure the green arrows are guaranteed shorter then *Tolerance*:




How does one come to a good value for the Displacement tolerance?

We look for the worst situation (from Top view and any side view). Then we draw a line without object snap (**red line** in following figure). With the length of this line we had to correct the bad beam.

We add something like 20% on this length to be safe and this will be a good tolerance. Tolerance should be significantly shorter than the shortest beam involved.



Beams – Adjust steel

	Command name:	PCADJUSTSTEEL
	Ribbon Tab:	As-Built Plant

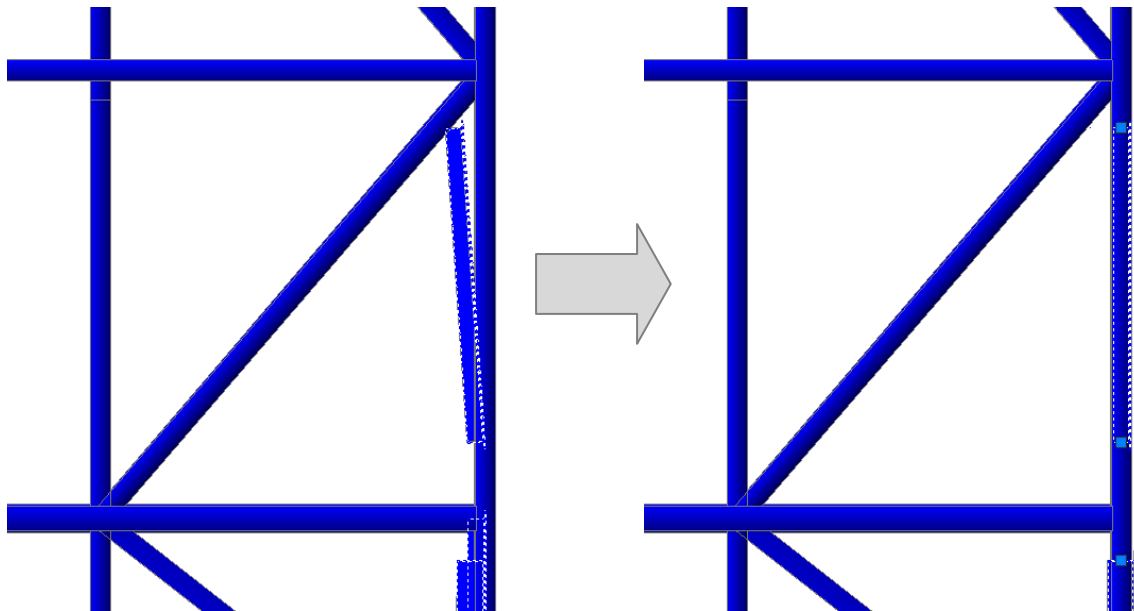
With this command you can align a set of beams (kubitplantbeam) to one reference beam.

Usually this is used as a post edit command after PCACSTEEL. If after the global command some beams are still not adjusted (e. g. because for them a longer tolerance is needed) you may apply this command.

First select a set of beams which you want to adjust. There is an option [Tolerance] with same meaning like in PCACSTEEL


Last step is to define the reference beam.

Here we adjust three beams to the rightmost beam:

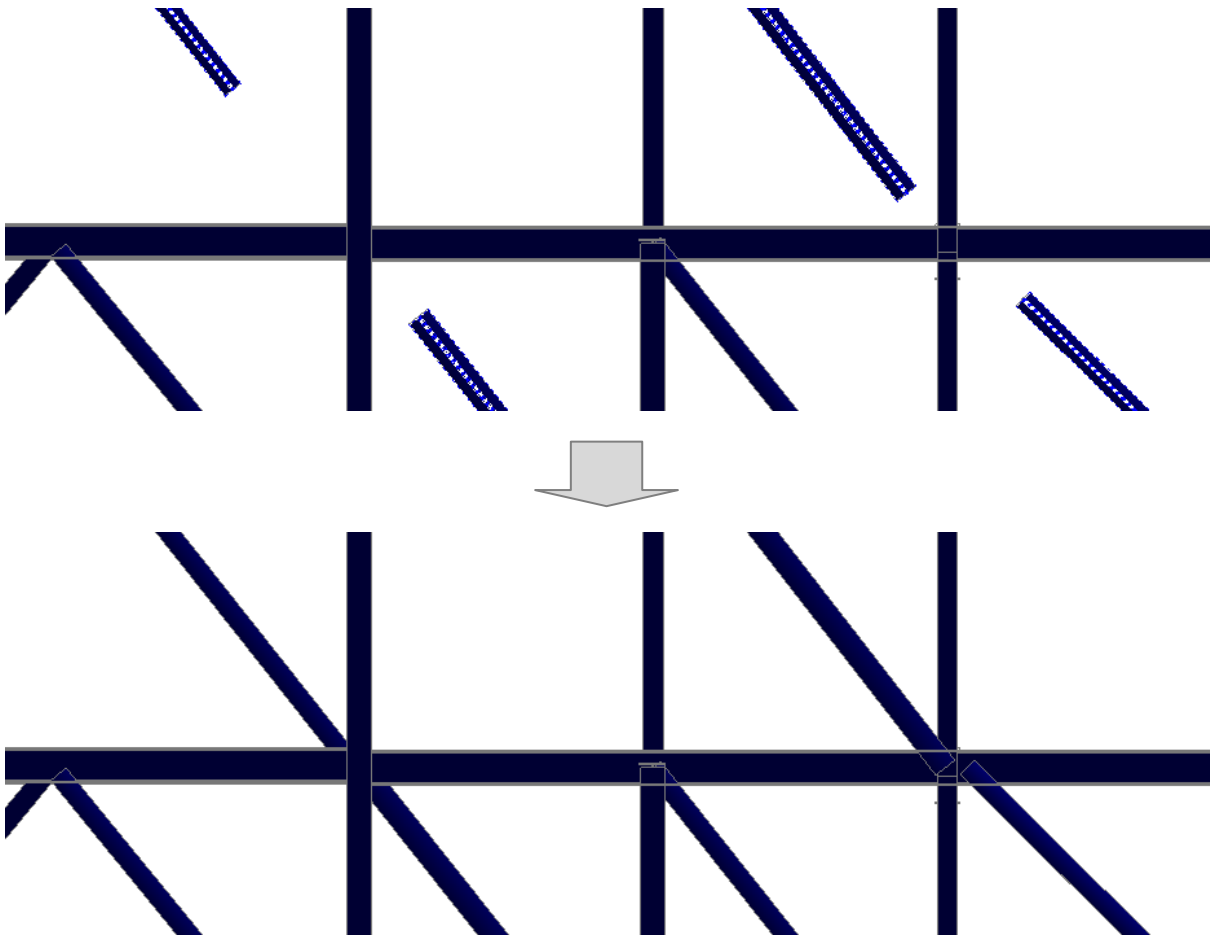


The maximum deviation (always smaller than the tolerance) will be given in the command line.

Beams – Extend steel to steel


	Command name:	PCEXTENDSTEELTOSTEEL
	Ribbon Tab:	As-Built Plant

With this command you can extend a set of beams (kubitplantbeam) to one reference beam. Usually this command is used as a post edit command after PCACSTEEL and PCADJUSTSTEEL.



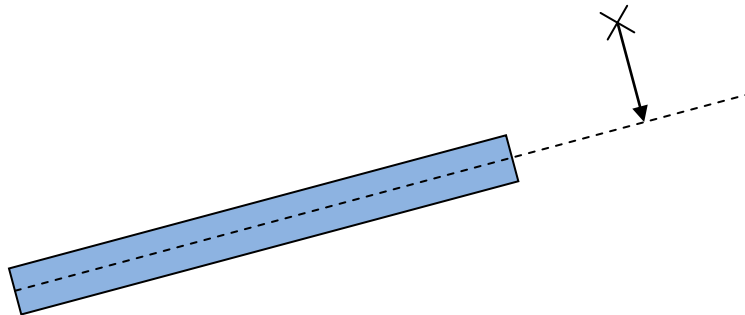
Usually all beams including the reference will be coplanar but command will work anyway. The maximum deviation (distance between the planes containing the beams) will be given in the command line.

Beams – Extend steel to point


	Command name:	PCEXTENDSTEELTOPOINT
	Ribbon Tab:	As-Built Plant

With this command you can extend a set of beams (kubitplantbeam) to a point. Usually this command is used as a post edit command after PCACSTEEL and PCADJUSTSTEEL.

For every beam the point will be perpendicularly projected on beam's axes. The beam will be extended to projected point.



Beams – Intersect beam axes

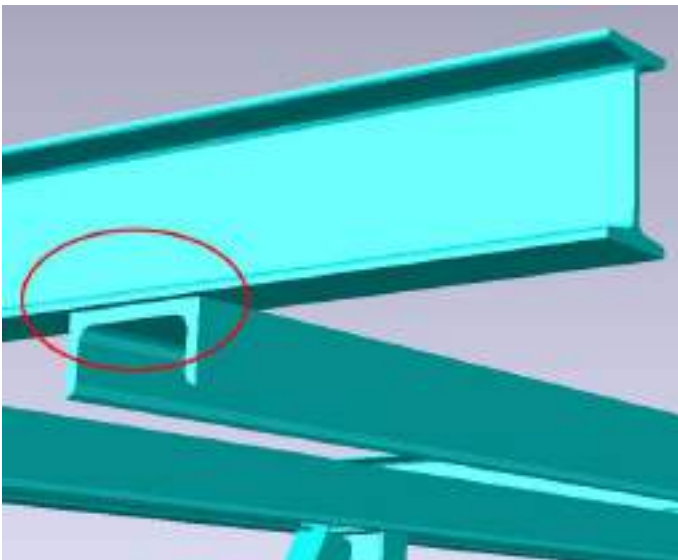
	Command name:	PCINTERSECTAXISSTEEL
	Ribbon Tab:	As-Built Plant

With this command (KUBITPLANTBEAM) the steel beams can be trimmed. Thereby all of the selected steel beams are either extended or shortened. But the straight beam axes are not (!) changed.


This command is usually the second step in a process, at the end of which there exists a perfectly aligned network of steel beams.

1. Fit beams PCFITBEAM (→Beams – Fit beam)
- 2. Intersect beams PCINTERSECTAXISSTEEL**
3. Check intersection and where necessary, by using the grip handles, manually adjust it to the approximate correct length.
4. Show axes by using PCSHOWAXISSTEEL (→Beams – Visualize beam axes)
5. Automatically select axes with PCADJUSTAXISSTEELAUTO (→Beams – Align beam axes automatically)
6. Check and where necessary correct axes with PCADJUSTAXISSTEEL (→Beams – Align beam axes manually)
7. Align the beams orthogonally and coplanar with PCACSTEEL (→Beams – ApplyConstraints for steel)

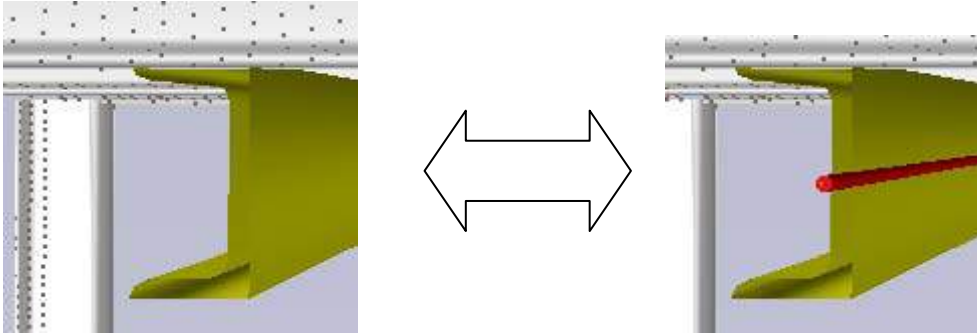
The intersection of the beams has a significantly higher priority than PCADJUSTAXISSTEELAUTO (→Beams – Align beam axes automatically). The fixing of the axes is important when dealing with “stacked” beams, where the axes should not lie in the middle of the profiles.



Beams – Visualize beam axes


	Command name:	PCSHOWAXISSTEEL
	Ribbon Tab:	As-Built Plant

With this command (kubitplantbeam) the axes of the steel beams can be hidden and shown.



The axes can be exported (→Export; PCEX) and are used by PCACSTEEL (→Beams – ApplyConstraints for steel) and aligned to each other.

Beams – Align beam axes automatically

	Command name:	PCADJUSTAXISSTEELAUTO
	Ribbon Tab:	As-Built Plant

With this command the software looks for, in all of the selected beams, those axes which lead to a complete and connected axis system. “Stacked” beams can be seen in the image. At the arrow there is a gap, because the axes of the beams still run centrally:



The objective is to select the axes, for example, in such a way that touch each other:



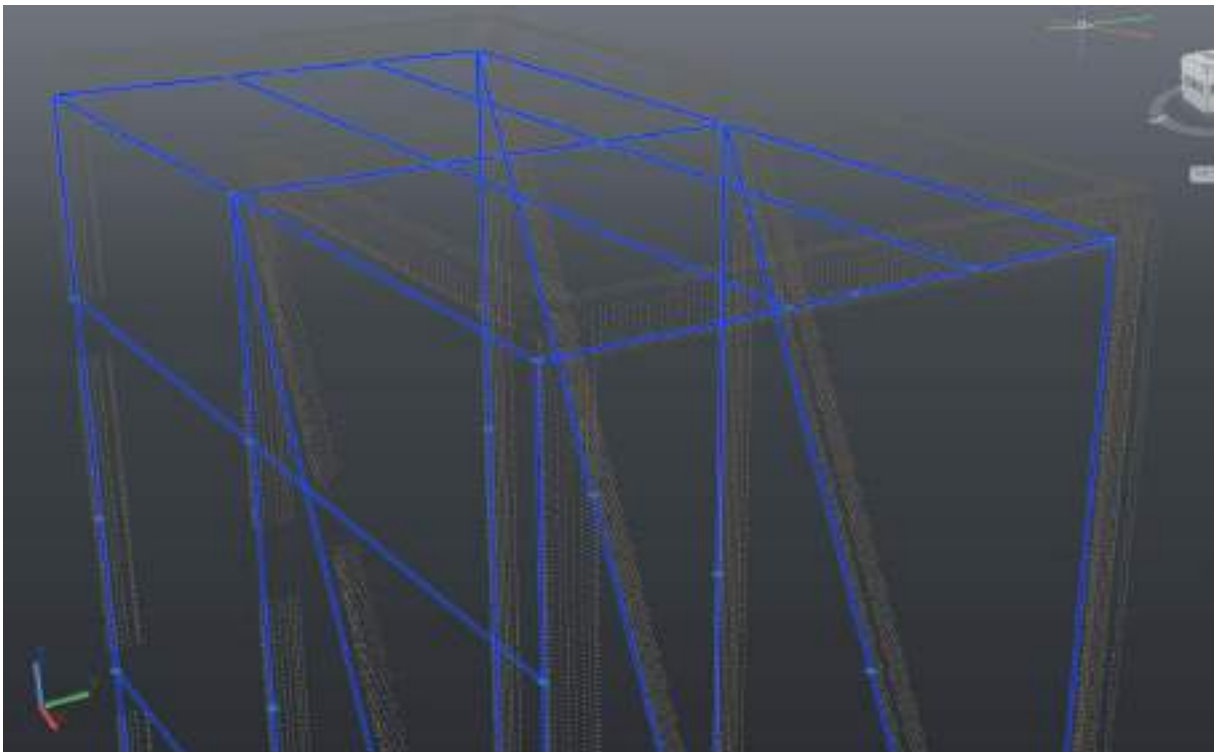
For the automatic detection of an optimal axial position it is necessary that the axes of the steel beams are brought together and that they touch each other. This is achieved by previously running the command PCINTERSECTAXISSTEEL (→Beams – Intersect beam axes).

The results of the axis search can still be manually revised: PCADJUSTAXISSTEEL (→Beams – Align beam axes manually).

Typically the next step after axis alignment is PCACSTEEL (→Beams – ApplyConstraints for steel). So is a connected axis system produced:




Here is the axis model (→Export; PCEX):

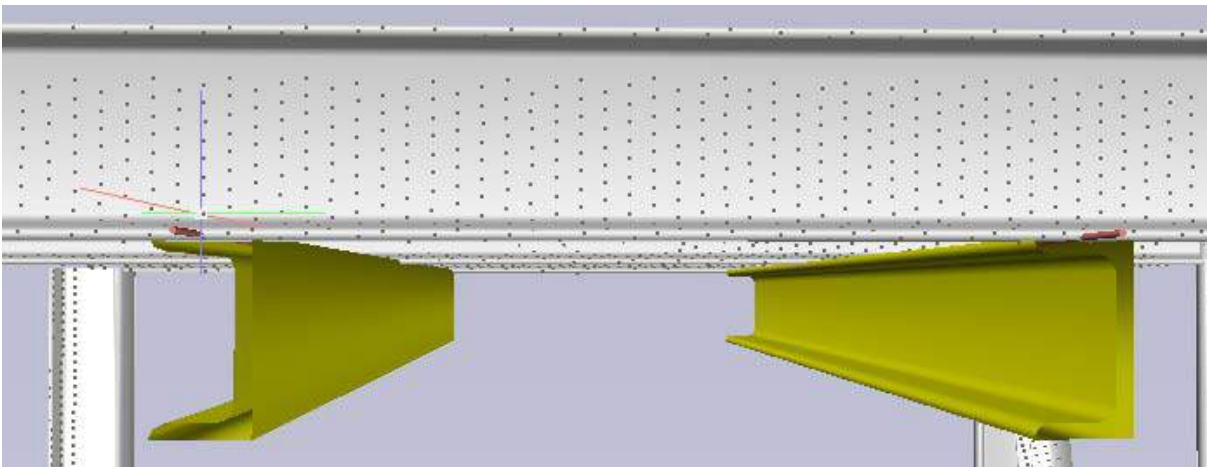
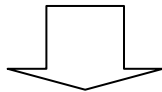
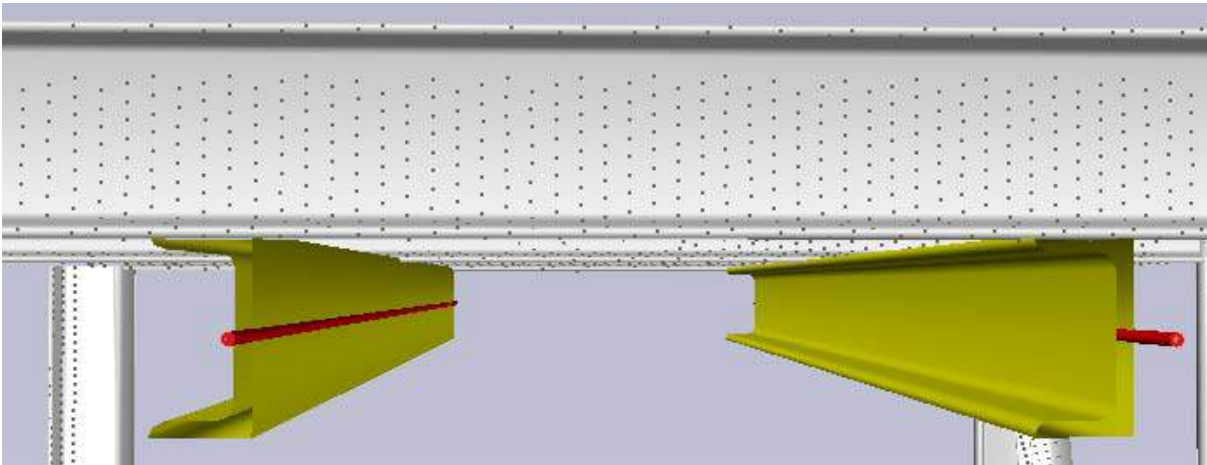


The complete workflow for stacked steel beams is shown in PCINTERSECTAXISSTEEL (→Beams – Intersect beam axes).

Beams – Align beam axes manually

	Command name:	PCADJUSTAXISSTEEL
	Ribbon Tab:	As-Built Plant

With this command the axis in the selected beams can be changed.




One clicks in one of the selected beams in the vicinity of the desired axis. The axis change will be applied to all of the selected beams.

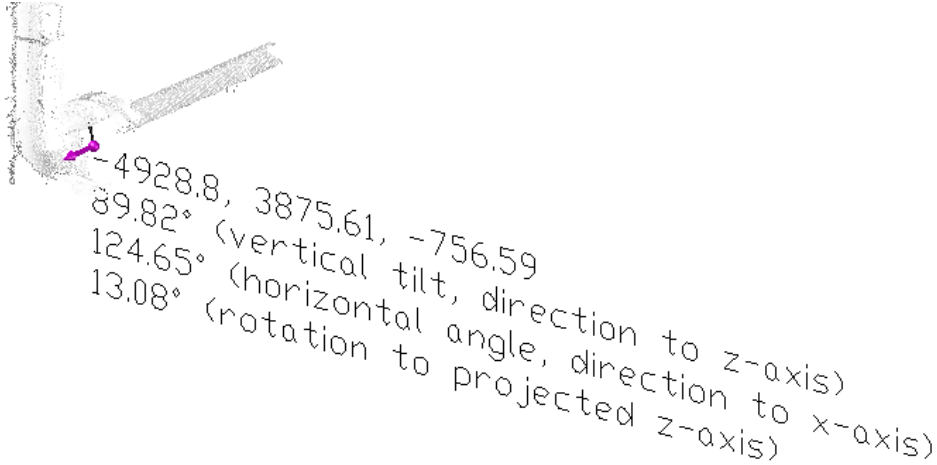
Usually the reason for this manipulation is that in the end one wants to have a complete and connected axis system.

The complete workflow for stacked steel beams is shown in PCINTERSECTAXISSTEEL (→Beams – Intersect beam axes).

Tie-in points – Determine tie-in points

	Command name:	PCTIEINPOINT
	Ribbon Tab:	As-Built Plant

This command is for determining the position of a tie-in point (Tie-in Point). In particular its coordinates and direction are determined. Additionally the alignment of the bolt holes (angular positioning around the centreline) can be ascertained. Here is a possible result:



Starting the command the displays following dialogue box:

The screenshot shows the 'Tie-In-Point' dialog box with the following elements and callouts:

- Methods for determining the centerline:** Three 3D models of a flange and pipe assembly showing different ways to define the centerline.
- Face or back of Flange:** Radio buttons for 'Front of Face' and 'Back of Face' (selected). Callout: 'When measuring to the rear face of the flange: enter the flange thickness here or use that of the selected flange'.
- Offset (Flange thickness):** Input field with '41.148 mm'. Callout: 'Determine the plane (front face of flange)'. A 'Copy Thickness of selected Flange' button is below.
- Matching Flanges:** A list box containing 'FLANGE WN, 8" ND, 300 LB, RF, ASME B16.5, ASTM A105'. Callout: 'Select from the appropriate flanges in the PSPM'.
- Rotation:** A button. Callout: 'Direction of the centreline will be reversed'.
- Swap Direction:** A button.
- Quality information:** A callout pointing to the 'maximum Deviation of plane: 217.34'.
- Plot the results:** A callout pointing to the 'OK' button, with a sub-callout showing a plot of a tie-in point with coordinates '-4912.5' and '89.72°' and the text 'and/or create it as plant object'.
- On [OK] retain the construction graphic (which points have been picked):** A callout pointing to the 'Keep defining graphic' checkbox.
- Determine the alignment of the bolt holes:** A callout pointing to the 'Rotation' button.

The command allows for several procedures. Depending on the shadows in the scan select a suitable possibility. The following variants are possible for determining the tie-in point data. It is initially assumed that the front face of the flange, or at least its front edge, is visible in the point cloud:

The pipe that is coaxial with the flange is easily recognisable



Select this button and click on two points somewhere along the pipe. The software fits a cylinder as precisely as possible to the pipe between the two picked points. The centreline of the cylinder is also the centreline of the tie-in point. The tie-in point must lie somewhere along this straight line.

This is usually the most precise method of determining the centreline, as a large number of points has been used - the whole cylinder between the two picked points are used for the fitting procedure.

Now the plane of the front face is determined by clicking the button depicted on the right. At least one point must be clicked on the front face, or if need be on the edge of the front face. Should more points be selected then an averaging process follows which gives a more precise result. Thereby all the data for the tie-in point has been calculated.



The cylindrical external face of the flange is easily recognisable



If there is no recognisable coaxial pipe, this button can be used. At least three well spread points should be picked on the lateral surface of the flange. The centreline has not yet been determined.

In the second step the plane of the front face of the flange is determined. Click on the button shown on the right.

Then click on at least three points. A plane is determined from these points. The software can together with the first three points (that project a circle onto the plane) calculate all the data for the tie-in point.



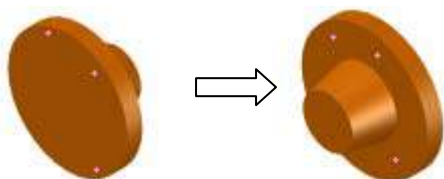
Die The edge of the flange is easily recognisable



In this method all of the data for the tie-in point are ascertained in the first step. After clicking on the button shown on the left, three points are picked that lay well spread out on the edge. These points are used both for the determining the plane and also for determining the lateral surface. This method is especially imprecise if only three points are used, because in laser scans the limitations of the system only allow for an inaccurate definition of edges.

The accuracy can, in all three cases, be assessed in the "Quality information" status field. Both figures are length information given in the AutoCAD linear units and should lie within the bounds of the desired accuracy. The distance shown is that of the worst point to the centre of the plane and to the centre of the circle. In some cases, quality values can only be calculated if more points are picked than above mentioned minimum number (and the number shown on the button icons).

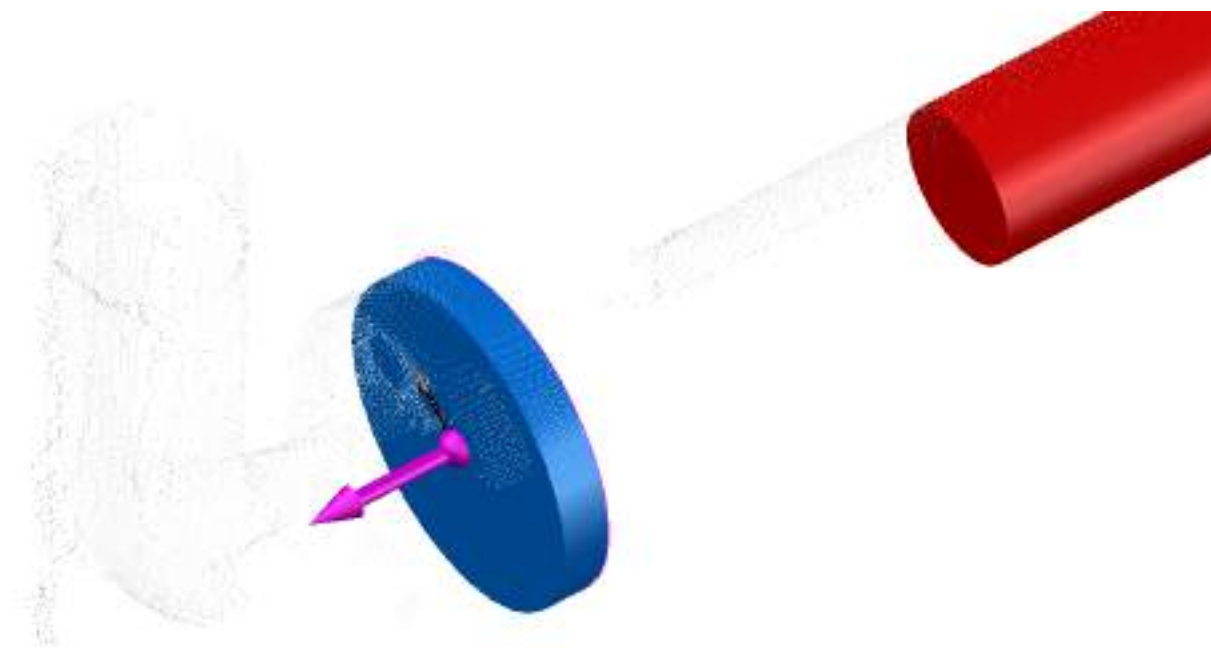
Often the front face of the flange in the point cloud is not discernible because it is covered by another flange or is in a shadow. In this case all variants can also be used on the rear face. Then it is necessary to additionally give the "Shift", the flange thickness. To illustrate the point, all icons on the buttons will be rotated when entering a shift value that is not equal to zero:



The flange thickness as a value for the move can also be automatically set using the button "Use the thickness of the selected flange".

For **better accuracy** measurements on the edges (rim of the flange) should be avoided. Consider using measurements on the rear face of the flange (with the additional shift information). Keep a watch on the quality information. Use more points than the absolute minimum. If there is a coaxial pipe, use it.


After the data for the tie-in point has been ascertained, a [Rotation angle] can be optionally entered. This describes the rotation of the flange by means of the position of the bolt holes.



The graphic, with the exception of the arrows, disappears when [OK] is clicked, the option "Received defining graphic" is then active. Textual information on the position and alignment of the tie-in point is added. Lines are to be found on the inside of the arrow that can be used for further construction work. If you need these then simply erase the arrow.

Or else use the possibility, to directly create a Plant object, which then, for example after exporting as an Autodesk Plant 3D component, use it as the starting point for some advanced modeling.

Export – Export

	Command Name:	PCEX
	Ribbon Tab:	As-Built Plant

This command is used after the creation of pipes using PCWTR (→Pipes – Walk The Run) and alignment using PCAC (→Pipes – ApplyConstraints) or after the creation of beams using PCFITBEAM (→Beams – Fit beam) and alignment using PCADJUSTSTEEL (→Beams – Adjust steel). It will be export objects or export files that will be created, depending which was selected from the available options in the following dialogue box:



All of the possible export possibilities for pipes and steel beams are offered in this dialogue box. For exporting as AutoCAD objects there are additional options that can be selected by ticking the appropriate box and can therefore be combined in many different ways.

After confirming using the [OK] button the dialogue window is closed and in the AutoCAD command line the following prompt appears:

Select plant objects:

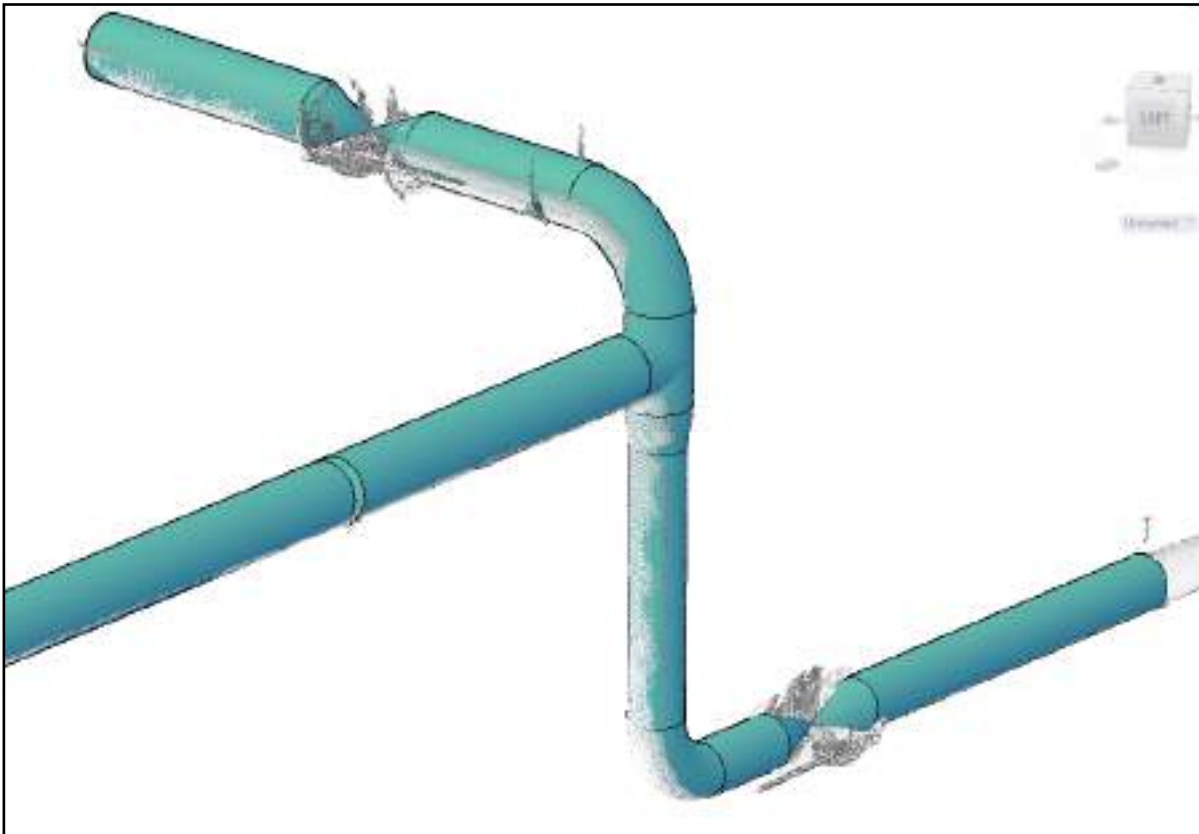
For steel beams all of the objects that are to be exported must be selected. For pipes it is sufficient to select just a single component of a connected section, as the whole of the associated network will be automatically recognised. When you are finished select all of the desired objects, press the enter key and in the command line you will see:

Review all connected objects [Continue] <Continue>:

All of the Plant objects, that are connected to the objects that you selected, will be simultaneously highlighted in green, so that you can visually check if the resulting network is as you intended. If this is the case, confirm the selection by pressing the enter key to start the export function that you defined.

Notes on exporting pipes:

Next situation could exist:



Option AutoCAD Objects, Export to tagging objects

The export as tagging objects needs a structure definition in the drawing. If there is no one you get a dialog for selection. Choose a structure definition with the extension "export" and run the export command again.

The predefined structure definitions contain following classes:

Line: group of all items of a pipe run until next branch

PlantObject: base class for certain types as elbow, pipe etc.

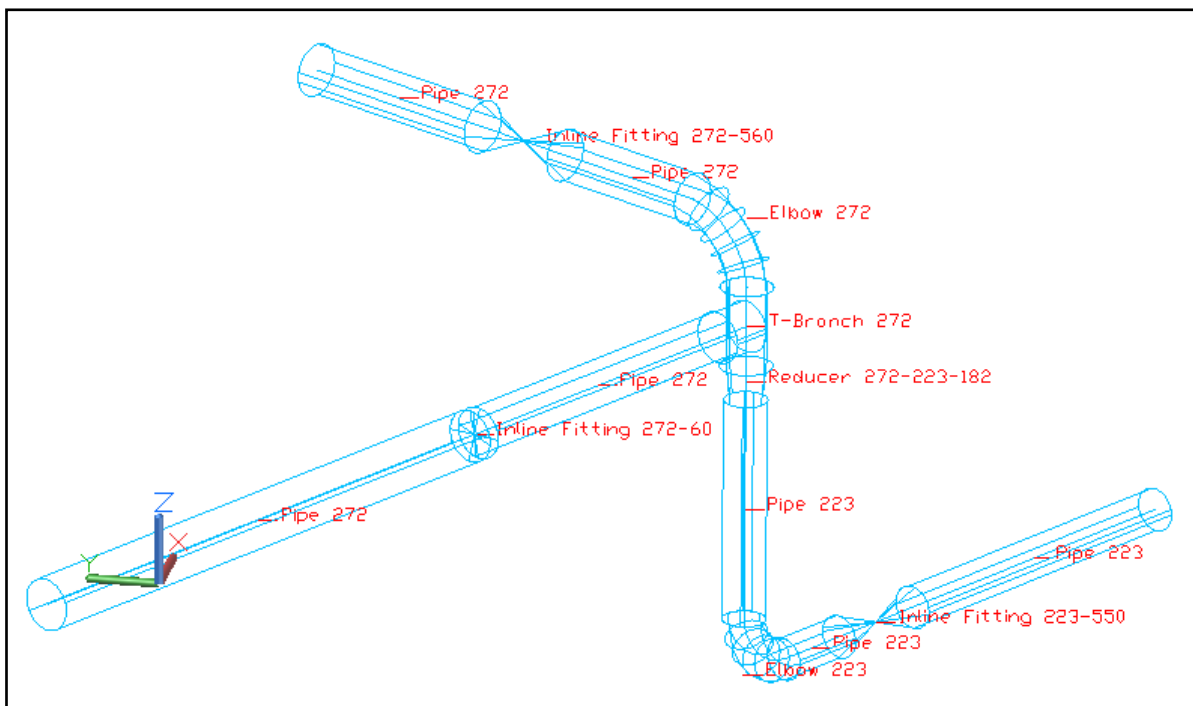
Pipe: type pipe

Elbow: type elbow

InlineFitting: type inline fitting

Branch: type Branch

ConnectionPoint: common point at two entities are connected via their connection points



The export result is also confirmed in the command line:

Export (tagging) has created 16 objects.

For more detailed information please refer →**Tagging in 3D space**.

Option AutoCAD Objects, Export names to text

Option AutoCAD Objects, Export as polylines (simple)

Option AutoCAD Objects, Export the start points

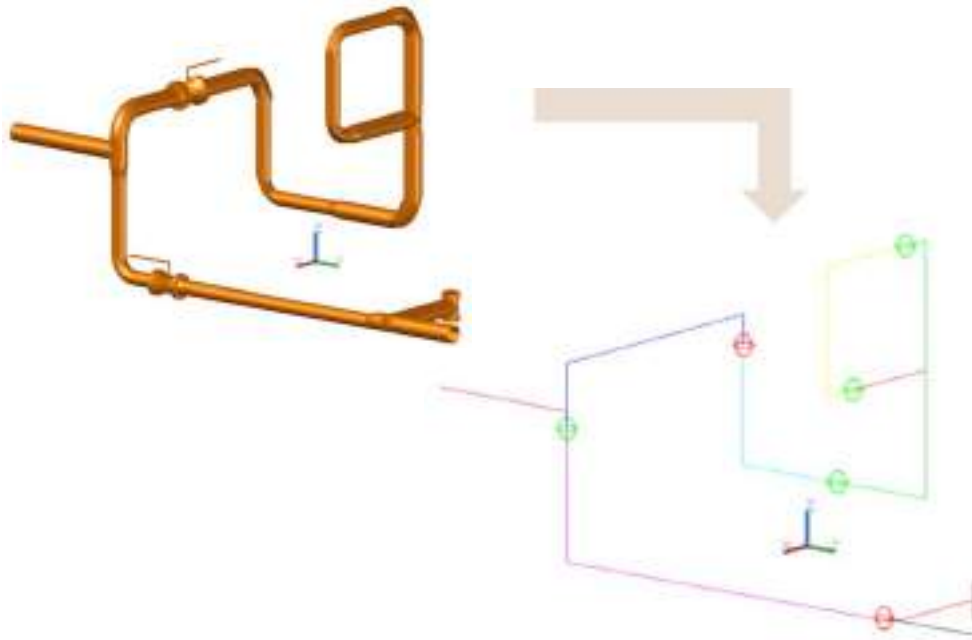
These three options may be used as single option as well as in all possible combinations.

By combining all three of these options, the results are 3D polylines which represent the axes lines plus AutoCAD text which contains the object names.

The polylines run from branch to branch or end at free pipe ends.

The texts have their insertion points on the axis and in the middle of the labelled objects.

Start points are placed at start and end points of all objects (like valves, flanges and elbows).



This special export function facilitates, by using a special workflow, to relatively easily “trace” the pipeline from the resulting elements in AutoCAD Plant 3D or other applications. Commands which convert centerlines to pipe runs can be applied on the content of a entire layer because on this layer it will find only axis of a specific nominal size!

In this way you can also in Plant 3D, for example, set pipe bend connectors into the pipe run. Furthermore it is recommended not to first place valves and other in-line fittings in this way when walking the run with As-Built Plant (→Pipes – Walk The Run), but later to first set them in the target application.

Option AutoCAD Objects, Export as 3D solids

The As-Built plant objects are exported as AutoCAD solids. The plant objects will be preserved.

The diameter of the new solids contains an eventual insulation.

The confirmation you get via command line:

```
Export (solids) has created 16 objects.
```

Hint: If you want to export only single As-Built plant objects to AutoCAD solids just use the EXPLODE command.

Option Plant 3D (AutoCAD Plant 3D 2013 and above)

If you work with AutoCAD Plant 3D (version 2013 and above) you can export the As-Built plant objects as Plant 3D objects. The As-Built plant objects will be preserved.

For a successful export please observe the following:

- The used Plant Model (PSPM) has to be imported from a AutoCAD Plant 3D spec.
- The catalog name has to correspond to the spec name.
- When you run the command PCAC use the strategy 2. This way you maintain the defined geometry of elbows and Tees, Thus a correctly linked Plant 3D model can be generated from the exported components..
- If you use flanged fittings like valves you do not have to create the flange at the pipe side. The export joints the components according to the joint types defined in the project setup. That means gasket and bolt set will be created automatically.
- For the automatic connection between two flanges you have to place both flanges with a distance equal the thickness of the suitable gasket in the specification. It is easier to place only one flange. The counterpart will then be created automatically by Plant 3D.

The confirmation you get via command line:

```
Export (Plant 3D) has created 16 objects.
```

Option Revit MEP:

For this option you must have first created a As-Built plant model from a pipe classes file in As-Built for Revit (→Prepare – Edit Plant Model, “Import plant model...”) button. The model so created can then be exported to a text file and from that the pipe run can be imported into Revit MEP with As-Built Revit.

Option generic:

This option is for working in a special file format, that has been developed for external developers. Thereby the following workflow should be possible for any pipeline software, even when they don't support any point clouds:

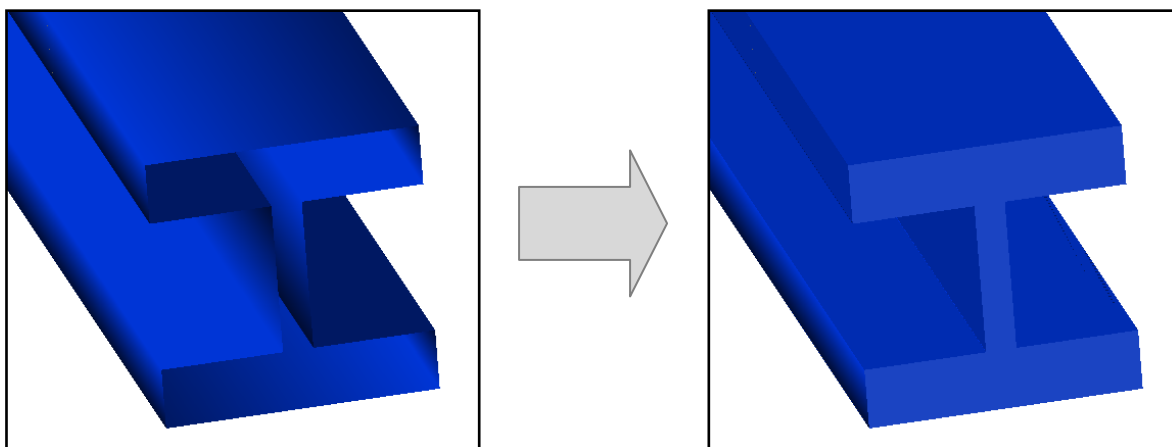
External software → *.PMD file (Plant Model Definition) → As-Built plant model

Such a *.PMD file can then be imported into As-Built Plant (→ Prepare – Edit Plant Model, “Import plant model...” button). After walking the run and using Apply Constraints comes the export using the “Option generic”:

Command PCEX, → *.PMI file (Plant Model Instances) → Import in external software

Notes on exporting beams:**Option AutoCAD Objects, Export as 3D solids**

This option serves to convert proprietary As-Built beams (kubitPlantBeam created using PCFITBEAM, → Beams – Fit beam) into 3D solids. The standard AutoCAD objects so created can, for example, using a DWG or IFC interface, be sent to other software programs or be further processed with the on-board resources of AutoCAD.

**Option AutoCAD Objects, Export as axis lines:**

Axis lines will be created from the start to the end of the beam. The layer where the line entities will be placed is configurable (→ Prepare – Settings).

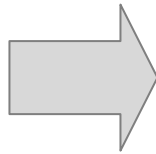
Option AutoCAD objects, export as annotation text:

An AutoCAD text entity with the name of the beam will be inserted on the centre point of the beam axis. The layer where the text entities will be placed is configurable (→ Prepare – Settings).

Option SDNF

A SDNF file will be created from the As-Built steel beams, that can be read by many steelwork software applications (for example Autodesk Revit, but also by other products such as INTERGRAPH and AVEVA).

SDNF stands for **S**teel **D**etailing **N**eutral **F**ormat.



```

mySdnf - Editor
-----
Datei Bearbeiten Format Ansicht |
-----
#*
#* SDNF VERSION 2.0
#*
#* COMPANY: kubit Software
#*
#*
#
Packet 00
"Name of Engineering Firm"
"Name of Client"
"Name of Structure"
"Name of Project"
"2/19/2014" "10:47"
| "Issue Code"
"Design Code"
0
#
Packet 10
"millimeters" 3
| | "beam" "" 1

```

Option Advance Steel

This option can only be used if the extensions “Advance Steel and Advance Concrete Plug-in for Revit 2015” are also installed. Depending on the version of the plug-ins either a *.GTC or *.GMXL will be created, which will allow the import of the adapted and aligned beams into AdvanceSteel or Revit.

Please note that the export only works correctly if you have imported Advance Steel catalogs (see →Prepare – Edit Plant Model S.266).

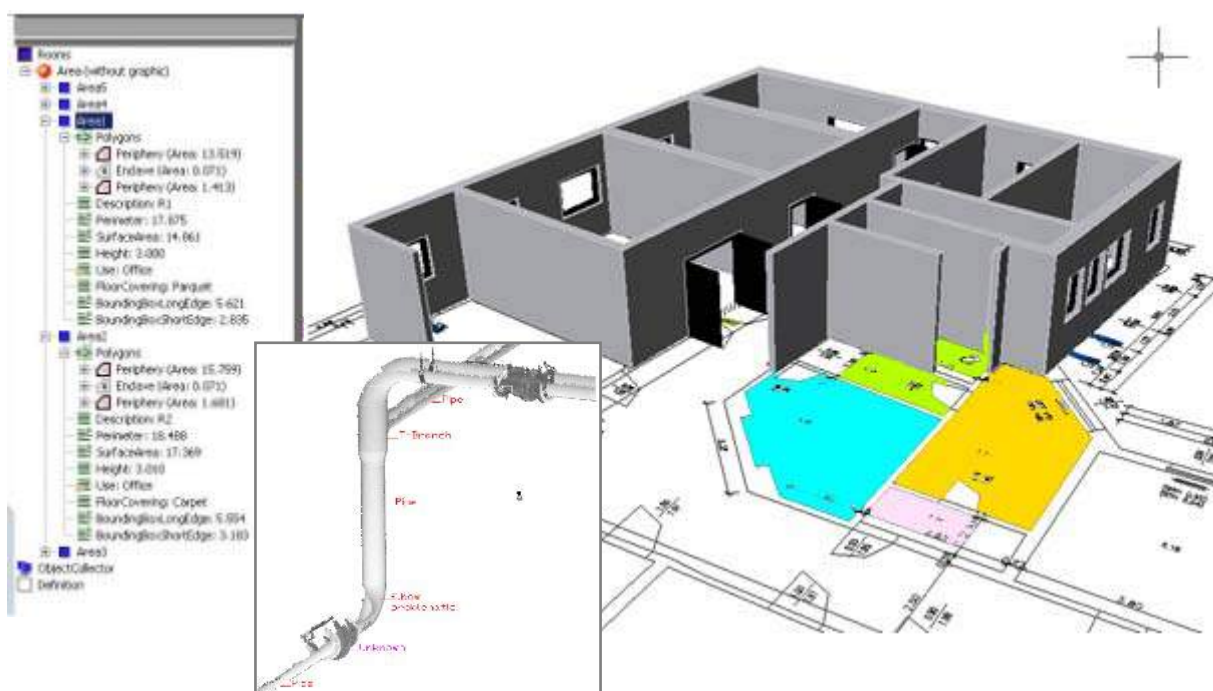
10.4 As-Built Feature Data Management

Feature Data Management

	Ribbon Tab:	As-Built Feature Data

The **Feature Data Management** of As-Built for AutoCAD contains a large set of commands. It serves for appending additional alphanumeric data to your drawing. To each graphic object project-specific feature data or links to further documents (e.g. pictures) can be assigned. These will be displayed in a clearly arranged data tree and can be processed further.


The Feature Data Management can be used for the highly efficient recording and structuring of building floor plans. With the aim of creating qualified CAFM⁴ suitable data structures. Components and pipelines in industrial plants can be labeled and managed or a green space and tree cadastre can be created.



Further information about the feature data management of As-Built for AutoCAD can be found in the [manual for the Feature Data Management](#), which is part of your installation. You may find a link on the As-Built Feature Data ribbon tab under “Help”.

⁴ Computer Aided Facility Management

10.5 As-Built Building Plans

Walls – Align walls		
	Command Name	PCALIGNWALLS
	Ribbon Tab	As-Built Building Plans

This command serves to automatically align and axially align existing (wall) lines. The aim is the **creation of architectural type plans**, that, where possible, have:

- Wall lines that are parallel to each other
- Rooms that have square corners
- Rooms/walls that are axially aligned

When surveying existing building data it is difficult to achieve this aim without the help of additional means. It often results in only approximate right angles and almost parallel lines. It is here where the algorithm of this new command comes in to play:

The lines and polylines selected by the user are analysed in their entirety with respect to similarly aligned lines and approximate axial alignments. Reference directions are computed from numerous other similar directions and axial alignments, to which the lines are then oriented.

Workflow

“Align Walls” is carried out before the wall lines are given any further building elements such as doors, windows, blocks etc.

1. Read in the point cloud and prepare it (clipping, define slices ...)
2. Create wall lines (e.g. with “Fit outline plan”)
- 3. Align Walls**
4. Create other building elements (e.g. doors, windows, stairs, equipment blocks ...)
5. Dimension / annotate / complete the plan

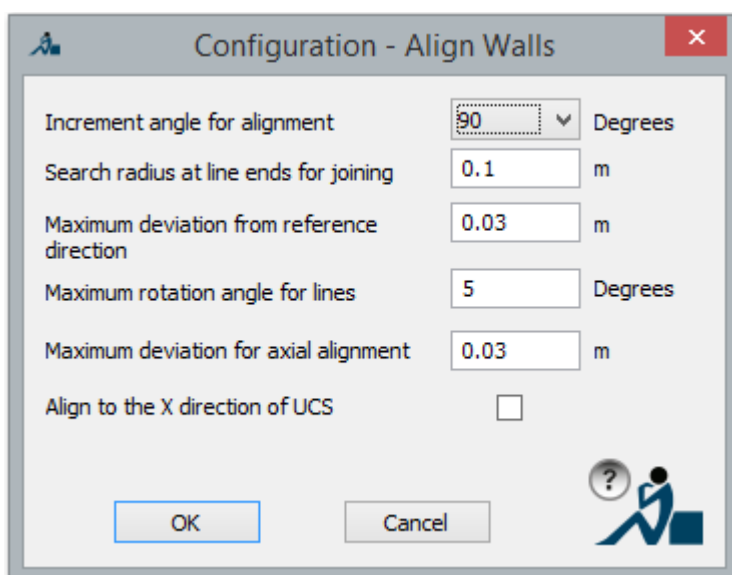
Command sequence

After starting the command the wall lines must be as first selected. Allowed at the moment are lines and polylines without any curved elements.

Command: **_KUBT_ALIGNWALLS**

Select wall lines and polylines for fitting:

Then the settings dialogue box is displayed:



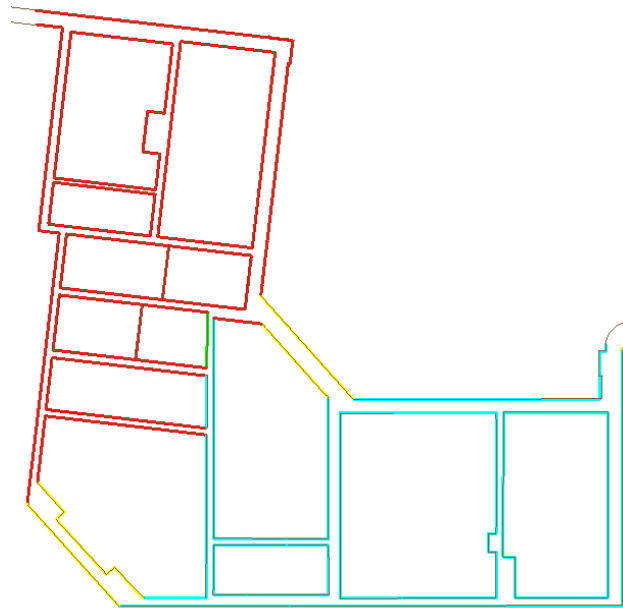
- **Incremental angle for the reference direction:** In which angular steps should the walls be aligned? The values 90° (default), 45°, 22,5° and 30° can be used.
- **Search radius at line ends for joining:** When aligning walls, the existing line connections are first disconnected from one another. The command automatically reconnects the ends of the line sections, when the distance between the end points of the lines lays within this radius.
- **Maximum deviation from reference direction:** Wall lines are only aligned when the distance from the original line does not exceed this value.
- **Maximum rotation angle for lines:** Wall lines are only aligned when the rotation angle does not exceed this value.
- **Maximum deviation for axial alignment:** Wall lines are only axially aligned with each other when the distance from the original line does not exceed this value. Normally this value should be set somewhat higher than “Maximum deviation from the reference direction”, so that the directionally fitted walls can still be axially aligned.
- **Align to the X direction of the UCS:** This setting should be activated when the X axis of the current UCS is to be used as the reference direction, that means the plan must be aligned on the UCS. If it is not activated, the command computes the reference direction itself, based on the selected wall lines.

Confirm with [OK] when you are finished.

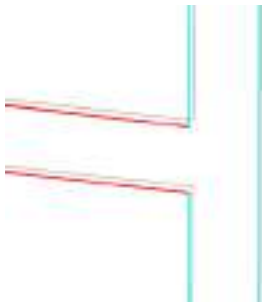
Now you get a preview of the aligned lines.

The aligned wall lines are displayed coloured next to the original wall lines. Every reference direction used is assigned its own colour. It is thereby indicated which lines have been aligned with each other and those which have not.

Lines with the same colour belong to the same reference system and have thereby the same alignment. In the adjacent example three reference directions have been defined.



On closer inspection it can be seen that:



The thin lines are original walls, the coloured lines are the aligned walls.

With the option "**Keep**" you accept the aligned lines. You then replace the original lines. The colours disappear. The option "**Discard**" returns you to the original state. The settings dialogue box is re-opened and if necessary you can start a new attempt with different parameters.

The procedure in detail

To be able to better understand the results and the settings the procedure will be described in more detail in the following:


The command has three steps in its procedure:

1. **Rotate** all of the selected wall lines 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 the reference direction" which on the one hand, 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. The reference direction is calculated by the algorithm based on the lines selected. It could be several.
2. **Axial alignment = move** all wall lines, when the amount of the move lies within the set tolerance value "Maximum deviation for axial alignment". It is usual to set

to a higher value than the first parameter "Max. deviation for reference directions", thereby the rotated walls can also be axially aligned.

3. **Fix** wall lines with each other: When their end points lay close enough to each other, that means within the set radius ("Search radius at line ends for joining"), then they will be trimmed or extended with each other, thereby the result is once again the creation of a closed floor plan.

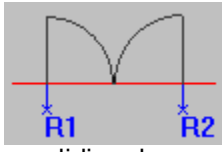
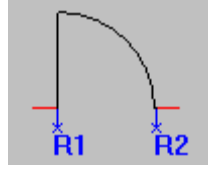
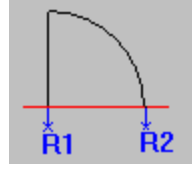
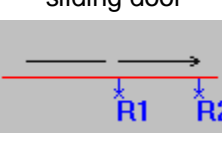
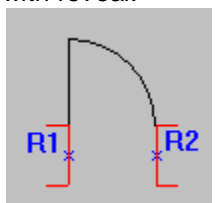
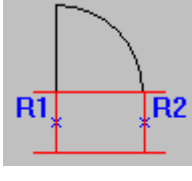
Building Elements – Door

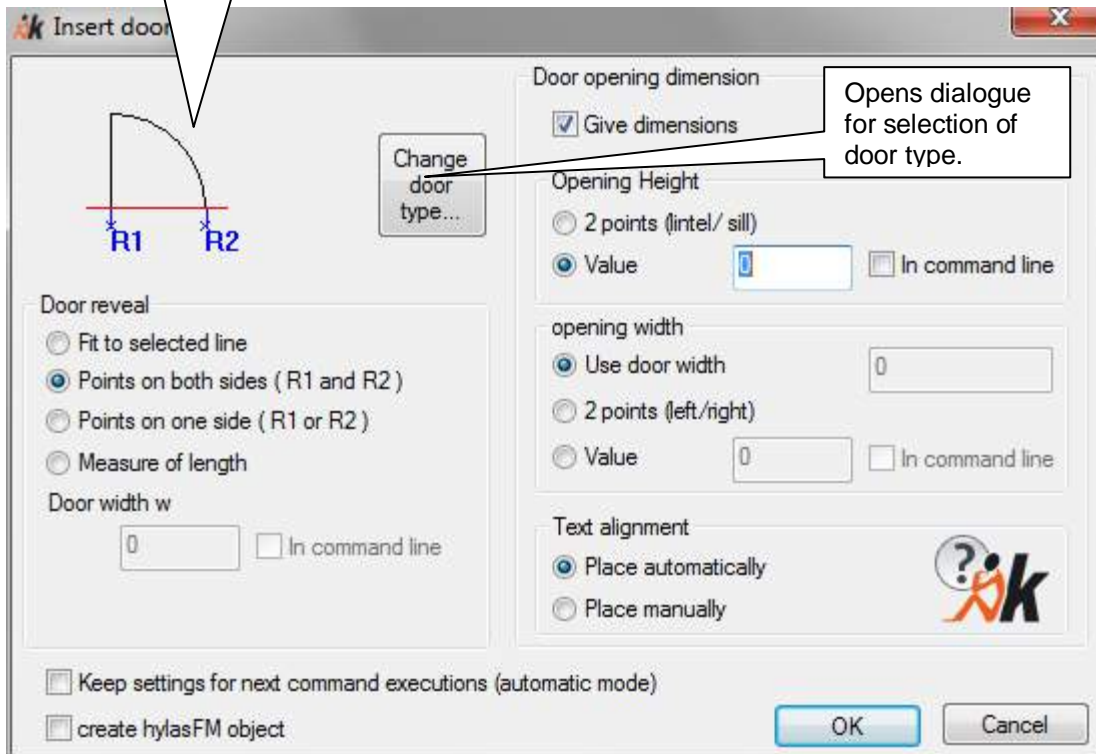
	Command Name:	PSBINSERTDOORS
	Ribbon Tab:	As-Built Building Plans

The command draws a door symbol into an existing wall line and annotates it optionally with height and width of the opening.

This command contains options for point measurement with a total station, options for clicking in a point cloud, as good as options for distance measurement.

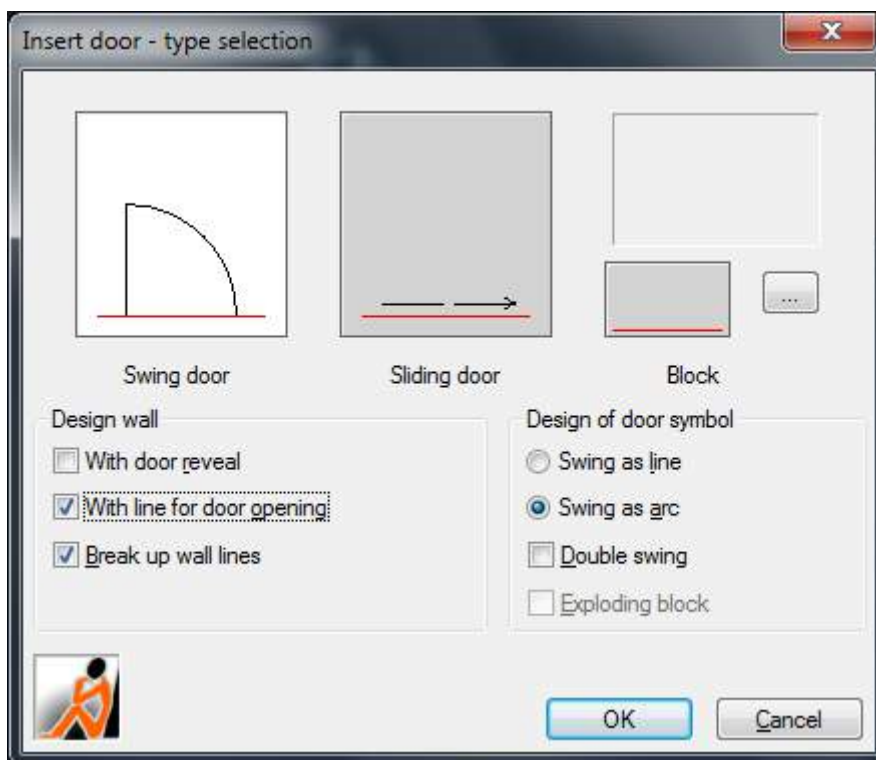
The following dialogue opens:

<p>Depending on the selected door type the preview looks different:</p>	<p>Example single door: Simple door symbol:</p>	<p>With threshold:</p>
<p>double door</p> 		
<p>sliding door</p> 	<p>with reveal:</p> 	<p>With reveal and threshold:</p> 



Opens dialogue for selection of door type.

The button **Change door type...** opens a selection dialogue:



By clicking an image you select a type (white background). You can choose between swing doors and sliding doors but you also have the chance to select a door symbol you defined yourself (AutoCAD block). For the latter you have to design the symbol beforehand and save it in a separate drawing. With the button "...“ you select this block template.

In the frame **Design Wall** you may set if and how the wall lines shall appear when inserting the door. You can decide if threshold and reveal shall be drawn as well or if you only want to draw the door symbol and if the wall lines shall be broken up or not.

In the frame **Design of door symbol** you can choose between different designs for the door symbol and you can state if it shall be a double door. Depending on your selection the sketches will show you what the result will look like. If you selected "Block" as door type you can decide if it shall be exploded subsequently.

Confirm your selection with **OK**. You will return to the dialogue "Insert door". (see figure above). The preview shows you the elements to be measured. Now you can make further settings for prompting and input of items to be measured, e.g. **door reveal** and **opening dimensions**.

Door reveal

- Option **Fit to selected line**: The reveals of the door are defined by the selected end points of the wall line. In case that the wall line has been determined by two points only this option is usable. After closing the dialogue only the orientation of the door symbol is prompted in the command line:

Select wall line or [two Points] <Points>:

(Click the line)

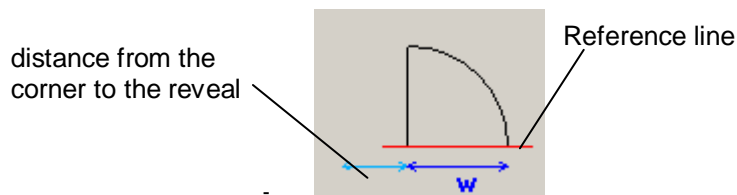
Select point for direction of door swing or [Undo]:

(Click point for opening direction)

- Option **Points on both sides (R1 and R2)**: If you are able to pick both reveals of the door from the point cloud choose this option. After closing the dialogue the following is prompted in the command line:

Select wall line or [two Points] <Points>:	<i>(Click the line)</i>
Select point on the reveal or [Undo]:	<i>(Click point on first reveal)</i>
Select second point on the reveal or [Undo]:	<i>(Click point on second reveal)</i>
Select point for direction of door swing or [Undo]:	<i>(Click point for opening direction)</i>
- Option **Point on one side (R1 or R2)**: If you are able to pick only one reveal of the door from the point cloud you have to enter an additional value **w** for the width of the door. You can enter the width directly in the dialogue or you activate the option "In command line" and you will be prompted for it later. After closing the dialogue the following is prompted in the command line:

Select wall line or [two Points] <Points>:	<i>(Click the line)</i>
Select point on the reveal or [Undo]:	<i>(Click point on reveal)</i>
Enter value for door width or [Undo]: 1.1	<i>(Enter the value)</i>
Select side or [Undo]:	<i>(Click side for opening direction)</i>
Select point for direction of door swing or [Undo]:	<i>(Click point for opening direction)</i>
- Option **Measure of length**: Here you do not have to pick any point. You can enter the width **w** of the door directly in the dialogue or you activate the option "In command line" and you will be prompted for it later. Additionally you will be prompted for the distance from the corner to the reveal. This means the length measure from the end of the selected reference line. It is important to pick the line on that end from which you measured this length.



After closing the dialogue the following is prompted in the command line:

Select wall line or [two Points] <Points>:	<i>(Click the reference line)</i>
Enter distance from the corner to the reveal [Undo]: 0.57	<i>(Enter measured distance)</i>
Enter value for door width or [Undo]: 1.3	<i>(Enter the value)</i>
Select point for direction of door swing or [Undo]:	<i>(Click point for opening direction)</i>

Door opening dimensions

The dimensions of the opening of the door is optional. It is activated by ticking "**Give dimensions**".

You can either enter the opening dimensions manually or pick two points.

- **2 Points (lintel/sill) respectively (left/right)**: You will be asked in the command line for two points which represent the opening measurement. The distance of the points in Z direction is used for the **opening height**, while the distance of the points in the XY plane is used for the **opening width**.

Measure point on the threshold or [Undo]:	<i>(Click point on lower edge of the door)</i>
Measure point on the soffit or [Undo]:	<i>(Click point on upper edge of the door)</i>
Height of opening: 2.34216	<i>(calculated door height)</i>
Select first point for opening width or [Undo]:	<i>(Point on one side edge of the door)</i>
Select second point for opening width or [Undo]:	<i>(Point on other side edge of the door)</i>

Width of opening: 0.90246

(calculated door width)

- **Value:** This option is used for directly entering the measurement. You can either enter it in the dialogue or it will be asked in the command line later if “In command line” is activated.

Enter value for opening height or [Undo]: **2.3**

*(enter measured value
or overtake it from hand-held laser box)*

Enter value for opening width or [Undo]: **0.9**

*(enter measured value
or overtake it from hand-held laser box)*

- **Use door width:** An additional option for the opening width is to adopt it from the measured door width.
- Finally you can additionally set if the dimensioning shall be placed **automatically** or **manually**.

Automatic mode

The automatic mode is activated by ticking “**Keep settings for next command executions (automatic mode)**”. This option is intended to insert doors of the same kind without having to make all settings anew every single time. The next time this command is called up, the dialogue will not be displayed. The settings will be adopted as they had been made the last time. We recommend to use “In command line” for entering the values (e.g. door width). If needed the automatic mode can be switched off by a command line option.

After having done all settings you close the dialogue with **OK**.

Next you are prompted to enter a line or two points where the door symbol shall be placed. If you select “two points” the door is placed matching between those. If you select “line” you will be prompted for the position and width of the door.

In case you selected a symbol for a door with reveal you will be asked in the command line for a point or a line for the second wall line or for the wall thickness.

Enter thickness of wall or [Line/Position/Undo]<Line>:

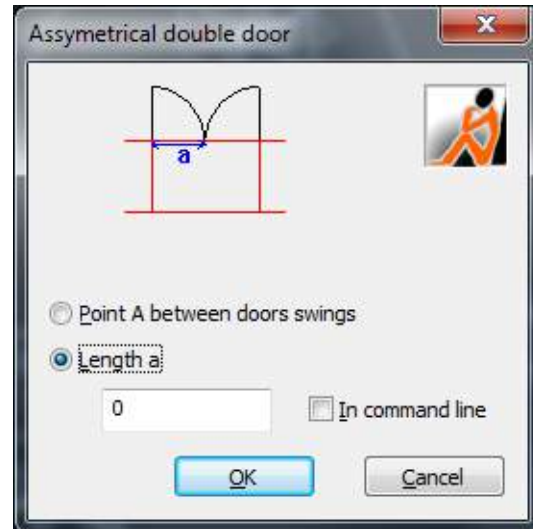
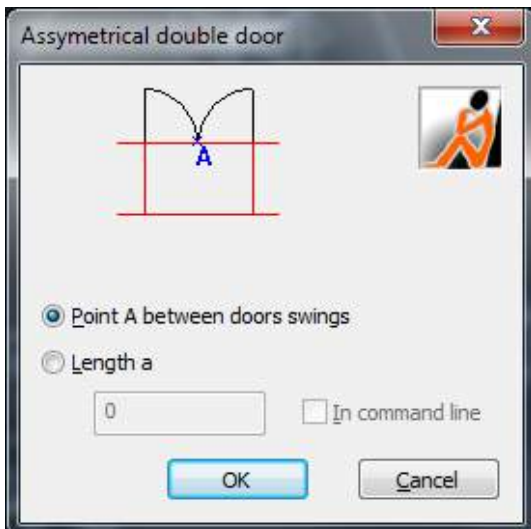
(Enter the thickness of the wall!)

Now the values and points are prompted according to the settings in the dialogue.

The door symbol is inserted at the height of the first point of the reveal. It is by standard put on the layer DOOR. It lies parallel to the XY plane of the UCS. The reference line is broken up at the reveal. The dimensioning block is by standard put on the layer DOORDIMENSIONING. The door reveal as well can be put on a specified layer.

You can adapt the default layers for the individual door elements with the command **→As-Built – Settings**.

If the door type double swing has been selected, the window “Insert door“ offers the additional option “Asymmetrical swings“. If this option is selected and the button **Measurements** is clicked, a window opens where one can choose if the asymmetry shall be determined by picking a point or entering a value for the length.




Create feature data objects

This option can only be applied, when you use the FARO feature data management. If the used structure definition has a predefined class "Door", a feature data door object will automatically be generated in addition. The attributes "Height" and "Width" will automatically be filled with the opening measurements.

Hint: If you use the AutoCAD function „Geometric constraints“, in some cases the wall lines may not broken automatically.

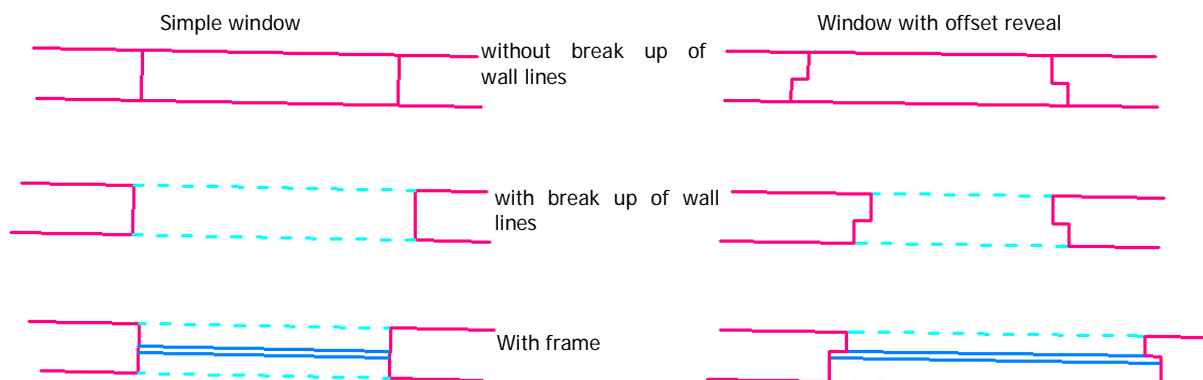
Building Elements – Window

	Command Name:	PSBINSERTWINDOW
	Ribbon Tab:	As-Built Building Plans

The command inserts a window symbol into an existing wall line.

This command contains options for point measurement with a total station, options for clicking in a point cloud, as good as options for distance measurement.

The command supports a certain set of default representations of windows:



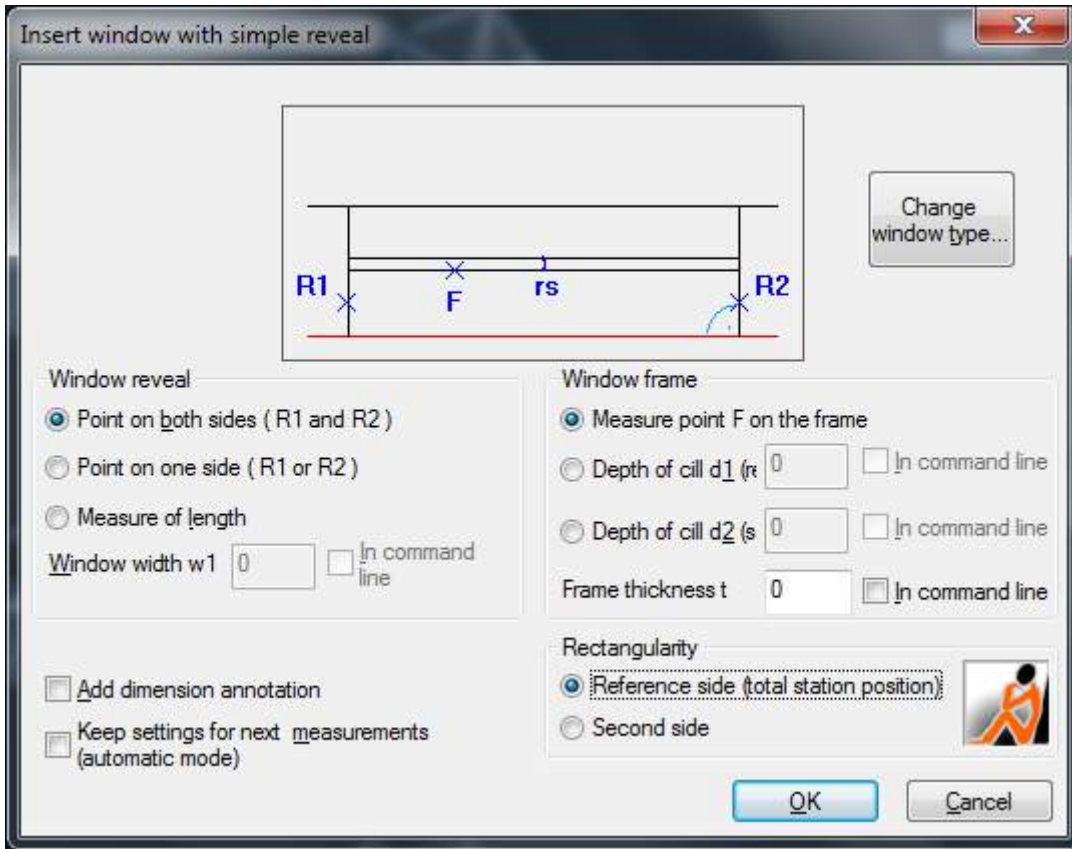
General rules for FARO windows

- A reference line has to be set. The **reference side** is that side of the wall from which the window is measured. (Total Station interface: The side where you stand with your instrument.) For the reference side more values are measured (e.g. two points of the reveal) than for the opposite side (just one point of the reveal).
- The window is perpendicular to one of the two sides of the wall. You may choose the side.
- The window frame is always placed directly at the window rabbet.
- The window lines will automatically be placed on different layers
 - The window reveal is by default put on the layer WINDOWREVEAL.
 - The reference line is interrupted at the reveal, and the intermediate part is assigned to the layer WINDOWOPENING.
 - The lines symbolising the window frame are by default put on the layer WINDOWFRAME.

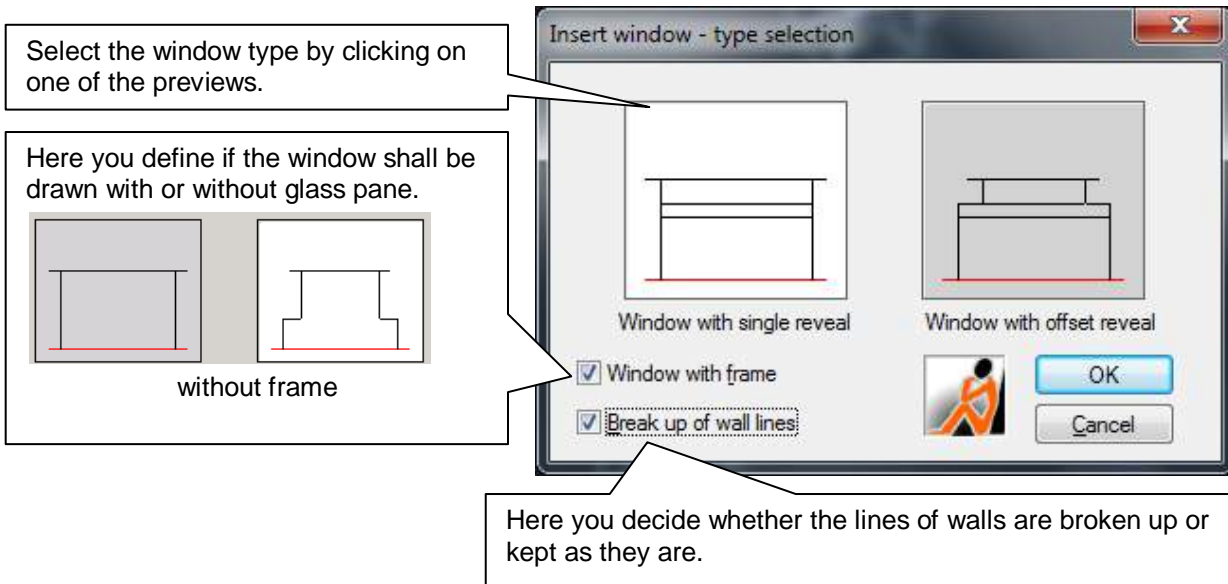
You can adapt the window elements to your individual demands with the command **→As-Built – Settings ...**

Workflow

6. When running the command the following dialogue opens:



Here you set how the window is measured/inserted. The several possibilities are explained later on. With the button **Change window type** you can select the **representation of the window**:



Confirm your selection with **OK**. You get back to the main dialogue.

7. Next you are prompted in the command line to select the lines of the wall where you want to insert the symbol for the window. The second line of the wall can optionally be defined by:

- Specification of the thickness of the wall
- Clicking a point
- Specification of a second existing line

Select reference line (inner wall):

Enter wall thickness or [Line/Position/Undo] <Line>: .24

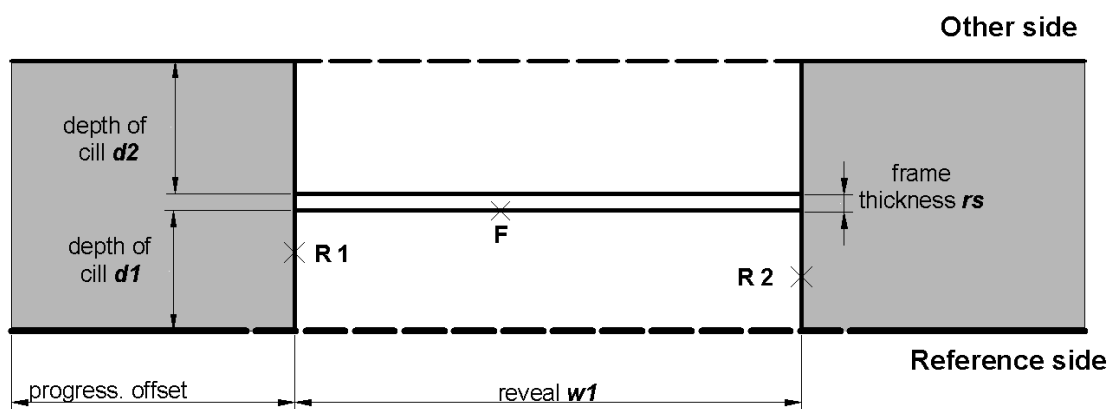
(Click line or polyline for reference wall)

*(Enter value for line thickness
or option 'P' for 'Point on other side of wall'
or option 'L' for 'Line for other side of wall')*

8. Following the values and points are prompted according to the settings in the dialogue.

Simple Window

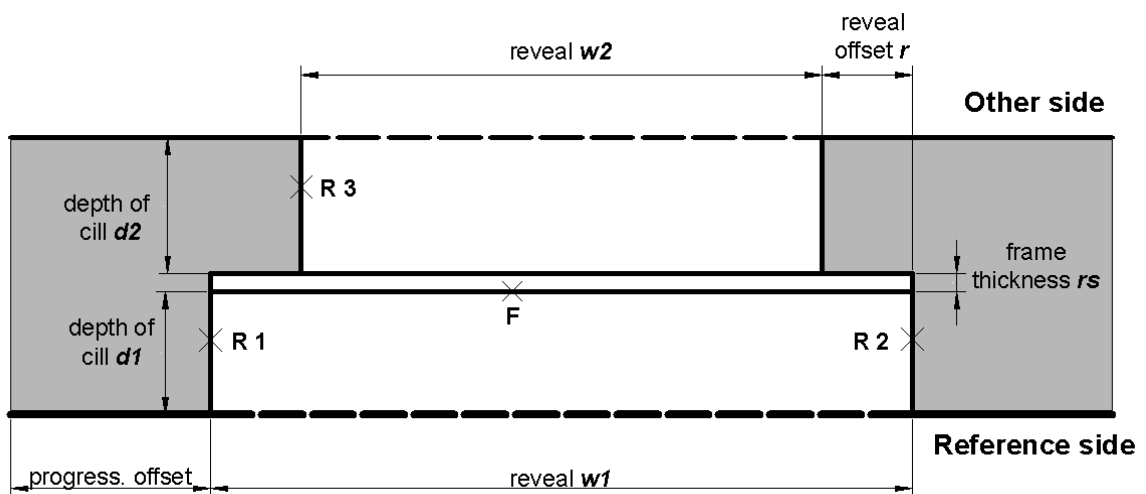
The window is assumed to be symmetric. It has no rabbet and the reveal is rectangular to one of the wall lines.



The reveals are parallel to each other.

Window with offset reveal

The window is assumed to be symmetric. The following measurement options are available:



The window with offset reveal has two different reveal widths. The rabbet is rectangular. The measurement of the reveal on the reference side is similar to the simple window.

The particular elements to be measured:

Window reveal (reference side)

This setting is valid for all window types.

- **Point on both sides:** If you can pick both sides of the reveal from the scan, select this option. After finishing the dialogue, following queries appear in the command line:

Select reveal point (R1) or [UNdo] *(Click point Ri1 on one reveal side)*

Select second reveal point (R2) or [UNdo]: *(Click point Ri2 on second reveal side)*

- **Point on one side:** If you can pick only one side of the window reveal from the scan, you have to enter a measurement $w1$ for the window width. You can either enter this measurement in the dialogue or you activate '**In command line**' and it will be asked in the command line later. After the dialogue has been closed, following command line sequence will appear:

Select reveal point (R1) or [UNdo]: *(Click point Ri 1 or Ri2 on reveal side)*

Enter value for width of the window ($w1$) or [UNdo]: **1.4**
(Enter measured value for window width wi)

Select side or [UNdo]: *(Select point for the side of the window)*

- **Measure of length:** Here you do not have to pick any points at all. You can either enter the window width $w1$ in the dialogue or you activate '**In command line**' and the measurement will be prompted in the command line later. You will also be asked for a progressive distance in the command line. This means that you have to enter a distance measure from the end of the reference line. It is important to pick the reference line on the side from which the measures have been taken:

Enter distance from the corner to the reveal [UNdo]: **0.57**
(enter measured progressive distance)

Enter value for width of the window ($w1$) or [UNdo]: **1.4**
(enter value for window width wi)

Window reveal (second side) (only 'window with offset reveal')

This settings determines how much the window opening is tapered or widened by the reveal. It is assumed that the window is symmetric. There are three possibilities:

- **Point R3 on one side:** Click a point on the reveal behind the rabbet. After the dialogue has been closed, following query appears in the command line:

Select reveal point second side (R3) or [UNdo]:

- **Reveal offset r:** If you cannot pick a point on the reveal, enter a manually measured value for the offset. It is only possible to enter positive values for the reveal offset. For a widening of the window opening to the "other side", select the "to widen" option and for a tapering select the "to narrow" option. If you have stated the reveal offset within the command line you will be prompted:

Enter value for reveal offset (r) or [UNdo]:
Enter measured or gripped value for the reveal value r.)

- **Window width w2:** Alternatively, you can also measure and enter the second reveal width by hand.

Window frame

The settings determine how deep the reveal and the frame sit in the opening. There are three ways to determine the depth:

- **Point F on window frame:** If you can pick the front of the reveal directly, choose this option. If you have selected a window type 'with frame' you will be prompted to pick the front of the frame. After closing the dialogue, following prompt appears in the command line:

Select point F on the frame (F) or [Undo]:

- **Depth of cill d1 (reference side) or d2 (second side) :** If you cannot pick a point, enter a value measured manually. After closing the dialogue, following query appears in the command line:

Enter value for cill depth (d1) or [Undo]:.12

- **Frame thickness t:** This item is only shown if you have selected a window type 'with frame'. It defines the distance between the two lines representing the glass pane. If you enter Zero only one line is drawn.



Rectangularity

This setting decides, to which wall side the window will be perpendicular. You can choose between "Reference side" and "Second side". If both wall lines are parallel to each other, this setting makes no difference.


Automatic mode

The automatic mode is activated by ticking the field 'Repeat these settings for next command executions'. This option is intended for inserting several windows of the same kind without having to make all settings anew every single time. The next time the command is called up, this dialogue will not appear. We recommend using the option 'in command line' for entering values (e.g. reveal width).

Window dimensioning

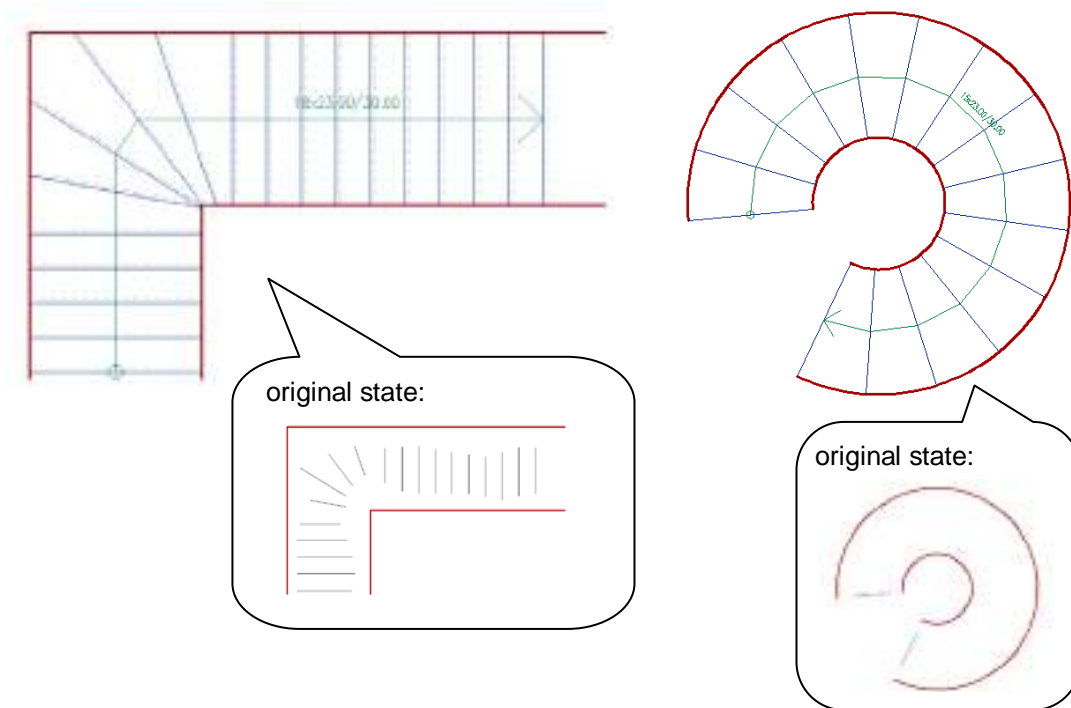
If you select the field 'Add dimension annotation', the command →Dimension – Window dimension annotation is called up after this command.

Building Elements – Staircase

	Command Name:	PSBINSERTSTAIR
	Ribbon Tab:	As-Built Building Plans

The command enables you to generate a staircase as a floor plan illustration.

It contains options for point measurement with a total station, options for clicking in a point cloud, as good as options for distance measurement.



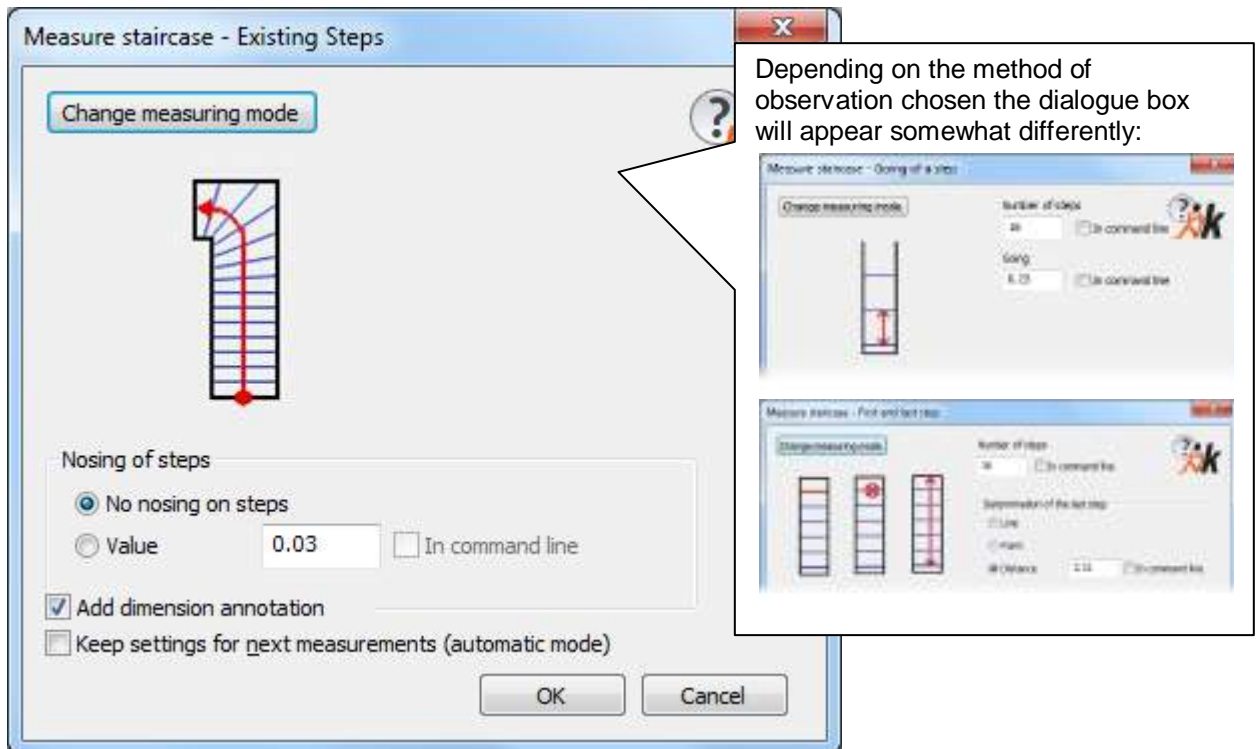
Requirements

Before this command can be executed, several graphic elements of the staircase have to exist already.

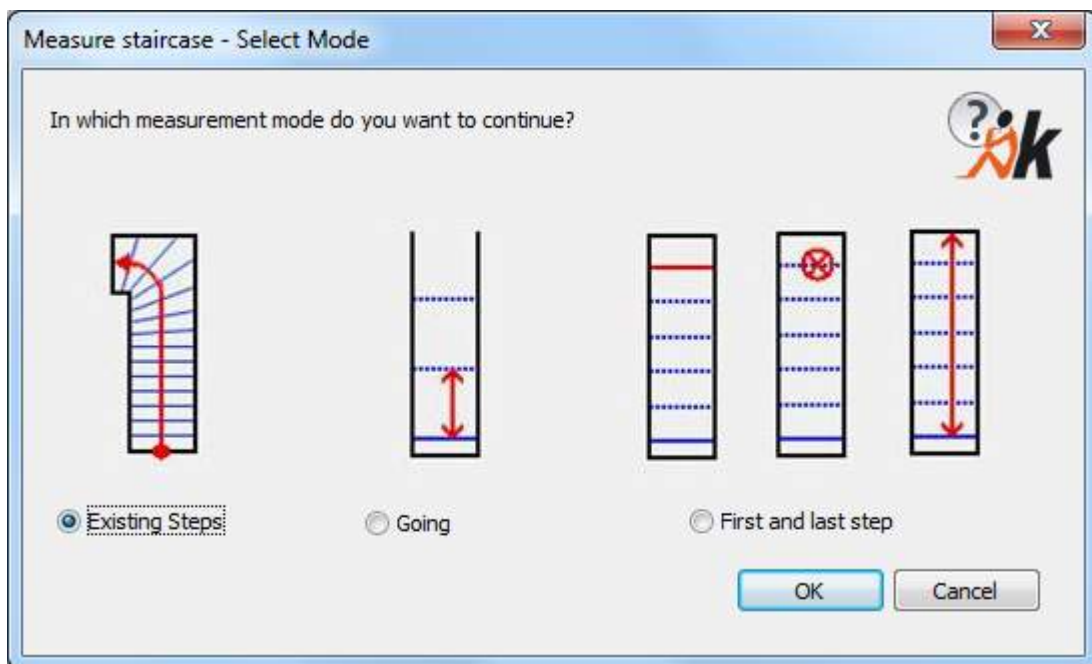
- **The edge of the staircase:** The boundary line to the right and the left of the steps may consist of a mass of lines, polylines and arcs.
- **The steps:** These have only to be available as rough measurement, which means they do not have to reach the edge of the staircase or they may exceed the staircase. Not all step lines have to be stated; optionally As-Built may calculate the steps. All step lines have to be within the edges of the staircase (stair string). Steps may be lines or one-segment polylines (without arc).

Command sequence

After starting the command the following dialogue box opens up:



The button [Change measuring mode...] opens a selection dialogue box:



You can choose one of three methods of observation:

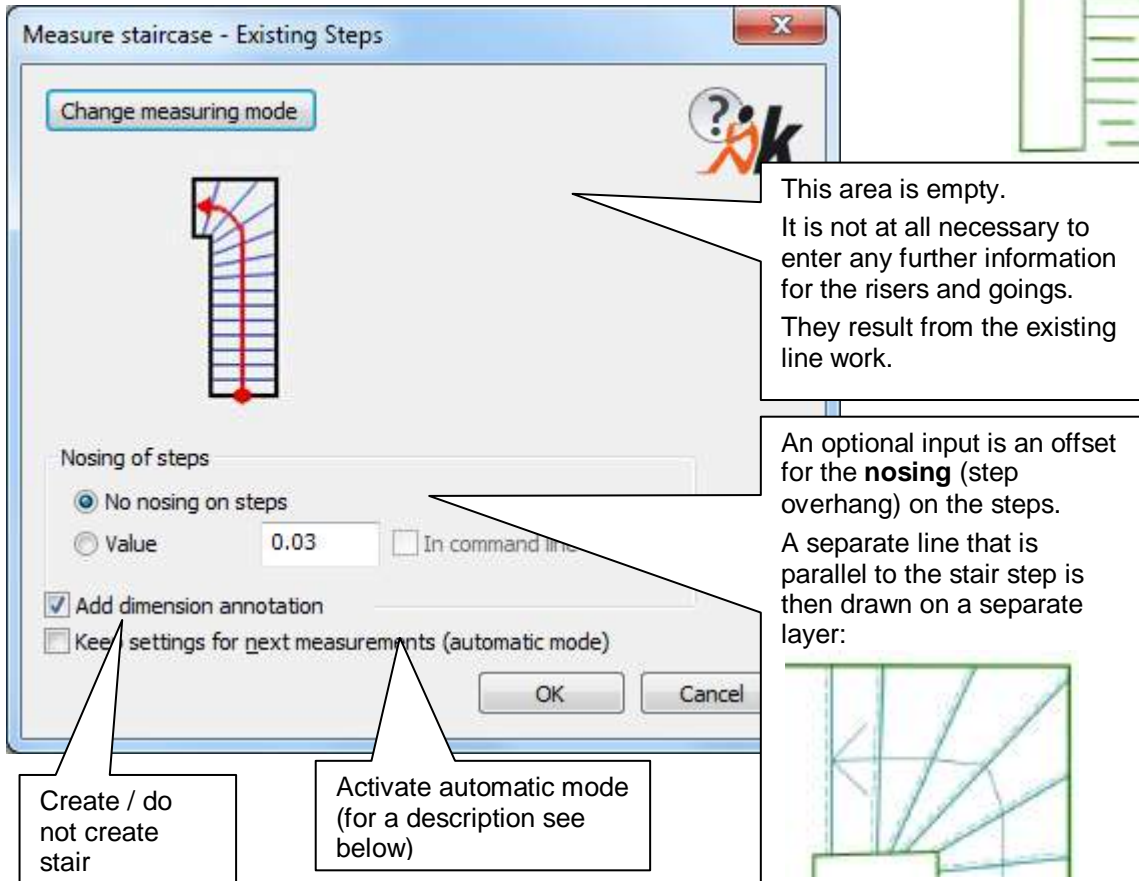
- Existing steps
- Going
- First and last step

After you have made a selection dismiss the dialogue box with [OK] and you return to the dialogue box "Measure Staircase".

Depending on the measuring mode the subsequent procedure will be different:

Existing steps

All steps have been surveyed in advance and exist as line segments. These lines now have to be selected.



Confirm the dialogue box with [OK].

You will now be prompted in the command line for the step lines.

First for the **bounding lines** for the flight of stairs:

Select boundary lines for the first edge of the staircase:

Select objects: *(Select a number of lines, polylines or arcs for one of the two stair strings. Confirm with ENTER!)*

Select boundary lines for the second edge of the staircase

Select objects: *(Select a number of lines, polylines or arcs for one of the two stair strings. Confirm with ENTER!)*

Now the data for the **steps**:

Select Line for the first step of the staircase or [UNdo]: *(Select a line or a one-segment polyline for the first step).*

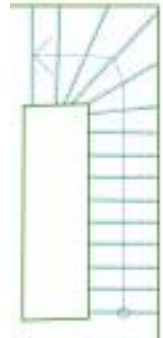
Select existing step lines: *(Select lines or one-segment polylines for the other steps).*

The last prompt is for the **slope direction**.

Point for the arrow direction or [Confirm] <Confirm>: *(Move the mouse until the arrow points in the right direction and then click the mouse)*

The result can look something like this:

Tip: To efficiently survey the stair steps first call the command → "Survey segments".



Going:

Here, a line for the first step will be requested, as well as a value for the going (depth of a step).

Measure staircase - Going of a step

Change measuring mode

Number of steps: 16 In command line

Going: 0.25 In command line

Nosing of steps:

- No nosing on steps
- Value: 0.03 In command line

Add dimension annotation

Keep settings for next measurements (automatic mode)

OK

Based on the number of steps entered and the goings of the steps, the flight of stairs will be constructed. The first step is given and all other steps run parallel to this step.

"In the command line" means that you will be prompted later in the command line for the value. This can make sense when using automatic mode.

An optional input is an offset for the nosing (step overhang) on the steps. A separate line that is parallel to the stair step is then drawn on a separate layer:

Create / do not create stair dimensions

Activate automatic mode (for a description see below)

Confirm the dialogue box with [OK].

You will now be prompted in the command line for the step lines.

First for the **bounding lines** for the flight of stairs (see "existing steps" above):

Now the data for the **steps**:

Select Line for the first step of the staircase or [UNdo]: *(Select a line or a one-segment polyline for the first step).*

Specify Point for direction of the staircase: *(Click or measure a point in the direction of the staircase).*

The last prompt is for the **slope direction**.

Point for the direction of arrow or [Apply] <Apply>: *(Move the mouse until the arrow points in the right direction and then click the mouse)*

The result can look something like this:



First and last step:

Here you have to enter the first and the last step. The first step has to be a line or a polyline; the last step may be entered as additional line, as measurement point or as linear measure.



Line and second line



Line and point



Line and length

Measure staircase - First and last step

Change measuring mode

Number of steps: In command line

Based on the number of steps entered the going (step depth) for a step will be automatically calculated.

Determination of the last step:

Line

Point

Distance:

Nosing of steps:

No nosing of steps

Value: In command line

Add dimension annotation

Keep settings for next measurements (automatic mode)

Create / do not create stair dimensions

Activate automatic mode (for a description see below)

"In command line":

You will be prompted later in the command line for the value. Possibly makes sense when using the automatic mode.

The three ways for entering the length of the stairs

An optional input is an offset for the **nosing** (step overhang) on the steps. A separate line that is parallel to the stair step is then drawn on a separate layer:

Confirm the dialogue box with [OK].

You will now be prompted in the command line for the step lines.

First for the **bounding lines** for the flight of stairs (see "existing steps" above)

Now you will be prompted for the **data for the stair steps**:

Select Line for the first step of the staircase or [UNdo]:

(Select a line or a one-segment polyline for the first step).

Select line for last step of the staircase:

(Enter a line or a one-segment polyline for the last step).

OR

Select point for last step of the staircase: *(Select a point to define the length of the flight of stairs)*

OR

Specify Point for direction of the staircase:

(Click on a point for the direction of the flight of stairs)

The last prompt is for the **slope direction**.

Point for the direction of arrow or [Apply] <Apply>: *(Move the mouse until the arrow points in the right direction and then click the mouse)*

The result can look something like this:



Stair dimensions

If you ticked the box for "Add dimension annotation", then following this command the command → **Annotation – Staircase dimension annotation** will be called.

Layer

The individual elements of the stairs are automatically placed on a special layer:

Element	Layer
Steps	STEPS
Edge of the staircase (stair string)	[stay on their layer]
arrow	STAIRCASEDIMENSIONING
dimensioning block	STAIRCASEDIMENSIONING
Nosing on steps	STEPSNOSING

Automatic mode


The automatic mode is activated when you ticked the box "**Keep settings for next measurements (automatic mode)**". This option allows you to survey several similar stairways without having to enter new values each time. The next time the command is called the dialogue box will not be displayed. The settings will be used that you set last time. When using the automatic mode it is recommended that to enter values that could change (e.g. the number of steps) then use the option "in command line" for each item.

Individual customisation

There are the following possibilities to individually customise the appearance:

- Layer: The preset layer can be altered with the help of the **→Settings...** command. In order to do so, change the according entries in the category "Layer - stairs".
- Use the **→Settings...** under the entry "Building - Stairs – Arrow size" in order to adapt the size of the arrows.

Building Elements – Segments

	Command Name:	PSBSTARTSTOPLINE
	Ribbon Tab:	As-Built Building Plans

This command allows the simple **measurement of line and polyline segments**. With this command you can consecutively measure strings of lines, without any further command input.

The number of points per segment and type of object can be chosen freely.

Here is the command sequence for lines where the number of points = 2, individual lines each with a start point and an end point:

Command: **PSBSTARTSTOPLINE**

Number of points per segment or <2>:

Select object type or [Polyline/Line/UNdo] <Line>:

1. segment point or [UNdo]:

2. segment point or [UNdo]:

1. segment point or [UNdo]:

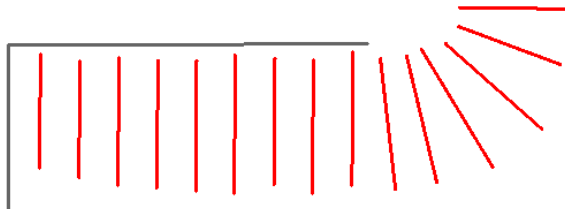
2. segment point or [UNdo]:

1. segment point or [UNdo]:


(...and so on)

With the option "undo" the last observed point will be undone.

The graphic result of the surveying of e.g. steps could look like this:



Building Elements – Rectangle

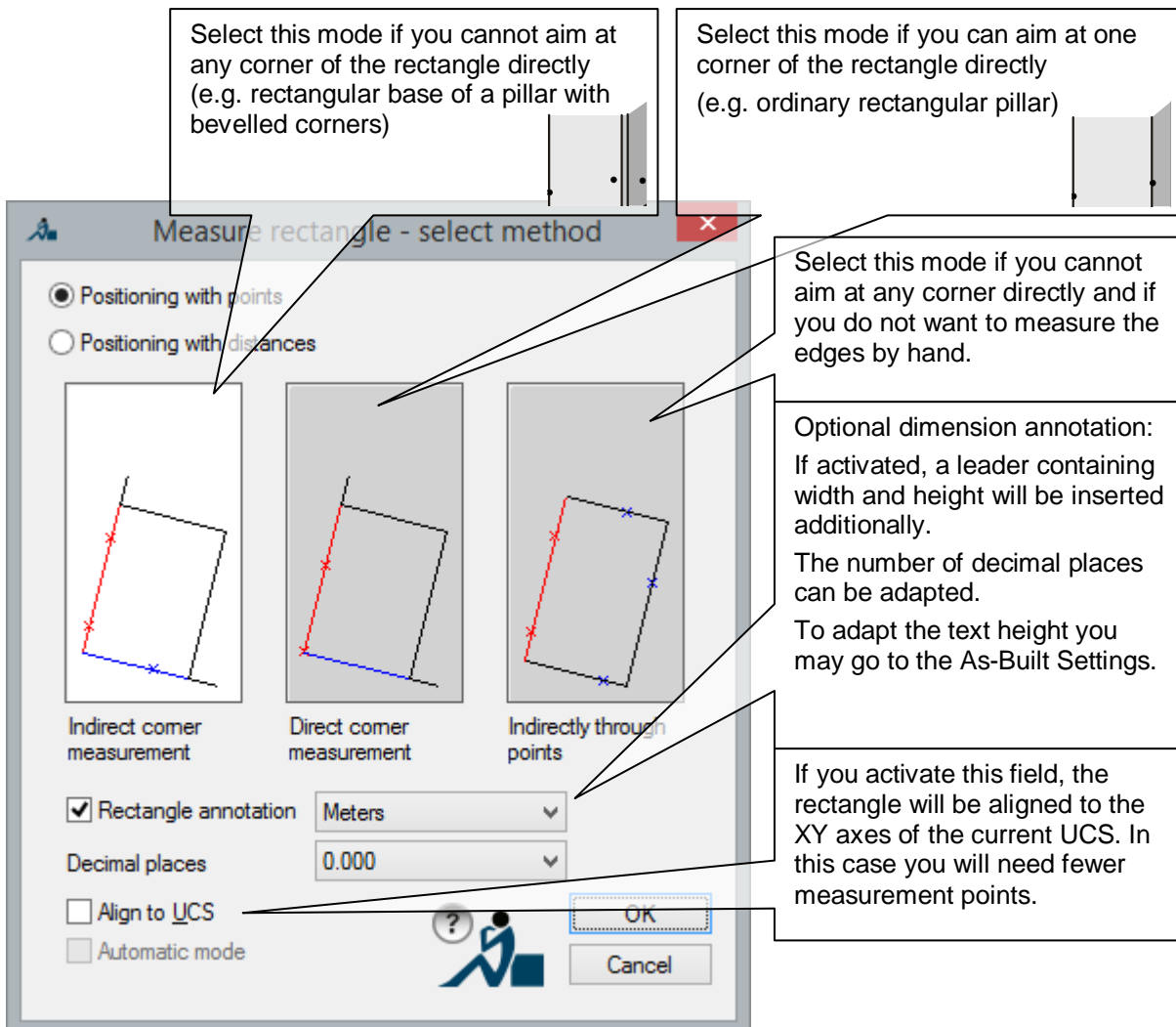
	Command Name:	PSBRECTANGLE
	Ribbon Tab:	As-Built Building Plans

The command allows you to draw a rectangle (e.g. sectional view of a pillar) with just a few clicks in the point cloud. The rectangle always lies in the XY plane of the current UCS with the Z coordinate of the first picked corner point. When selecting the option “Positioning with distances” the Z coordinate will be set to Zero.

Previous hint: This command contains options for point measurement with a total station and options for distance measurement.

There are different measurement modes depending on the points that you can pick easily from the point cloud.

Running the command opens following window:



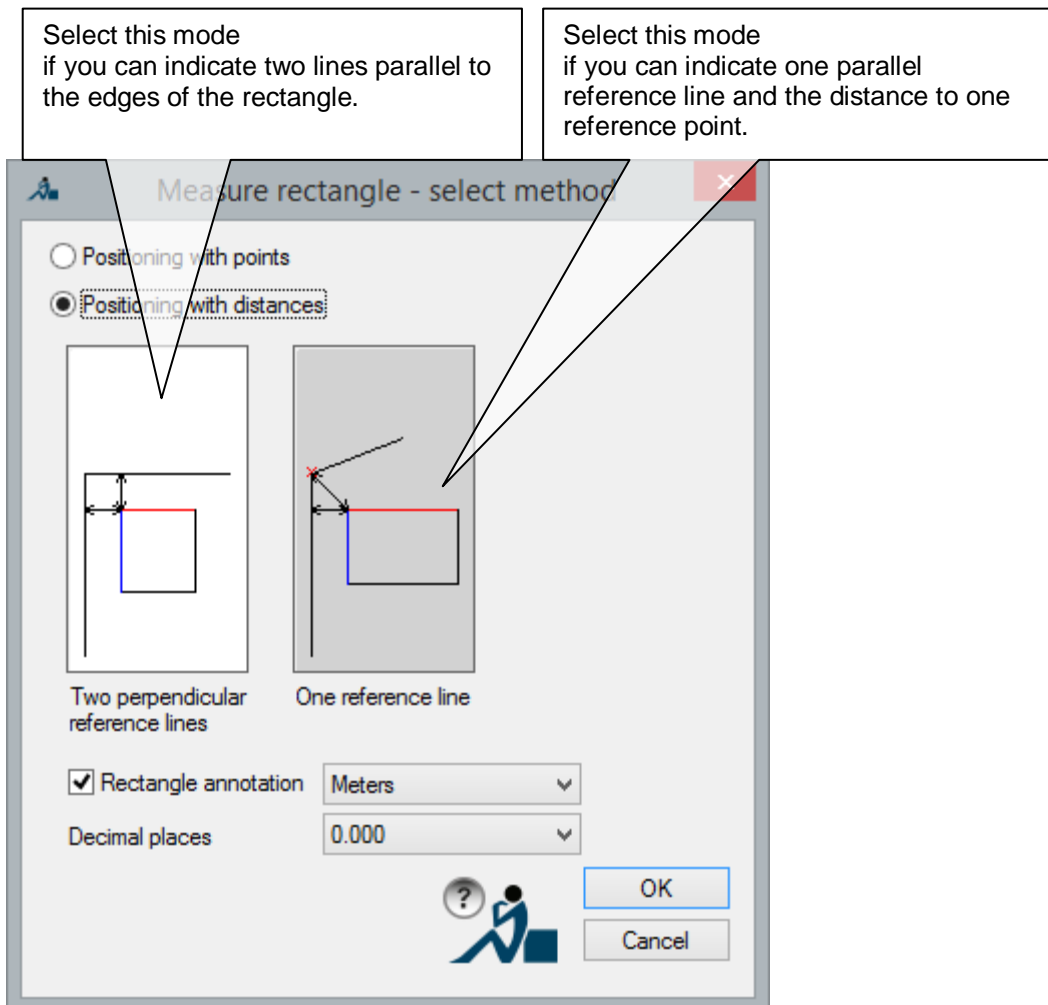
The dialog box, titled "Measure rectangle - select method", contains the following elements:

- Two radio buttons: "Positioning with points" (selected) and "Positioning with distances".
- Three measurement mode thumbnails:
 - Indirect corner measurement:** Shows a rectangle with red and blue lines indicating measurements from an external point to the corners.
 - Direct corner measurement:** Shows a rectangle with red and blue lines indicating measurements directly to the corners.
 - Indirectly through points:** Shows a rectangle with red and blue lines indicating measurements through intermediate points to the corners.
- Checkboxes: "Rectangle annotation" (checked), "Align to UCS" (unchecked), and "Automatic mode" (unchecked).
- Fields: "Meters" (unit), "0.000" (decimal places), and "OK" / "Cancel" buttons.

Callouts provide additional information:

- Indirect corner measurement:** Select this mode if you cannot aim at any corner of the rectangle directly (e.g. rectangular base of a pillar with bevelled corners).
- Direct corner measurement:** Select this mode if you can aim at one corner of the rectangle directly (e.g. ordinary rectangular pillar).
- Indirectly through points:** Select this mode if you cannot aim at any corner directly and if you do not want to measure the edges by hand.
- Rectangle annotation:** Optional dimension annotation: If activated, a leader containing width and height will be inserted additionally. The number of decimal places can be adapted. To adapt the text height you may go to the As-Built Settings.
- Align to UCS:** If you activate this field, the rectangle will be aligned to the XY axes of the current UCS. In this case you will need fewer measurement points.

The option “Positioning with points” is recommended for picking from the point cloud. Select “**Positioning with distances**” if you like to add something using a hand-held laser or measuring tape.



You select a mode by clicking on its icon. Confirm with **OK**.

Now you are asked to measure the points of the rectangle. Depending on the mode, the process differs:

Mode “Indirect corner input”:

Mode = indirect corner

(the current mode is displayed)

First point on first side or [Undo]:

(Click one point on one side of the rectangle)

Second point on first side or [Undo]:

(Click a second point on this rectangle side)

Point on the second side or [Undo]:

(Click a point on an adjoining side of the rectangle)

Following dialogue appears:

The screenshot shows a dialog box titled "Measure rectangle - define lengths". It contains a preview window at the top showing a rectangle with two sides highlighted in red and blue. Below the preview are input fields for "Red length" (0.7) and "Blue length" (0.4), each with a checkbox for "Calculate from measured points". Underneath is an "Aspect ratio" section with three radio buttons: "Rectangle" (selected), "Square red length", and "Square blue length". At the bottom, there is a checkbox for "Keep settings for next measurements (automatic mode)", a help icon, and "OK" and "Cancel" buttons.

The following callouts describe the dialog box:

- The preview is immediately adjusted to the settings.
- Enter the length of the rectangle edges here. If “calculate from measured points” is activated, the length of the edge is determined by the measured points.
- Here you can indicate whether the rectangle is to be a square.
- If the **automatic mode** is activated, the dialogue will not appear the next time. The settings are adopted as you have made them during the last usage. This option is suitable for measuring several rectangles of the same kind without having to make all settings anew.

Mode “Direct corner input”:

Mode = direct corner

(the current mode is displayed)

Corner of rectangle or [Undo]:

(Click one corner of the rectangle)

Point on rectangle side or [Undo]:

(Click a point on one edge of rectangle)

Following dialogue appears:

The screenshot shows a dialog box titled "Measure rectangle - define lengths" with a close button (X) in the top right corner. The dialog contains a preview window showing a rectangle with a red edge and a blue edge. Below the preview are input fields for "Red length" (0.5) and "Blue length" (0.7), each with a checkbox for "Calculate from measured points". There are also radio buttons for "Aspect ratio" (Rectangle, Square red length, Square blue length) and "Choice of side" (Draw to the right, Draw to the left). At the bottom, there is a checkbox for "Keep settings for next measurements (automatic mode)", a help icon (question mark), and "OK" and "Cancel" buttons.

The callout boxes provide the following explanations:

- The preview is immediately adjusted to the settings.
- Enter the length of the rectangle edges here. If “calculate from measured points” is activated, the length of the edge is determined by the measured points.
- Here you can indicate whether the rectangle is to be a square.
- Determines on which side the rectangle is to be drawn.
- If the **automatic mode** is activated, the dialogue will not appear the next time. The settings are adopted as you have made them during the last usage. This option is suitable for measuring several rectangles of the same kind without having to make all settings anew.

Mode “Indirectly through points”:

Mode = indirectly through points

(the current mode is displayed)

First point on first side or [Undo]:

(Click a point on one edge of the rectangle)

Second point on first side or [Undo]:

(Click another point on this edge of the rectangle)

Point on the second side or [Undo]:

(Click a point on an adjoining edge of rectangle)

Point on third side or [Undo]:

(Click a point on an adjoining edge of rectangle)

Point on fourth side or [Undo]:

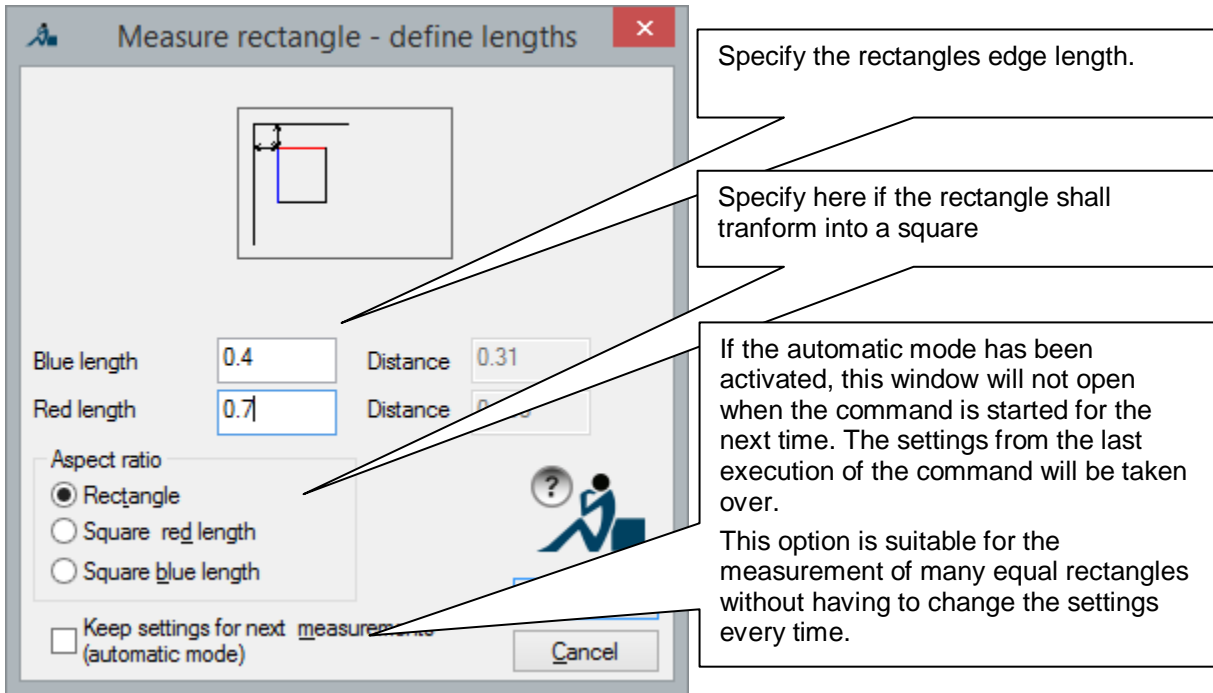
(Click a point on an adjoining edge of rectangle)

There is no separate dialogue for this mode.

Mode "Two perpendicular reference lines" :

- Mode = Two perpendicular reference lines *(the current mode is displayed)*
- Select first reference line [Undo]: *(specify one reference line)*
- Select second reference line [Undo]: *(specify a second reference line)*
- Distance to first reference line [Undo]: *(specify a distance)*
- Distance to second reference line [Undo]: *(specify a distance)*

The following window will appear:



The lengths may also be transferred directly from the hand laser into the window using a Bluetooth connection. The same applies for the previously prompted distances to the reference lines.

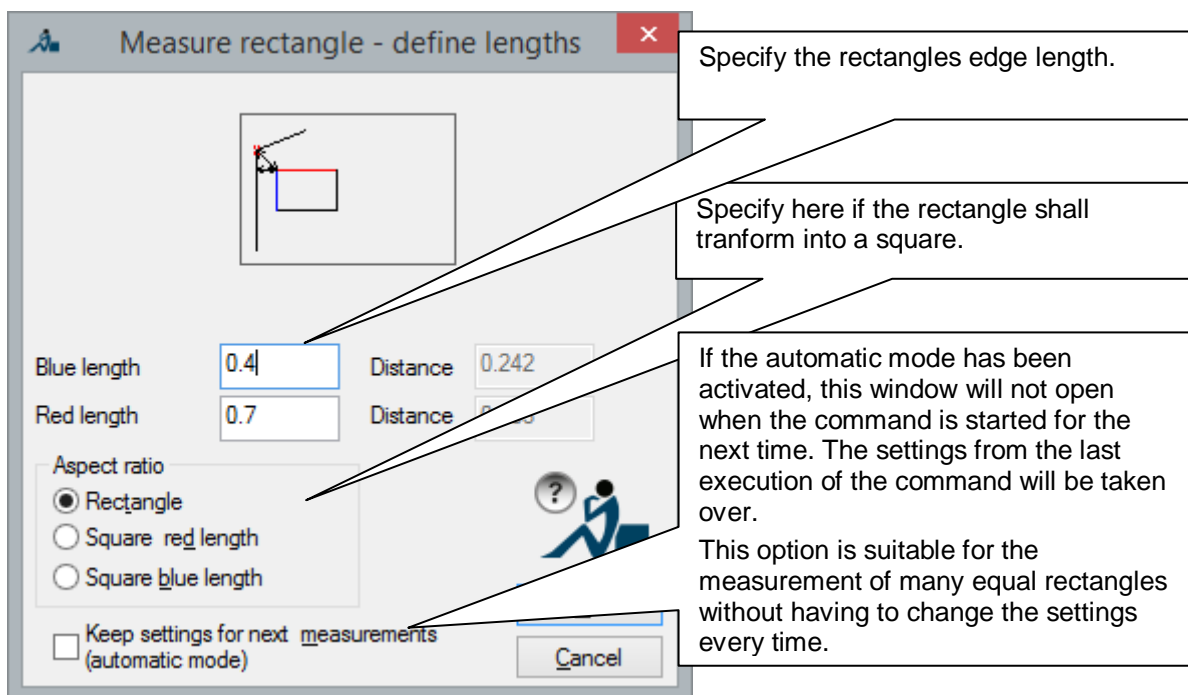
After setting the dialog options and confirming these with **OK** the following prompt will appear:

- Select position [Undo]: *(specify the position of the rectangle)*

Mode "One reference line":

- Mode = one reference line *(the current mode will be shown)*
- Select reference line [Undo]: *(specify a reference line)*
- Select reference point [Undo]: *(specify a point)*
- Distance to reference line [Undo]: *(specify a distance)*
- Shortest distance to reference point [Undo]: *(specify a distance)*

The following window will appear:



The lengths may also be transferred directly from the hand laser into the window using a Bluetooth connection. The same applies for the previously prompted distance to the reference line and to the reference point.

After setting the dialog options and confirming these with OK the following prompt will appear:

- Select position [Undo]: *(specify the position of the rectangle)*

Building Elements – Measure Recess/Ledge

	Command Name:	PSBRECESSMEASURE
	Ribbon Tab:	As-Built Building Plans

With the command you can measure a rectangular recess or a rectangular spur. The recess or the ledge will automatically be drawn into an existing (wall) line.

The command is composed in a way, so you are able to determine the needed data with a manual measurement or with the total station.

After you have started the command, the following window will open. Here you will have to select, with which measurement method you want to measure the individual parameters of the recess or the ledge.

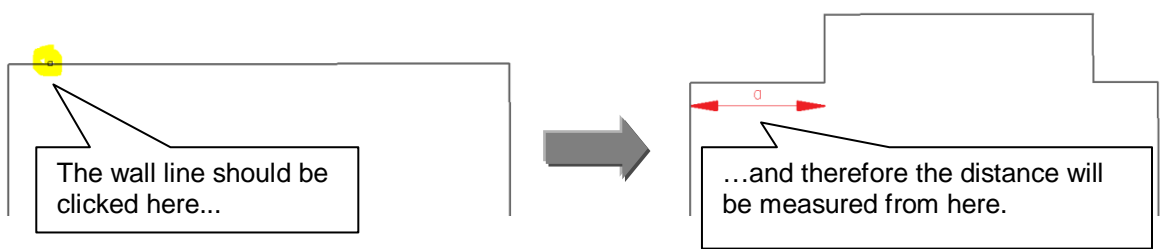
The screenshot shows the 'Insert recess' dialog box with the following settings:

- Recess reveal:** Point R1 on reveal (value: 0.4575, In command line)
- Recess width:** Recess width b (value: 0.7521, In command line)
- Recess depth:** Recess depth t (value: 0.169, In command line)
- Keep settings for next command executions (automatic mode)

Callouts provide additional information:

- Top callout:** The schematic diagram shows the measurement elements selected below. (Points to diagrams showing R1, R2, D, b, t, and a)
- Middle callout:** If you activate "In command line" the value will be prompted within the command line after closing the window. It is suggestive to use this command when also using the automatic mode.
- Bottom callout:** **Automatic mode:** If it is activated, the command will start the next time without the window but directly with the command line prompt. All existing settings from the window will be regarded. It is advisable to use this when you want to measure several recesses at a time.

Distance from corner a: The distance from the corner of the room to the beginning of the recess or the ledge. Click on the line/polyline segment that forms the side from which the distance should be measured.



This value will either be sent into the field with a Bluetooth laser distance meter or entered manually.

Point R1 on the reveal: The first point of the reveal will be measured with the total station.

Recess width:

Recess width b: The width of the reveal or the ledge is determined manually and entered here.

Point R2 on the reveal: The second point of the reveal will be measured with the total station.

Recess depth:

Recess depth t: Measure the depth of the reveal manually and enter.

Point D in recess: Measure point on the back side of the back of the recess (or the forward area of the spur).

After you have adjusted all settings, close the window with the **[OK]** key.

Now you will be prompted for a wall line. This can be a line or a 2D or 3D polyline.

Select wall line or [UNdo]:

In the next step you will be prompted for the set measuring elements in the command line. An example:

Select reveal point R1 or [Distance/UNdo]:

Select reveal point R2 or [Width/UNdo]:


Select point D or [Depth/UNdo]:

Thereafter is the command finished, and a rectangular alcove will be inserted into the selected line/polyline.

Automatic mode

The automatic mode will be activated by checking "**Keep settings for next measurements (automatic mode)**". This setting is useful when you have to measure many similar recesses or ledges, then you don't have to adjust the settings again each time. When calling up the command again, the window will not open. The settings will be adapted, as set the last time. It is suggested to use the "in command line" option for the data entry (width of the reveal) when being in the automatic mode. If needed, the automatic mode can be deactivated via an option within the command line.

Building Elements – Draw grid

	Command Name:	PSBDRAWGRID
	Ribbon Tab:	As-Built Building Plans

With the command “Survey grid” you can fill a closed contour with grid lines. The position, alignment and grid spacing are precisely definable with the assistance of various optimised methods of measurement.

Typical areas of use are the drawing of ceiling grids or floor tiles/floor slabs.

This command includes options for point observations with a total station, measurements in the point cloud, as well as options for measuring distances.

The methods of measurement can also be combined depending on the spatial situation.

After calling the command the following dialogue box is displayed:

You select here the method of observation and with which individual grid parameters you want to survey.

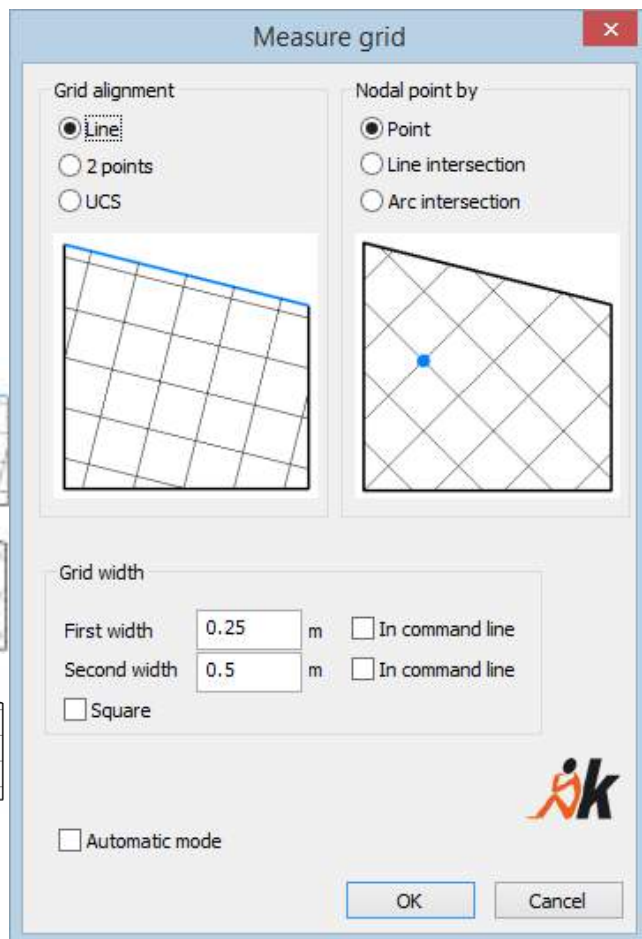
Grid alignment:

In which direction should the grid run?

Line: Select a line with which the grid should be aligned parallel to.

2 Points: Click or observe two points, where the grid should be aligned parallel to the line joining the two points.

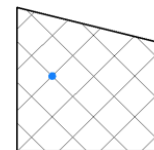
UCS: The grid should be aligned parallel to the XY axes of the UCS.

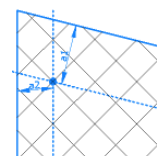
Node:

Where should the intersection points lie? With both of these methods you must explicitly define an intersection point:

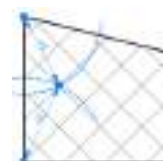
Point: Simply click or observe a point.



Line intersection: Select two known lines and two parallel distances from these lines. The intersection point of the parallel lines defines the node.



Arc intersection (Bilateration): The node is defined by selecting two known points and giving two radial dimensions. The intersection point of the two arcs is the node.

**Grid spacing:**

The spacing of the grid lines in two directions. If “In command line” has been selected, then the value will be asked for in the command line after the dialogue box has been closed. This makes sense when used in connection with automatic mode.

Automatic mode:

If this is selected then the next time that the command is called it will not display a dialogue box, but immediately the command line prompts. All of the current settings in the dialogue box will be taken into account. This makes sense if several grids should be consecutively surveyed.

After you have entered all of the settings, dismiss the dialogue box with **[OK]**.

In the command line you are now prompted to enter each of the appropriate dialogue settings:

Command: **_TCADDRAWGRID**

Grid boundaries:

Select objects: *(Select the objects that form the closed contour)*

Select reference line or [Cancel/UNdo]: *(Pick the line)*

Or:

1st point for grid alignment or [Cancel/UNdo]: *(pick or observe a point)*

2nd point for grid alignment or [Cancel/UNdo]: *(pick or observe a point)*

Node for the grid or [Cancel/UNdo]: *(pick or observe a point)*

Or

Nodal point by line intersection, select 1st line or [Cancel/UNdo]: *(Pick the line)*

Nodal point by line intersection, select 2nd line or [Cancel/UNdo]: *(Pick the line)*

Enter distance to the first reference line or [Cancel/UNdo] <1.0000>: **0.5** *(Enter value)*

Enter distance to the second reference line or [CAnceL/UNdo] <1.0000>: **0.3** (Enter value)

Or

Nodal point by arc intersection, select 1st point or [CAnceL/UNdo]: (pick or observe a point)

Nodal point by arc intersection, select 2nd point or [CAnceL/UNdo]: (pick or observe a point)

Select a side:


Horizontal distance from first point or [CAnceL/UNdo] <0.0000>: **1** (Enter value)

Horizontal distance from second point or [CAnceL/UNdo] <0.0000>: **2** (Enter value)

Enter 1st grid spacing or [CAnceL/UNdo]<1.0000>: **0.1** (Enter value)


Enter 2nd grid spacing or [CAnceL/UNdo]<1.0000>: **0.2** (Enter value)

The resulting lines are placed on a predefined layer called “Grid”.

With the assistance of the →**Settings**  command you can adapt the layer to your individual wishes.



Edit – Fix lines to corner

	Command Name:	PSBCORNERFIX
	Ribbon Tab:	As-Built Building Plans

The command is used for intersecting lines, polylines and arcs. It also works for skewed lines, polylines and arcs. 'Fix lines to corner' can be used if, for example, the corner of a room is obstructed, and you have to stop measuring on both sides before the room corner.

Command: PSBCORNERFIX


Select first line, polyline or arc:

Select second line, polyline or arc:

In order to tell the command the direction in which you want to intersect, you always have to pick the side of the line, polyline or arc to be intersected at.

Only the XY coordinates of the current UCS will be used for calculating the point of intersection. That means that, for example, two skewed lines, polylines or arcs will still be skewed after intersection i.e. do not really intersect, but appear to intersect from top view in the UCS.

Edit – Perpendicular return

	Command Name:	PSBPERPRETURN
	Ribbon Tab:	As-Built Building Plans

The command is used to drop a point perpendicular onto a line.

Command: PSBPERPRETURN

Select line: *(Select a line or another curve.)*

Current modes: Projection=2D Break=On POints=On Line=On

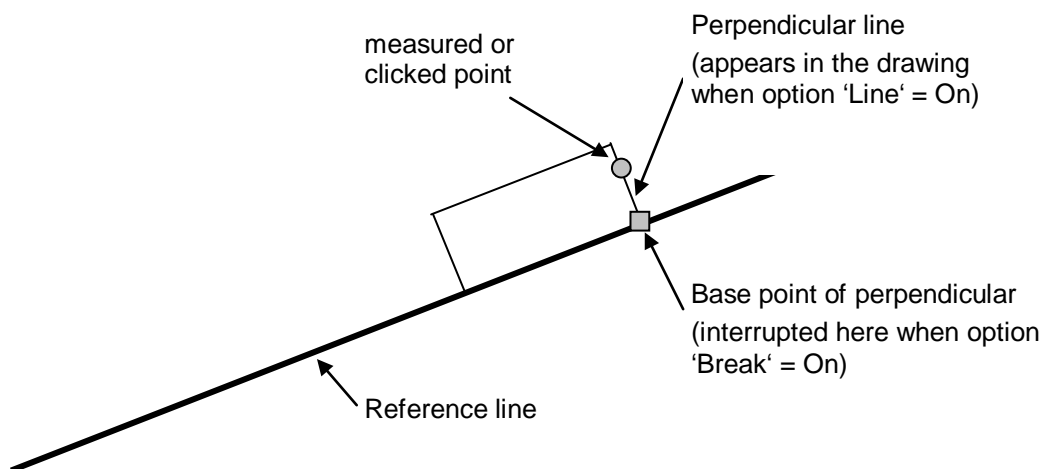
(The current settings of the options are displayed.)

Direction of return or [Projection/Break/POints/Line/Undo]:


(Click or measure the point onto which the reference line is to be dropped)

The returned point is determined by the shortest distance between the selected point and the reference line. The different command line options define how the point is processed:

Option	Value	Description
Projection	2D	The measurement to be fixed refers to the projection of the line into the XY plane of the current UCS. Thus, the line appears perpendicular to the reference line in the top view of the current UCS.
	3D	The measurement to be fixed refers to the line itself.
Break	On	The line is broken at the base point of the perpendicular.
	Off	
Points	On	An AutoCAD point is inserted at the base point of the perpendicular.
	Off	
Line	On	Measured point and base point of perpendicular are connected by a line.
	Off	



Edit – Horizontal extend

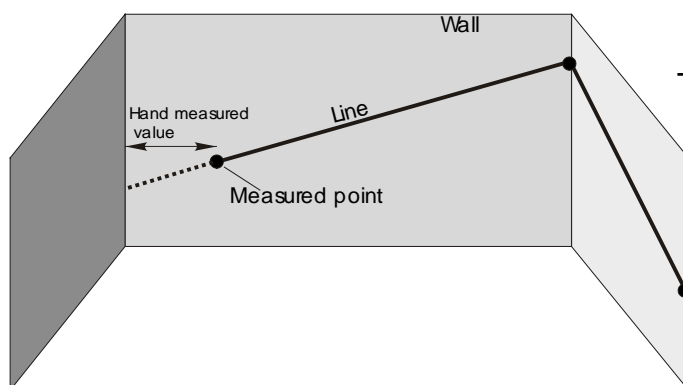
	Command Name:	PSBEXTEND (Option [Horizontal])
	Ribbon Tab:	As-Built Building Plans

This command is used to extend a line **horizontally** in the current UCS. That means that a line running diagonally through the space is extended by a value that does not refer to its direction in the space but to its projection on the XY plane of the current UCS.

Command: PSBEXTEND

Select the line: *(select the line to be extended)*

Line horizontal extend by [By/Total/Horizontal/3D/to Point]: **0.20** *(Enter extension measurements)*



The line is extended into the direction that lies closer to the selected point.

Very often you will find situations in which you cannot aim at room corners directly. In such cases just measure to a point on the wall with your instrument and the remaining distance with a tape measure

All other options are explained under command **3D extend**.

The command can be processed as transparent command within the command 'Line'. In this cases, the extension is carried out from the last measured line end point in the direction of the last line.

Command: **line**

Specify first point: *(e.g. measure the first corner)*

Specify next point or [Undo]: *(e.g. measure before the next corner)*

Specify next point or [Undo]: 'PSBEXTEND

(transparent call-up of the command)

Line horizontal extend by [By/Total/Horizontal/3D/to Point]: **0.20**

(Remaining distance to wall corner)


Specify next point or [Undo]:

(continue measuring)

The transparent use of the command 'PSBEXTEND' is of advantage when you want to avoid interrupting the measurement of a line (e.g. the wall intersection line of a room) because a corner is obstructed.

Remark: The extended part of the line should always be shorter than the original line. Never extend a line by a factor longer than the original line because a little measurement error could thus be multiplied.

Edit – 3D extend

	Command Name:	PSBEXTEND (Option [3D])
	Ribbon Tab:	As-Built Building Plans

This command is used for extending a line in a **three-dimensional** way.

Command: PSBEXTEND

Select the line: *(select line to be extended)*

Line 3D extend by [By/Total/Horizontal/3D/to Point]: **0.35**

(e.g. Extension value of a stringboard)

The line is extended into the direction that is closer to the selected point.

With the command line options 'By' or 'Total' you can decide whether the line should be extended by a certain value or to a certain total length. You can also enter negative values to shorten the line. The option point allows the extension to the height of a determined point. For further information about the option horizontal, please refer to the command **Horizontal extend**.

The command can be called up within the command 'Line' as transparent command. In this case, the line is extended from the last measured line end point along the direction of the last line.

Command: **line**

Specify first point: *(e.g. measure first wall corner)*

Specify next point or [Undo]: *(e.g. measure before the next wall corner)*

Specify next point or [Undo]: 'PSBEXTEND

(Call up the command as transparent command)


Line 3D extend by [By/Total/Horizontal/3D/to Point]: **0.20** *(Enter extension value)*

Specify next point or [Undo]: *(continue measuring as usual)*

If you use the command 'PSBEXTEND' as transparent command, you can avoid to interrupt a line since e.g. a corner cannot be measured directly.

Remark: The extended part of the line should always be shorter than the original line. Never extend a line by a factor longer than the original line because a little measurement error could thus be multiplied.

Edit – Perpendicular extend

	Command Name:	PSBPERPEXTEND
	Ribbon Tab:	As-Built Building Plans

The command is used for the perpendicular, horizontal (in the current UCS) extension of a line from the last line end point. The direction of the line (e.g. left or right) and its length are to be entered or to be picked.

You may call-up the command as transparent command within other commands, e.g. within the 'Line' command:

Specify first point: *(e.g. click first corner)*

Specify next point or [Undo]: *(e.g. click to recess)*

Specify next point or [Undo]: 'PSBPERPEXTEND'
(transparent call-up of the command 'PSBPERPEXTEND')

Enter length or [Right/Left]: **0.20** *(e.g. depth of recess)*

Select side [Right/Left] or pick point: *(Pick point on side or entry)*

Specify next point or [Undo]: *(continue working)*

In case you do **not** call up the command 'PSBPERPEXTEND' within another command you will be asked to select the line to be extended. The line will be extended on the side that is closer to the selected point. The new line will be drawn on the current layer.

Command: PSBPERPEXTEND

Select line: *(select the line to be extended)*

Enter length or [Right/Left]: **0.20** *(e.g. depth of wall recess)*

Select side [Right/Left] or pick point: *(Pick point on side or entry)*

Remark: The newly calculated line end point can be adopted with RETURN the next time 'Line' is called up.

Note: The base line for the perpendicular extension should be a long line. Never use a value for perpendicular extension that is longer than the base line since a little measurement error could thus be multiplied.

Dimension – Absolute height

2.41 ▽	Command Name:	PSBABSHEIGHT
	Ribbon Tab:	As-Built Building Plans

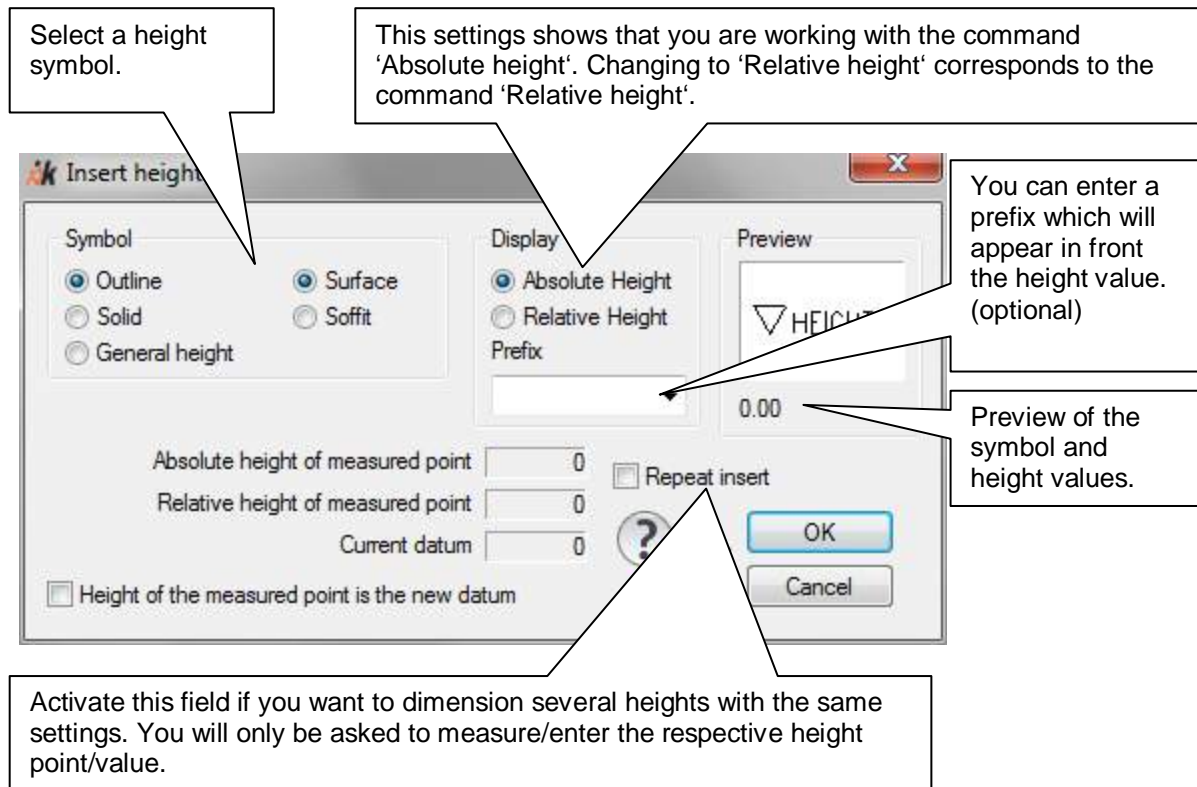
The command “PSBABSHEIGHT” works as the command “PSBRELHEIGHT” and is used for measuring and dimensioning absolute heights.

You start with following request:

Command: PSBABSHEIGHT

Measure height point: (Snap point)

Following dialogue appears:



If a height point has been entered, it also serves as insertion point for the dimensioning block.

The dimensioning block is drawn on the layer specified in the As-Built → **Settings** in the section “Layers” under ‘Height dimensions’. If this value is kept empty, the current layer will be used.

The dimensioning block memorises its relative height reference at the time of its creation. Thus, if the relative height is changed later, it does not influence already existing blocks.

Dimension – Relative height

[2.41] ▽	Command Name:	PSBRELHEIGHT
	Ribbon Tab:	As-Built Building Plans

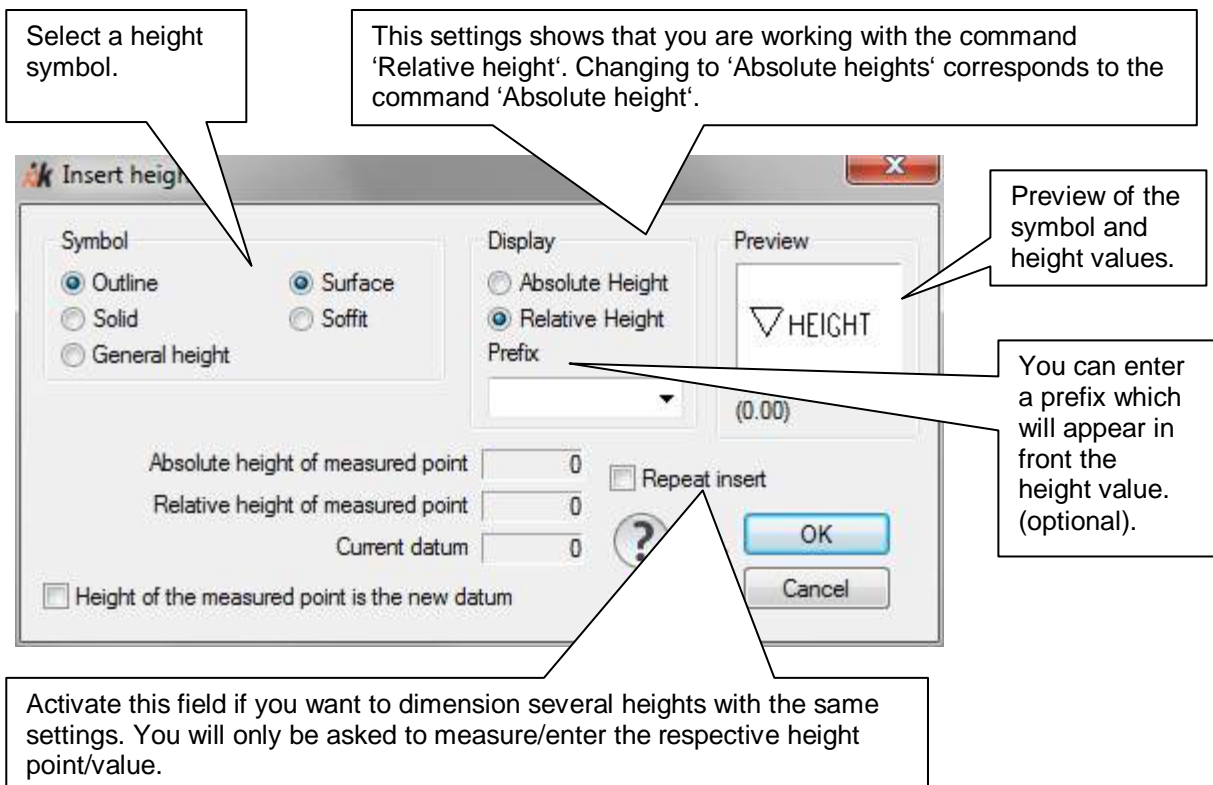
The command 'PSBRELHEIGHT' works as the command 'PSBABSHEIGHT' and is used for measuring and dimensioning relative heights (according to the current reference height).

You start with following request:

Command: PSBRELHEIGHT

Measure height point or enter [Value]: *(Snap point or option 'V' for manual entry)*

Following dialogue appears:




If a height point has been entered, it also serves as insertion point for the dimensioning block. Otherwise, it must be entered explicitly at the end of the command.

The dimensioning block is drawn on the layer specified in the As-Built → **Settings** in the section "Layers" under 'Height dimensions'. If this value is kept empty, the current layer will be used.

The dimensioning block memorises its relative height reference at the time of its creation. I.e. if the relative height is changed later, it does not influence already existing blocks.

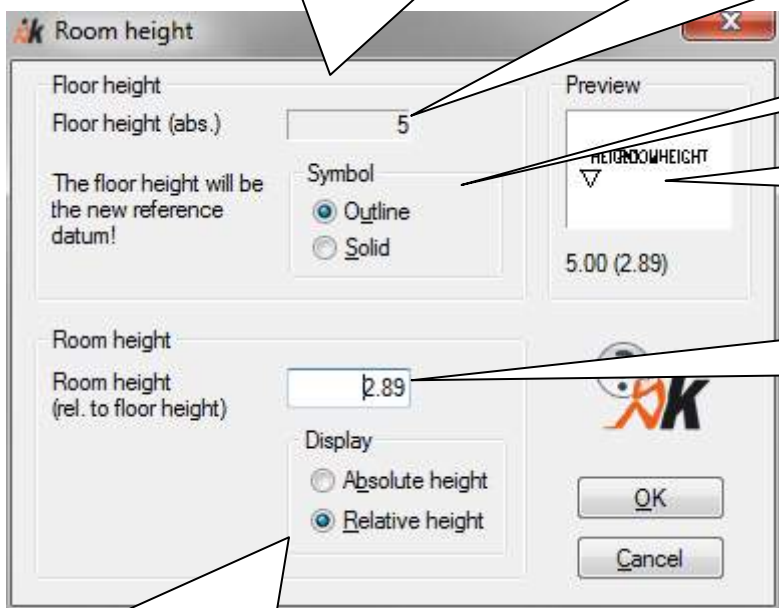
Dimension – Room height

	Command Name:	PSBROOMHEIGHT
	Ribbon Tab:	As-Built Building Plans

The command is used for dimensioning floor and room heights. After the command has been started, following request appears in the command line:

Insertion point on the floor (Snap one point on the floor)
 Point on the ceiling or enter [Value]: <Value> (Snap a point in the ceiling
 or select option 'M' to insert the room height manually)

After the points have been measured, following dialogue appears:



The measured floor height is set as new reference height for all following relative height dimensions.

Absolute height of the measured floor point.

Select a height symbol.

Preview of the symbol and height values.

Here you can change the room height that had been measured or manually entered.

Determine whether the room height is to be displayed in reference to the floor (relative) or to the zero point of the absolute height.


The dialog box 'Room height' contains the following elements:

- Floor height:** A text box containing the value '5'.
- Floor height (abs.):** A text box containing the value '5'.
- Symbol:** Radio buttons for 'Outline' (selected) and 'Solid'.
- Preview:** A preview area showing a height dimension symbol with the value '5.00 (2.89)'.
- Room height:** A text box containing the value '2.89'.
- Room height (rel. to floor height):** A text box containing the value '2.89'.
- Display:** Radio buttons for 'Absolute height' and 'Relative height' (selected).
- Buttons:** 'OK' and 'Cancel' buttons.

OK confirms the entries. The height symbol is inserted at the position of the floor point.

The dimensioning block is drawn on the layer specified in the As-Built → **Settings** in the section “Layers” under ‘Height dimensions’. If this value is kept empty, the current layer will be used.


Dimension – Convert relative to absolute height

	Command Name:	PSBRELTOABS
	Ribbon Tab:	As-Built Building Plans

The command is used to convert relative heights to absolute heights. The command works only for relative heights that have been created with kubit commands. These are:

- Dimension - Room height
- Dimension - Absolute height
- Dimension - Relative height
- Dimension - Window dimension annotation
- ...


Dimension – Convert absolute to relative height

	Command Name:	PSBABSTOREL
	Ribbon Tab:	As-Built Building Plans

The command is used to convert absolute heights to relative heights. The command works only for absolute heights that have been created with kubit commands. These are:

- Dimension - Room height
- Dimension - Absolute height
- Dimension - Relative height
- Dimension - Window dimension annotation
- ...

Dimension – Set relative reference datum

	Command Name:	PSBREFDATUM
	Ribbon Tab:	As-Built Building Plans

With this command you may set the reference datum for the relative height dimension. The datum can be taken from a point or be entered manually.

Command: PSBREFDATUM


The reference datum is 0.000000

Measure point or enter [Value]: *(Snap point for new reference datum or option 'V' for manual entry)*

The reference datum has been set to 2.500000

All following relative height dimensions refer to this new reference datum.

Dimension – Subsequent editing – Set reference height of height dimension

	Command Name:	PSBREFBLOCK
	Ribbon Tab:	As-Built Building Plans

With this command you may change the reference height of selected kubit height blocks in the current drawing.

Command: PSBREFBLOCK

Select height blocks.

Select objects:

(Select height blocks to be edited)

The reference datum is **0.9**

(Status of the current reference height)

Measure point or enter [Value]: **300.5**


(measure/enter reference height)

54 height blocks converted.

(Status information number of converted blocks)

The shown reference datum is the one currently set for the drawing. This might differ from the original reference height of the blocks to be edited.

Dimension – Subsequent editing – Show properties of height dimension

	Command Name:	PSBHBLOCKLIST
	Ribbon Tab:	As-Built Building Plans

With this command you may display the current properties for a kubit height block.

Command: PSBHBLOCKLIST

Select a block insert:


(Click on the block you want)

Attribute: HEIGHT: absolute height: 2.6600, reference height: 0.0000, displayed

height: Absolute height

(Display of properties)

Dimension – Subsequent editing – Set absolute reference datum

	Command Name:	PSBABSDATUM
	Ribbon Tab:	As-Built Building Plans

This command is used for changing the absolute height reference of the drawing.

A typical case for the usage of this command is when the absolute height reference only becomes known after the survey project has been started. Then it is necessary to adjust the already existing survey to the new height reference. The objects in the drawing will be shifted by a certain height offset (Z direction in WCS), and the attributes of the kubit height dimensions that already exist in the drawing will be adjusted to the new height reference. Following kubit blocks will be updated

- Absolute and relative heights, floor and room heights
- Window dimensions (Breast and lintel heights)
- Arc dimensions

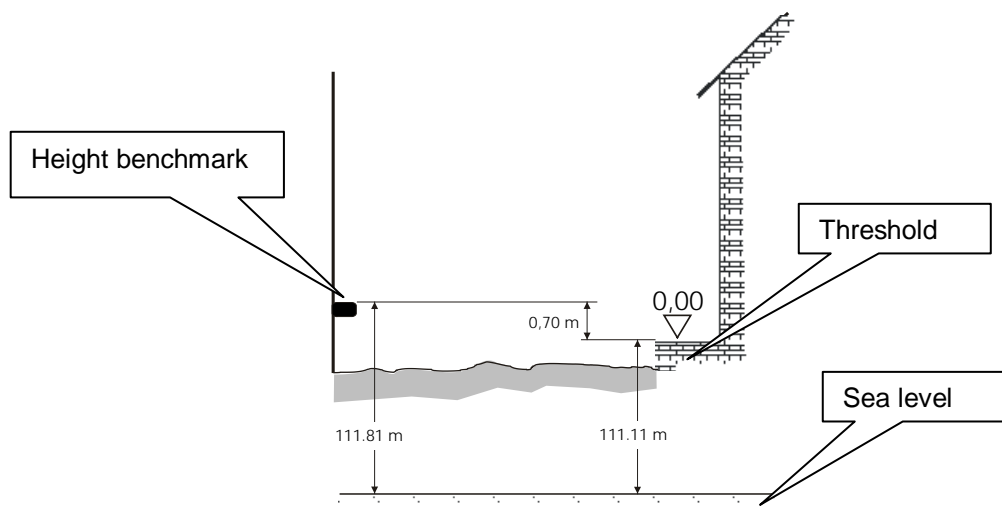
An example:

The start of a building survey: The first station is situated in the entrance area. The absolute height reference is determined on the threshold of the entrance door at 0.00m. All absolute height dimensions refer to this point.

However, later it is demanded that all building heights refer to sea level. A nearby height benchmark is measured and should be used as reference point. Its official height is 111.81m. Now the survey, as completed so far, needs to be adjusted to this height.

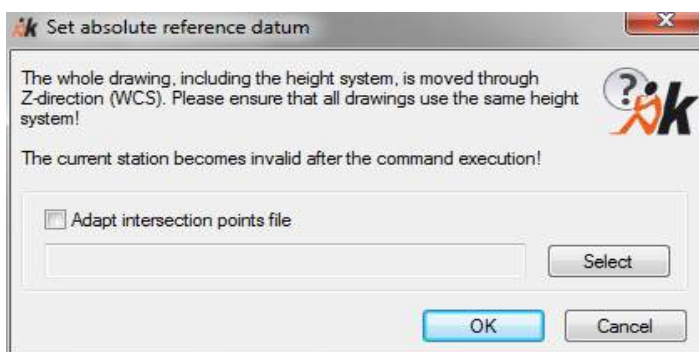
Following calculation ensues: The height benchmark had been measured at 0.70m (above the threshold), but its official height is 111.81m. Thus, the height of the threshold is:

$$111.81\text{m} - 0.7\text{m} = \mathbf{111.11\text{m}}$$



Therefore, the drawing needs to be shifted 111.11m.

When running the command the following dialogue opens.



Besides important advice it provides the possibility to select the intersection points file for adaptation. This file with the extension 'rim' is usually used for the 'arc intersection' command of the TachyCAD software. Close the dialogue with 'OK'. Further query follows in the command line:

Command: **absdatum**

Enter the value: **111.11**

(Enter value for height offset)

39 objects selected.

(status display, which and how many

5 window dimension(s) converted.

dimensions have been updated in the drawing)

1 arch dimension(s) converted.

3 height blocks converted.

Relative datum shifted.

(Watch relative height has been adjusted)

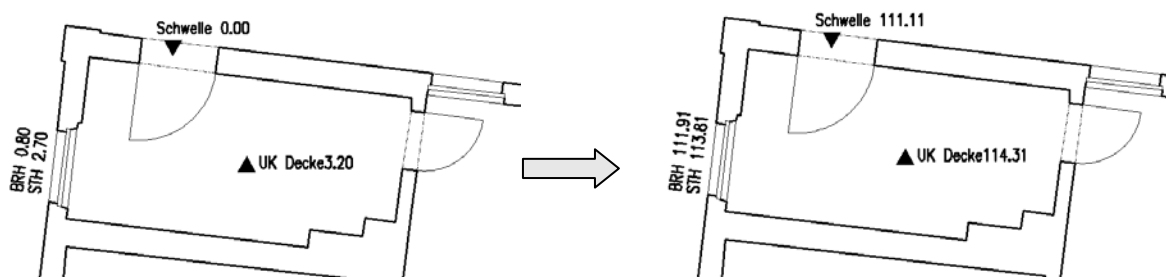
Intersection measurements converted.

*(Note: Data in the intersection points file TachyCAD.rim
have been adjusted)*

Current orientation deleted.

*(Note: Orientation data have been deleted,
new orientation necessary!)*

Total objects shifted: 39




Important advice!

If used the wrong way, this command can jumble up your whole reference system. If you change the current height reference, you should make sure to do so in all drawings belonging to the project in order to keep the reference system consistent.

Please do **not** change the absolute reference datum **after** having flattened the drawing (→Flatten drawing). Since this affects the display of the attributes of the height blocks. Unless, you excluded the height blocks from flattening by not selecting them in the according command and making the appropriate settings inside the dialogue window.

Dimension – Subsequent editing – Define blocks

	Command Name:	PSBBLOCKDEF
	Ribbon Tab:	As-Built Building Plans

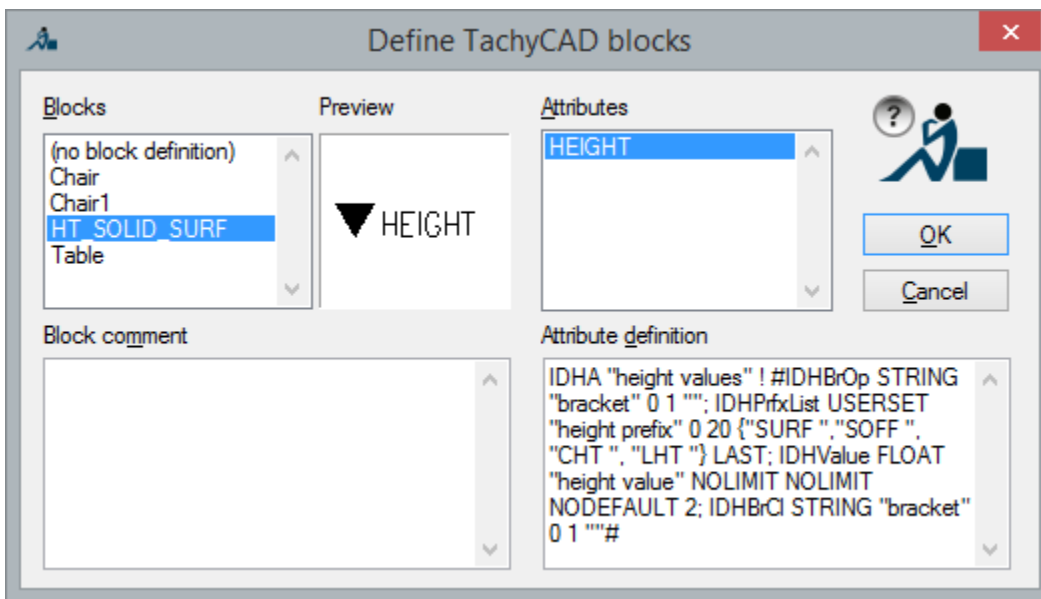
As-Built is in the position to extend AutoCAD attributes by certain information which allows to manage blocks, block attributes and their contents much safer and more flexible.

So-called extended attribute definitions that allow you to assign not only text, but also numbers, sets or constants can be added to attributes. For the insertion of the individual elements of this attribute, an input mask is defined with the help of the attribute definition.

You can use the command “Block definition” for two different uses:

- Adapting the blocks used in As-Built to meet your own requirements (e.g. The number of decimal places, values for prefixes and suffixes)
- Definition of your own blocks with extended attribute definitions. The associated command to insert the self defined blocks is → Insert blocks

After starting the command the following dialogue box opens up:



If the drawing already contains block definitions, these are displayed under “Blocks”. Select the block that is to be edited. Under “**Attributes**” select the attribute that is to be edited.

Block comment: You can add a comment for every block definition (optional).

Attribute definition:

Here is the actual extended attribute definition, that must follow a specific syntax. This syntax is described in detail below. You can find explanatory examples at the very bottom.

Syntax of the attribute definition

In this list the structure of such an attribute definition is summarised.

ATTRIBUTEDEFINITION	=	ATTRIBUTIDSTRING [COMMENTSTRING [!][#'(ELEMENTSTRING ;)* #]
COMMENTSTRING	=	TEXTSTRING
ELEMENTSTRING	=	ELEMENTIDSTRING TYPSTRING COMMENTSTRING ('PARAMETERSTRING)*
TYPSTRING	=	'CONST' 'SET' 'USERSET' 'INT' 'FLOAT' 'STRING'
PARAMETERSTRING	=	TEXTSTRING INTSTRING FLOATSTRING SETSTRING LIMITSTRING SETDEFAULTSTRING INTDEFAULTSTRING FLOATDEFAULTSTRING STRINGDEFAULTSTRING
SETSTRING	=	{(SETELEMENTSTRING ;)* }
SETELEMENTSTRING	=	TEXTSTRING
SETDEFAULTSTRING	=	SETELEMENTSTRING 'LAST' 'NODEFAULT'
INTDEFAULTSTRING	=	INTSTRING 'LAST' 'NEXT' 'NODEFAULT'
FLOATDEFAULTSTRING	=	FLOATSTRING 'LAST' 'NODEFAULT'
STRINGDEFAULTSTRING	=	TEXTSTRING 'LAST' 'NODEFAULT' 'NEXT'
LIMITSTRING	=	'NOLIMIT' FLOATSTRING
TEXTSTRING	=	'" 'Text' "
INTSTRING	=	Integer number
FLOATSTRING	=	Decimal number
ATTRIBUTIDSTRING	=	Text
ELEMENTIDSTRING	=	Text (Text must be without '#' or ' '!)

The sequences in parenthesis with added *, as in (ELEMENTSTRING ;)*, means that the sequence in parenthesis can be repeated any number of times. Entries in [] are optional, | means "or". All characters in inverted commas appear directly in the attribute definition. The individual sequences are separated by blank characters. There are no word wraps, all entries are done one after the other. Generally, every attribute definition consists of three parts:

ATTRIBUTEDEFINITION = ATTRIBUTESTRING [COMMENTSTRING [!][#'(ELEMENT STRING ;)* #]

- The ATTRIBUTESTRING is a simple sequence of characters and is used for the identification of the attribute definition.
- The COMMENTSTRING describes the comment that is to appear in the input mask. It is of the type TEXTSTRING and is therefore given in inverted commas. If an ! is given, the attribute is definitely shown and its values must be valid. If no ! is given, the user can decide whether he or she wants the attribute.
- The ELEMENTSTRING specifies the functionality of the input mask. An attribute may have a maximum of five elements. The mask can be designed quite variable by combining the individual elements. Following types of elements are possible: (the name in parenthesis is the respective name of the type):
 - Entry of constant texts (CONST)
 - Selection of constant text elements, fixed (SET)
 - Selection of constant text elements, changeable (USERSET)

- Whole numbers (INT)
- Decimals (FLOAT)
- Free text entry (STRING)

The difference between SET and USERSET is that the user can change the entry for the latter, while he can only choose from offered entries at SET. Beginning and end of an element string are marked by the sign #.

Syntax and semantics of the ELEMENTSTRING

An ELEMENTSTRING contains, depending on the type, following sequence of elements:

ELEMENTSTRING = ELEMENTIDSTRING `` TYPESTRING `` COMMENTSTRING (`` PARAMETERSTRING)*

- The ELEMENTIDSTRING is used for the identification of the element.
- In the TYPESTRING the element type described above is defined.
- In the COMMENTSTRING a comment can be defined that will then appears in the input mask above the element. It is of the type TEXTSTRING. If you do not want to enter a comment, enter blank inverted commas "".
- The PARAMETERSTRING takes a different shape depending on the type of element. Here it is determined what exactly can be selected or entered in the mask. Again, it consists of different parts. The table shows you which sequence must be used for which element type.

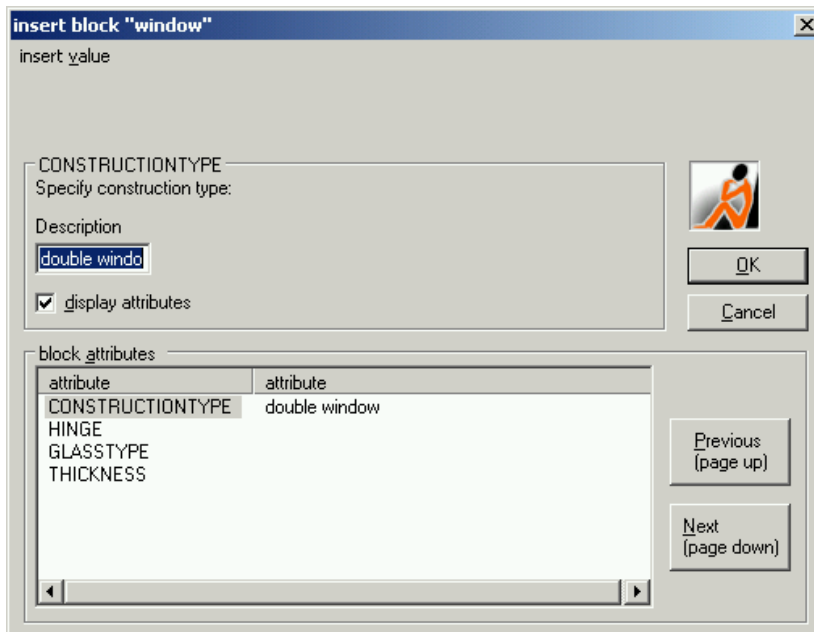
ELEMENT-IDSTRING	TYP-STRING	COMMENT-STRING	PARAMETER-STRING	PARAMETER-STRING	PARAMETER-STRING	PARAMETER-STRING
(identification-name)	CONST (constant text)	ignored	TEXTSTRING (default value)			
(identification-name)	USERSET (set of constant text elements, changeable)	(Comment)	LIMITSTRING (minimal amount of characters)	LIMITSTRING (minimal amount of characters)	SETSTRING (set definition)	SETDEFAULT-STRING (default value)
(identification-name)	SET (set of constant text elements, changeable)	(Comment)	SETSTRING (set definition)	SETDEFAULTSTRING (default value)		
(identification-name)	INT (Whole number)	(Comment)	LIMITSTRING (minimum value)	LIMITSTRING (Maximaler Wert)	INTDEFAULTSTRING (default value)	
(identification-name)	FLOAT (fractional number)	(Comment)	LIMITSTRING (minimum value)	LIMITSTRING (maximum value)	FLOATDEFAULTSTRING (default value)	INTSTRING (decimal places)
(identification-name)	STRING (text)	(Comment)	LIMITSTRING (minimal amount of characters)	LIMITSTRING (maximum amount of characters)	STRINGDEFAULTSTRING (default value)	

This structure is best explained by an example:.

Example:

```
IDConstType "Specify construction type:"
# IDValue STRING "Description" 1 30 LAST#
```

This attribute definition of the attribute CONSTRUCTIONTYPE leads to following dialogue box:

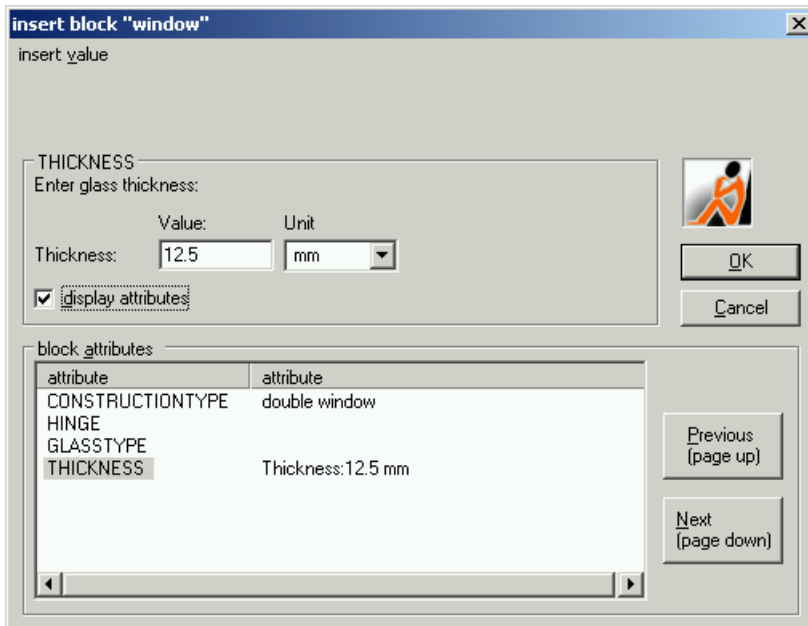


Only one element of the type STRING was defined. The parameter string of this type consists of the elements LIMITSTRING, LIMITSTRING and STRINGDEFAULTSTRING. The first two resemble the minimal or the maximal number of characters respectively which are intended for this attribute. In this example, text sequences between 1 and 30 characters can be entered. If the key word LAST is used as STRINGDEFAULTSTRING, it will be attempted to offer the entry that has been entered last as template. Conversely, NEXT is the successor of the last value, which is the next value for whole numbers. In text, 'Papu' becomes 'Papu1' and 'Papu3' becomes 'Papu4'.

Another attribute definition leads to the attribute **THICKNESS**:

```
IDThickness "Enter glass thickness"
# IDConst CONST "" "Thickness: " ;
IDWert FLOAT "Value:" 0.0 NOLIMIT LAST 1;
IDUnit USERSET "Unit" 1 3 {' mm', ' cm'} LAST#
```

This leads to following dialogue box:



Three elements have been defined. The first element IDConst of the type CONST puts out the constant text “**Thickness**”. The element IDValue of the type FLOAT allows the insertion of a decimal larger or equal to zero with one digit after the point. With the last element IDUnit of the type USERSET a list is generated, from which the units “mm” or “cm” can be selected, or an own value can be entered which has to be between one and three characters long.

Individually adapt FARO block templates

As-Built require, to be able to correctly run, various block templates, e.g. Instrument stations and control points, height symbols, various dimensioning blocks etc. These block templates are in the sub-folder \DWGBASE of As-Built¹.

For each block template there is a template drawing. They contain the individual AutoCAD entities (graphics and attributes), that belong to the particular block.

Open and edit these block template drawings as you require, the changes that have been made will always be used in later projects as the template.

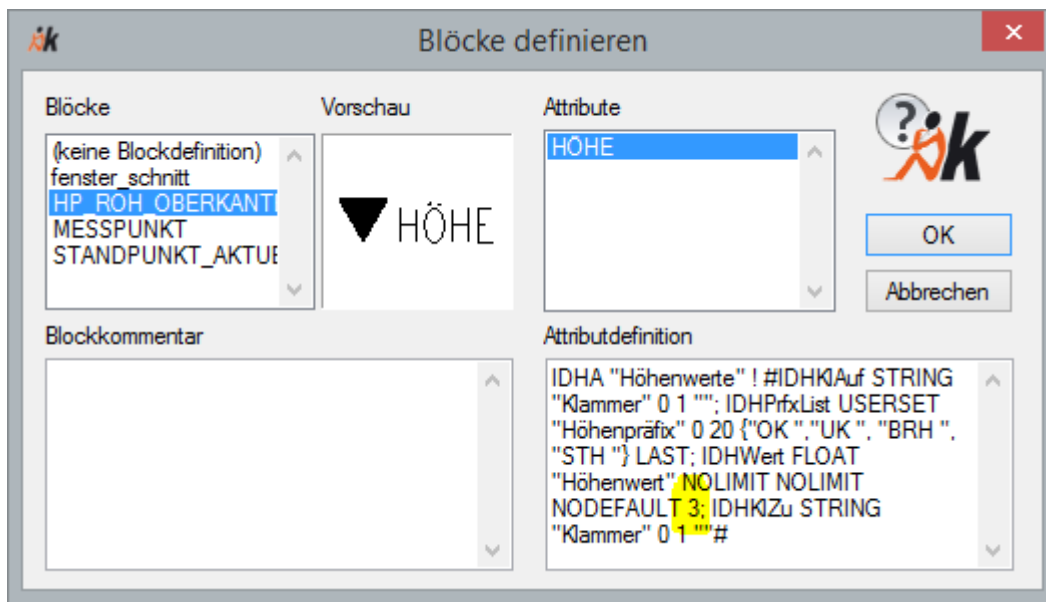
In this way you can, for example, adapt the number of decimal places or specific prefixes or suffixes in the attributes.

The number of decimal places of a height dimension should be adapted.




¹ You can also save them in a different folder, as long as the folder has been added to the AutoCAD support path (Pay attention to the order). You do that by using the AutoCAD Options command (Command “_OPTIONS” - Tab - “Files” - “Search path for support files”).

You must adapt the attribute definition in the following places:



IDHA "Height value" ! #IDHKITo STRING "Bracket" 0 1 "";	→ Definition opening bracket
IDHPrfxList USERSET "Height prefix" 0 20 {"OK ", "UK ", "BRH ", "STH "} LAST;	→ Definition prefix choices
IDHValue FLOAT "Height value" NOLIMIT NOLIMIT NODEFAULT 3;	→ Definition Height value From type FLOAT: NOLIMIT=Min.Value unlimited NOLIMIT= max.Value unlimited, NODEFAULT=no default value , 3=Decimal places
IDHKITo STRING "Bracket" 0 1 ""#	→ Definition closing bracket

Dimension – Subsequent editing – Adjust blocks to UCS

	Command Name:	PSBBLOCKADJUSTL
	Ribbon Tab:	As-Built Building Plans

This command is used for aligning FARO blocks with the XY plane of the current UCS.

The position of the block itself is not changed. However, the block is rotated in a way that it lies parallel to XY plane of the UCS.

Command: PSBBLOCKADJUSTL

Adjust all instrument and control points or selected [Blocks] <instrument and control points>:

(Select option)

If you confirm the selection "Control points" with **Enter**, all blocks named "CONTROL" (Control point blocks) will be adjusted.

However, if you select option "B" for "Blocks", you will be asked to pick objects:

Select blocks:


Select objects:

(Select object)

If you confirm the selection with **Enter**, all selected blocks will be adjusted.

Important: Some blocks and their position in space reflect a geometry, e.g. the north arrow. If you adjust such blocks with this command to an altered UCS, their correct position will be lost. You can avoid such unwanted effects by locking the respective layers during the processing of the command.

Dimension – Create rotated arch plan

	Command name:	PSHROTARCH
	Ribbon Tab:	As-Built Building Plans

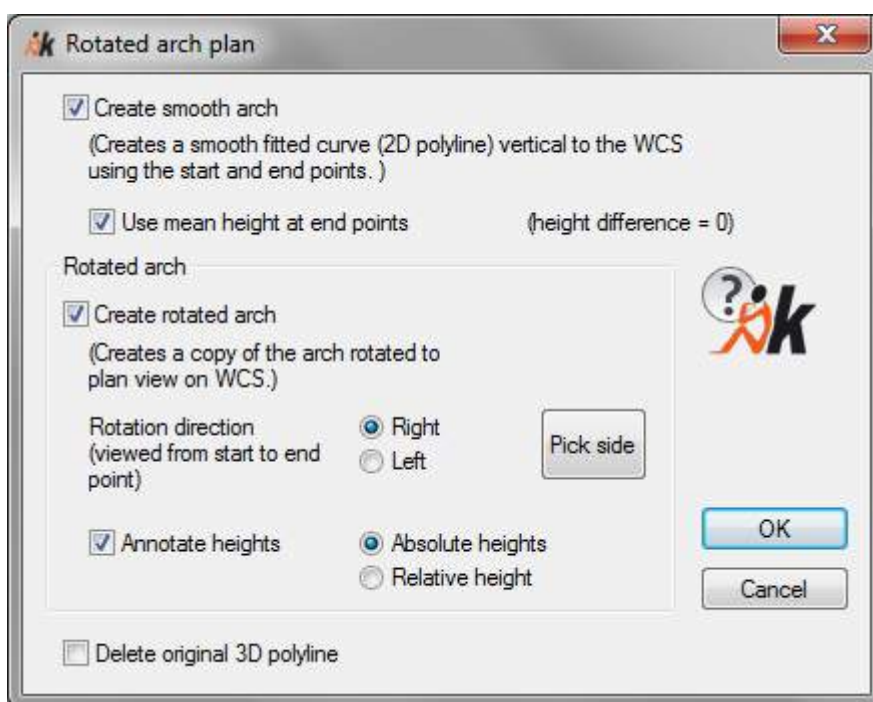
The command “PSHROTARCH” is used for the construction of irregular arcs, and for folding them into the ground map. It supports the description of contours which take the shape of an arc in the ground plan or front view (e.g. Arcs above windows and doors).

The command uses a 3D (or 2D) polyline as geometrical input information. The start and end point of this polyline must define the true position of the arc. Important: There should be a vertex of the polyline in the apex of the arc or close to it. The intermediate points are projected into the arc plane, which stands vertically on the XY plane of the UCS, and runs through the start and end point of the arc.

The command “PSHROTARCH” asks you to select a 3D polyline. The template for the object selection is the last polyline that had been created in the drawing.

Select polyline [lasT] <lasT>:

Now following dialogue appears:



Normally, the lowest height (for arcs pointing upwards) or the highest height (for arcs pointing downwards) are used for the rotation plane. The option “Use mean height at end points” mingles the heights of arc ends, which allows to ignore a height difference that exists due to building tolerances, but are not relevant for accuracy.


The annotations at the rotated arc consist of the arc apex (middle of arc) and the heights of the arc start and end points. However, if you have selected “Use mean height at end points”, only the height of the arc start point will be displayed.

The dimensioning of the middle of the arc (apex) always refers to the highest point of the adjusted arc created by As-Built Modeler. Following points need to be considered to ensure that the dimensioned value matches reality as good as possible:

- Measure enough points in regular distances along the arc. The more points you measure, the better the adjusted arc matches the real shape.
- If possible, also measure a point in (or close to) the apex of the arc. If this is not possible, measure the arc as symmetrical as possible.

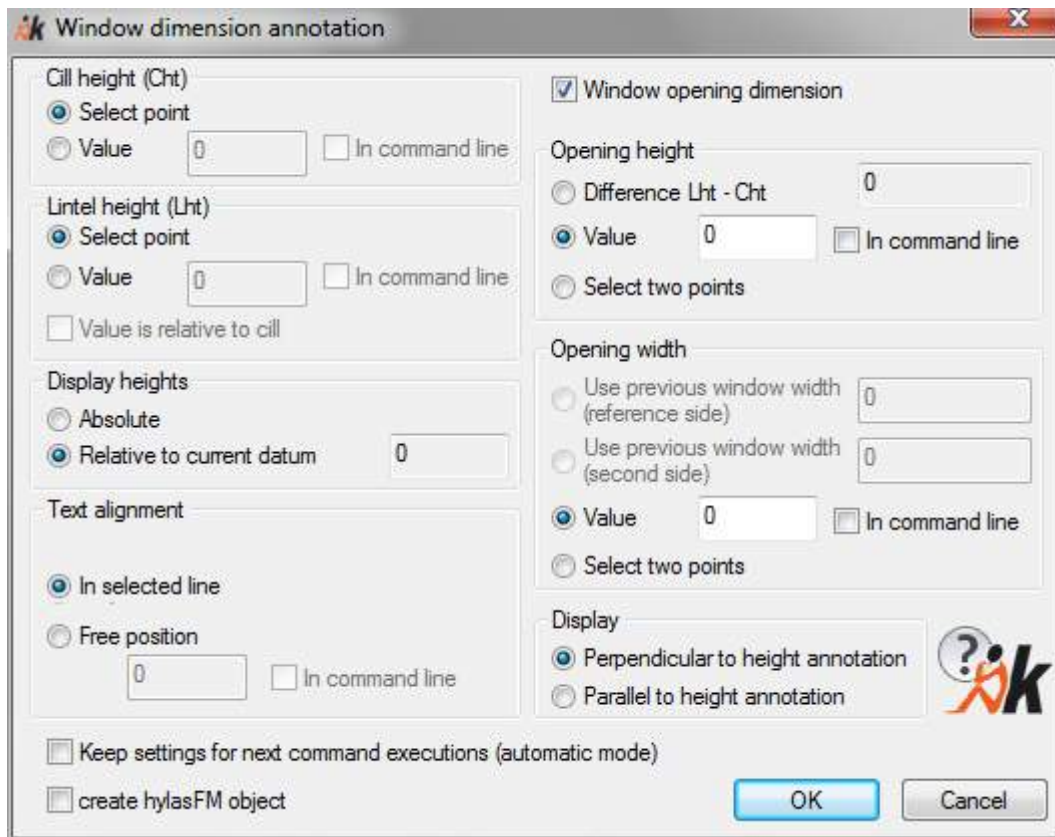
The adjusted arch is created on the same layer as the measured polyline. The rotated arc is set to the layer the name of which is set in the As-Built Modeler → **Settings** under the key “Layer” – “ArchPlanLayer”. If this value is kept empty, the current layer will be used. The key “ArchDimensionLayer” is used for the height dimensioning of rotated arch.

Dimension – Window dimension annotation

	Command Name:	PSBDIMWINDOW
	Ribbon Tab:	As-Built Building Plans

This command is used to dimension a window. The necessary values for cill and lintel height as well as – optionally - the opening height and width will be prompted.

Following dialogue appears:



Here you configure which measurements you want to indicate.

Cill height and lintel height:

The measurements can either be picked in the point cloud or inserted manually.

- **'Select point'** means that you will be asked for a point the Z coordinate of which represents the height.

Measure point on the cill (Click point for lower edge of window parapet)
 Measure point on the lintel or [UNdo]: (Click point for upper edge of window lintel)

- The option **'Value'** is used for the direct insertion of the height values. They are either inserted directly in the dialogue or will be asked later in the command line when **'In command line'** is activated:

Enter value for parapet height: **.8** (insert measured height)
 Enter value for lintel height or [Undo]: **2.9** (insert measured height)

The entry 'Display heights' determines which height the values for parapet and lintel height refer to.

Text alignment

Here you indicate how the dimensioning text is to be aligned.

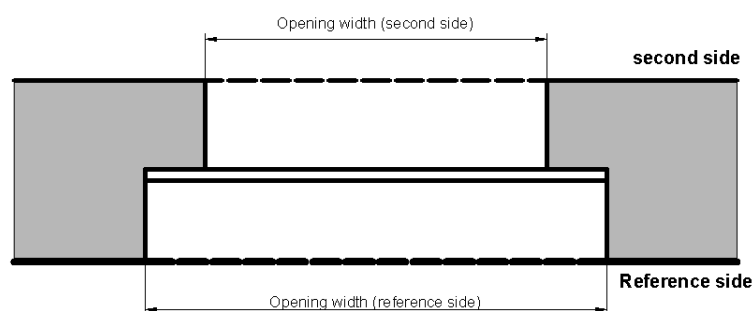
- **'In selected line'** means that you have to indicate a line at which the text is to be aligned.
- With **"Manually with statement of angle"** you may either insert an angle value for alignment or you click onto two points within the drawing ('Command line' option).
- With **'Free position'** you can either insert an angle value for the alignment or you click onto two points in the drawing. (Option 'Command line')

Opening height and Opening width

The dimensioning of clear height and width is optional. It is activated by ticking the field 'Window opening dimension'.

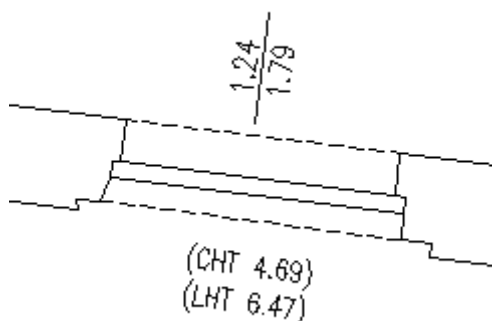
The clear measurements can either be measured or inserted manually.

- **Difference Lht - Cht:** This option automatically calculates the clear height from the difference between the parapet height Cht and the lintel height Lht. This is a default option. If for some reason the clear height does not correspond with this difference please use one of the following two options.
- **'Select two points'** means that you will be asked for two points in the command line. The distance of these two points represents the clear measurement. You may pick these points in the cloud or click on them within the drawing. The distance of two points in Z direction is used for clear height while the distance in the XY plane is used for clear width.
 - Measure first point for opening height or [UNdo]:
(Click point for lower edge of window)
 - Measure second point for opening height or [UNdo]:
(Click point for upper edge of window)
 - Height of opening: 2.34216 *(calculated clear height)*
 - Select first point for opening width or [LIne/UNdo] <LIne>:
 - Select second point for opening width or [UNdo]:
 - Width of opening: *(calculated clear width)*
- The option **'Value'** is used for directly inserting the clear measurement. It is either inserted directly in the dialogue or later in the command line if **'In command line'** is activated:
 - Enter value for clear height or [Undo]: **2.3** *(insert measured value)*
 - Enter value for clear width or [Undo]: **0.9** *(insert measured value)*
- The options **"Use previous window width (reference side)"** and **"Use previous window width (second side)"** for the statement of the clear width are active, when the dimensioning is opened via the **"Measure window"** command.

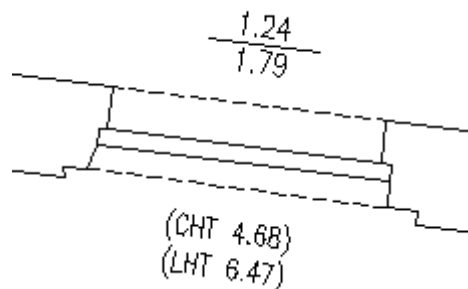


Display

You can control the text alignment of the opening annotations with 'Display':



perpendicular to height annotation



parallel to height annotation

Automatic mode

The Automatic mode is activated by ticking the field 'Keep settings for next measurements'. This option allows to dimension windows of the same kind without having to set all settings anew every single time. The dialogue will not be displayed at the next call up of the command. The settings will be taken over as they were set the last time. We recommend to use the option 'in command line' for inserting values in the automatic mode.

After the dialogue has been closed with OK, you will be prompted for the individual values and measurement points according to the settings. As an example, the above pictured dialogue setting leads to following command line sequence:

Command: PSBDIMWINDOW

Measure point on the cill or [UNdo]:

(Click a point on the window parapet)

Measure point on the lintel or [UNdo]:

(Click a point on the window lintel)

Height of opening: 1.325

Width of opening: 0.926852

Select line for text alignment or [UNdo]:

(Select a line)

Window dimension annotation

Insertion point of block:

Positioning of dimensioning block for parapet and lintel height)

Window opening annotation


(Positioning of dimensioning block for opening width and height)

The dimensioning blocks are created on the layer WINDOWDIMENSION by default. You can configure these pre-set layers individually for height dimensions and clear dimensions. Use the As-Built → **Settings** command.

Create feature data object

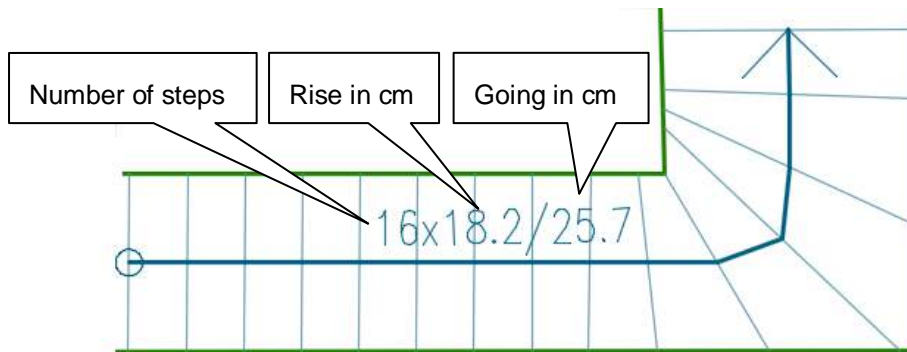
This option can only be applied, when you use the feature data management. If the used structure definition has a predefined class "Window", a feature data window object will automatically be generated additionally. The attributes "Height" and "Width" will automatically be filled with the opening measurements.

Dimension – Staircase dimension annotation

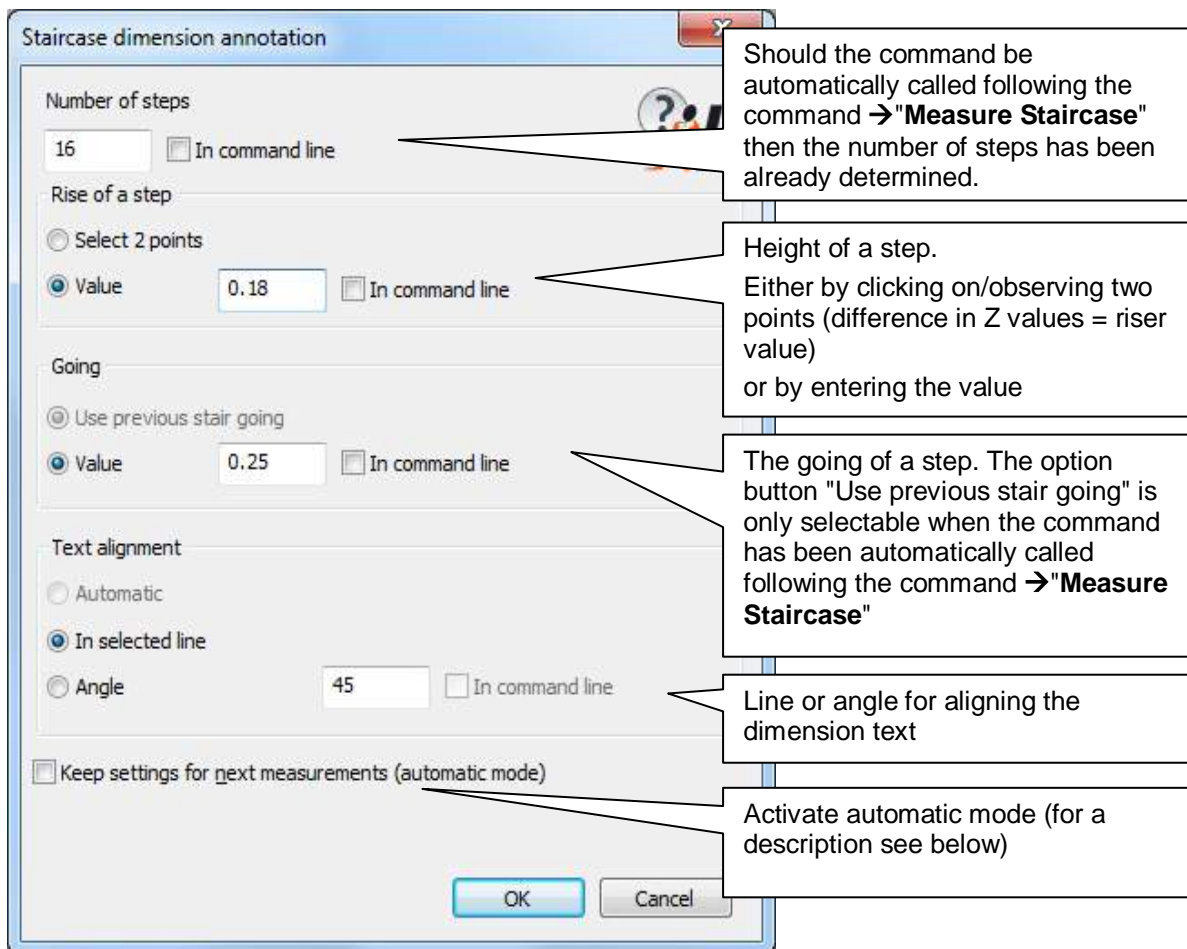
	Command Name:	PSBDIMSTAIR
	Ribbon Tab:	As-Built Building Plans

This command inserts a block with the block name "StairDimBlock" as staircase dimension into the drawing.

Three values are displayed as follows:



After starting the command the following dialogue box opens up:



Leave the dialogue box with [OK].

Depending on the selected options, there follows a few command line prompts.

For example:

Align dimension text to a line: *(Select line for the alignment of the dimension)*
Insertion point of block: *(Position dimensioning block with a mouse click).*

The block for the staircase dimensioning will automatically be inserted on the "STAIRCASEDIMENSIONING" layer.

Automatic mode


The automatic mode is activated when you ticked the box "**Keep settings for next measurements (automatic mode)**". This option allows you to consecutively dimension several stairways without having to enter new values each time. The next time the command is called the dialogue box will not be displayed. The settings will be used that you set last time. When using the automatic mode it is recommended that to enter values that could change (e.g. the number of steps) then use the option "in command line" for each item.

Individual customisation

You have the following possibilities to individually customise the appearance:

- You may adapt the attributes (output of text) to the dimension with the **→Define Extended block** command. That way you may change the number of decimal places or use the abbreviation "Rs" instead of the "x" sign.
- Layer: The preset layer may be changed with help of the **→Settings...** command. In order to do so change the "Layer – Stair –Annotation" entry.
- Unit: By default, pitch and going will be stated in centimeters. But you may also change the unit with the **→Settings...** command under the "As-Built Building Plans – Stairs – Dimension unit" entry.


Drawing – Simple HELMERT transformation

	Command Name:	PSBHTRAFO
	Ribbon Tab	As-Built Building Plans

This command runs a simple Helmert transformation without over determination. Only two pairs of points have to be assigned. The options 'shifting', 'rotation' and 'scaling' are always enabled.

Please read → **Drawing – Advanced HELMERT transformation** for further details on this command.

Drawing – Advanced HELMERT transformation

	Command Name:	PSBNTRAFO
	Ribbon Tab	As-Built Building Plans

With this command you may fit the plan geometry at a new position in the drawing. This may become necessary if one and the same object has been measured twice but in two independent measurements and the results of both surveys shall now be combined.

A 2D Helmert transformation with matching of elevations is done, which combines a two dimensional rotation within the XY plane of the world coordinate system (WCS) with a 3D shifting and a scaling of the selected plan geometry. The scale factor is the same for all three axes if the coordinate system.

The transformation parameters are calculated from a set of point assignments.

There are two ways of transformation. One way would be the calculation **with** an over determination (Advanced Helmert transformation **PSBNTRAFO**). The other one would a calculation with only two pairs of points (Simple Helmert transformation **PSBHTRAFO**).

After starting the command ,PSBNTRAFO' you first have to assign corresponding pairs of points from source and target geometry.

1. pair of points: point to be transformed [Finish]:
1. pair of points: target point:
2. pair of points: point to be transformed [Finish]:
2. pair of points: target point:
3. pair of points: point to be transformed [Finish]:
3. pair of points: target point:
4. pair of points: point to be transformed [Finish]:
4. pair of points: target point:
- ...
- n. pair of points: point to be transformed [Finish]: f

During the assignment of points you may customize the text height of the temporary point numbers using the command → **Drawing – Set text height**.

After having finished the assignment of points the command line displays a protocol that informs you about the possible versions of transformation.

Transformation with shift

1. pair of points: discrepancy(XY) = 0.065
2. pair of points: discrepancy(XY) = 0.146
3. pair of points: discrepancy(XY) = 0.089
4. pair of points: discrepancy(XY) = 0.112

Average square discrepancy (XY) = 0.011 (optimization criterion)

Average discrepancy(XY) = 0.103

Maximum discrepancy(XY) = 0.146

Transformation with shift and scaling:

1. pair of points: discrepancy(XY) = 0.082

2. pair of points: discrepancy(XY) = 0.078

3. pair of points: discrepancy(XY) = 0.069

4. pair of points: discrepancy(XY) = 0.128

Average square discrepancy (XY) = 0.009 (optimization criterion)

Average discrepancy(XY) = 0.089

Maximum discrepancy(XY) = 0.128

Scale factor = 1.14996

Transformation with shift and rotation:

1. pair of points: discrepancy(XY) = 0.076

2. pair of points: discrepancy(XY) = 0.140

3. pair of points: discrepancy(XY) = 0.098

4. pair of points: discrepancy(XY) = 0.059

Average square discrepancy (XY) = 0.010 (optimization criterion)

Average discrepancy(XY) = 0.093

Maximum discrepancy(XY) = 0.140

Rotation angle(XY) = -6.79165 gon

Transformation with shift, rotation and scaling:

1. pair of points: discrepancy(XY) = 0.093

2. pair of points: discrepancy(XY) = 0.063

3. pair of points: discrepancy(XY) = 0.082

4. pair of points: discrepancy(XY) = 0.078

Average square discrepancy (XY) = 0.006 (optimization criterion)

Average discrepancy(XY) = 0.079

Maximum discrepancy(XY) = 0.093

Rotation angle(XY) = -6.79165 gon

Scale factor = 1.14996

Pairs of points [Apply/ADd/Disable/Enable/New] <Apply>:

You may now add, enable or disable single pairs of points for the calculation. If you are satisfied with the displayed values of discrepancy, you select the option 'Apply' and proceed with selecting the objects to be transformed.

Select objects to be transformed:

Select objects: n found

Finally you select the mode of transformation. The options shifting, rotation and scaling can be combined.

Transformation (Shifting, Rotation, sCaling) [S/SC/SR/SRC] <SRC>:

Now the source geometry is displayed above the target geometry. When running the command again one may use the parameters and objects of the previous transformation or edit them.

Use last transformation parameters? [Yes/No] <No>: y

Pairs of points [Apply/ADd/Disable/Enable/New] <Apply>:


Transformation (Shifting, Rotation, sCaling) [S/SC/SR/SRC] <SRC>: vds

...

Remarks

- For the advanced Helmert transformation always at least 3 pairs of points have to be defined.
- The transformation that considers the scale has to be used with care. The scale is only calculated from the deformation within the XY plane but is eventually used for the Z coordinates too. This uniform scaling in X, Y and Z is necessary because most of the AutoCAD objects cannot be scaled in another way. Otherwise e.g. a circle would become ellipse, which is not desired. **Note!** Due to the alteration of the Z coordinates a transformation with scaling is not in every case suitable for 3D geometry.
- The colour of the temporary lines and point numbers for the visualization of the assigned points is defined by the colour of the current layer and has to be set before running the command. The numbers of the target points are displayed in brackets.


Drawing – Set text height

	Command Name:	PSBNTTEXTHEIGHT
	Ribbon Tab	As-Built Building Plans

With this command you may customize the text height for the temporary point numbers used during the Helmert transformation in case they are too big or too small.

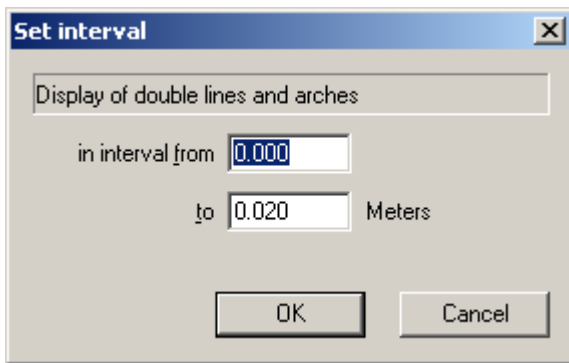
Also see → **Drawing – Advanced HELMERT transformation.**

Drawing – Plan analysis – Find double lines

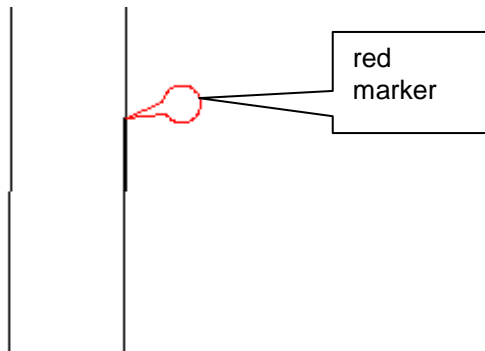
	Command Name:	PSBHYLASANALYSEDOUBLE
	Ribbon Tab	As-Built Building Plans

In some plans contain lines that have been drawn twice. These unattractive double draws can be detected using this command. They are marked by a red flag.

At first the following window opens.



Here you indicate a length interval. All lines whose maximum distance lies within this interval are marked. After confirmation with **OK** you are prompted to select the objects to be checked for double lines and arches. As-Built will now mark the double draws by a red symbol and you are able to revise the plan accordingly.



The size of the red marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command → **Drawing – Plan analysis - Purge**.

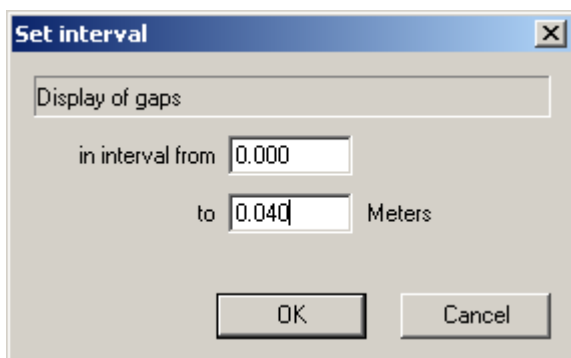
The search for double lines is an elaborate process of analysing the CAD graphics. It therefore may take some seconds (depending on the selected plan section) until the result is displayed.

Drawing – Plan analysis – Find gaps

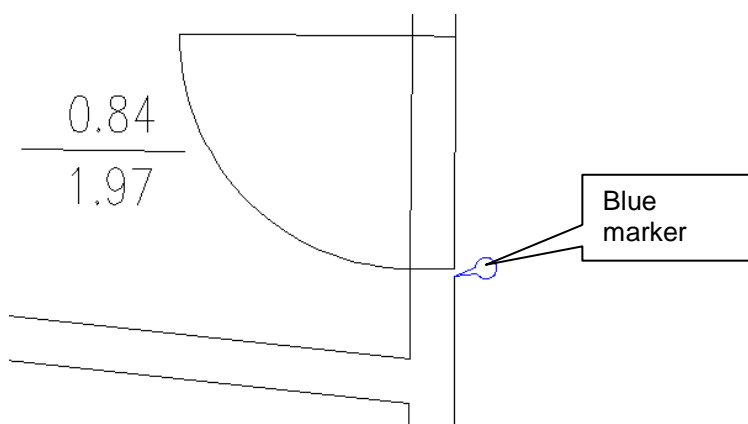
	Command Name:	PSBHYLASANALYSEGAP
	Ribbon Tab	As-Built Building Plans

Often there are plans where you will find small gaps between lines, which originate from drawing without using the AutoCAD OSNAP. Such gaps can be detected using this command.

At first the following window opens.



Here you indicate a length interval. All gaps that lie within this interval are marked. After confirmation with **OK** you are prompted to select the objects to be analysed. As-Built will now mark the gaps by a blue symbol and you may now revise the plan accordingly.



The size of the red marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command **→Drawing – Plan analysis - Purge**.

Why does the command prompt for an interval for the gap size and not for a maximum value?

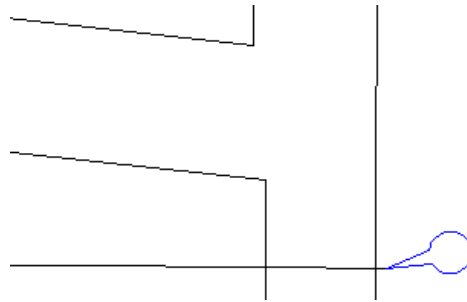
The answer is based on the two reasons that may cause you to look for gaps in a plan.

1. You simply want a neat plan. Then you really may set the lower limit of the interval to zero.
2. You want to use the feature data management to create polygons e.g. for rooms. The methods for outline tracing and pattern recognition do skip all gaps that are

smaller than a certain tolerance value. This tolerance value can be set using the As-Built command → **As-Built – Settings** (Feature data). Thus at least for working with As-Built you do not have to revise and remove the errors of the plan that are smaller than the tolerance.


That is why you may set the lower limit of the interval for the gap detection exactly to the value of the tolerance to ignore gaps. So only errors are displayed and need to be fixed that are larger than the tolerance value.

By the way, the gap analysis also detects small excess ends like in the following example.

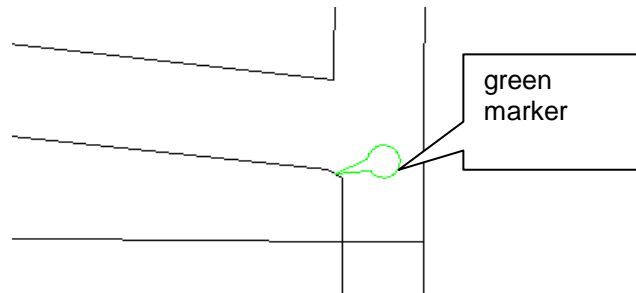


The search for gaps is an elaborate process of analysing the CAD graphics. It therefore may take some seconds (depending on the selected plan section) until the result is displayed.

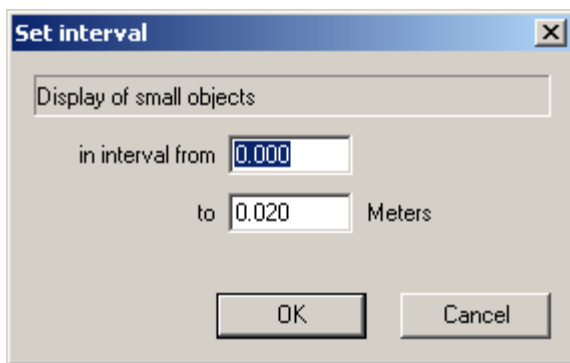
Drawing – Plan analysis – Find short lines

	Command Name:	PSBHYLASANALYSESHORT
	Ribbon Tab	As-Built Building Plans

You may find short lines using this command. They are marked by a green flag.



The length of the lines to be detected is prompted by this dialogue.




The size of the green marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command → **Drawing – Plan analysis - Purge**.

The search for short lines is an elaborate process of analysing the CAD graphics. It therefore may take some seconds (depending on the selected plan section) until the result is displayed.

Drawing – Plan analysis – Trace outline

	Command Name:	PSBHYLASANALYSEPATH
	Ribbon Tab	As-Built Building Plans

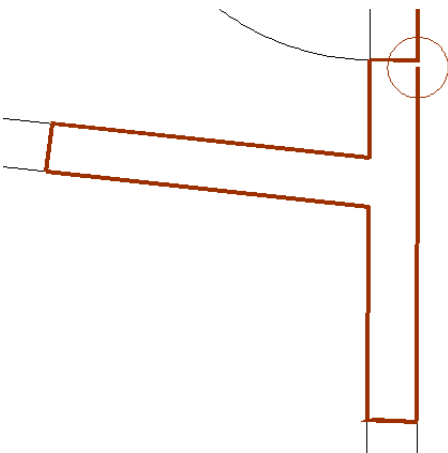
This command serves to trace outlines. It can be used to find gaps or to create closed polylines.

First the following prompt appears in the command line.

Plan analysis -

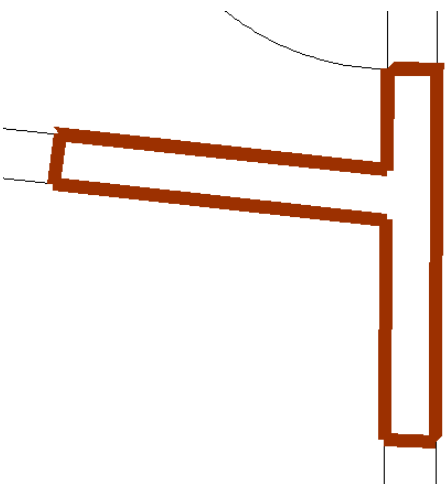
Click point next to line or [Configure]:

Now click close to a line in the drawing. The outline of this line is traced and marked in red. At arborizations the side of the line is preferred which you selected when clicking close to the line.

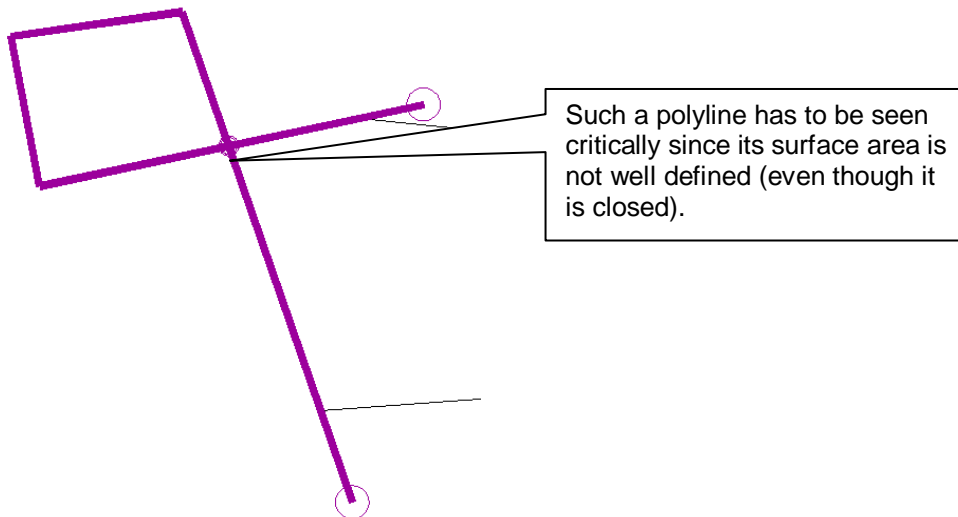


The ends of an outline are highlighted by circles.

Possible gaps within the line are ignored, provided they are smaller than a certain tolerance value. You can determine this value using the command line option 'C' (Configure). The width of the line marking also depends on this tolerance. In the following figure a larger tolerance value has been set and, thus, the outline tracing finds a closed polygon. When there is a closed polygon, no circles are displayed:



If the outline tracing results in a so called self-overlapping, the intersection points are marked by concentric circles. The outline line is also highlighted in purple.

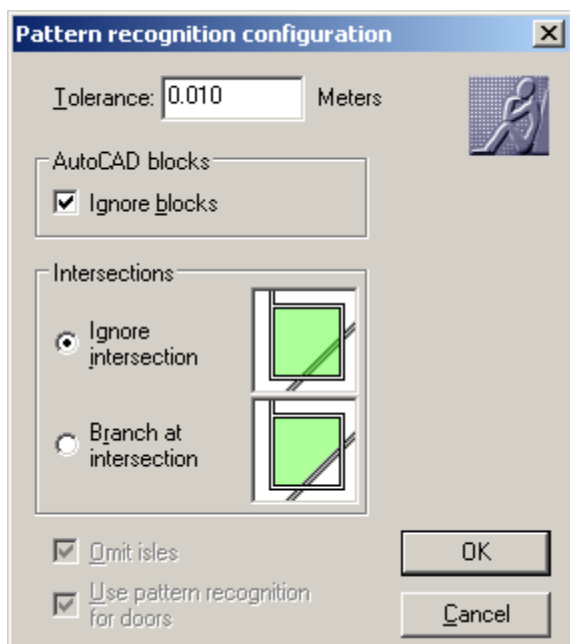


The outline tracing function runs a complex analysis of the CAD graphic. Therefore, the process may take a couple of seconds (depending on the selected plan section).

After you have traced an outline, you can run another analysis or take this outline over as AutoCAD 2D polyline. Following prompt will appear repeatedly in the command line until you cancel the command.

Click point next to line or [insertPolyline/Configure]:

The option 'Configure' opens following dialogue.

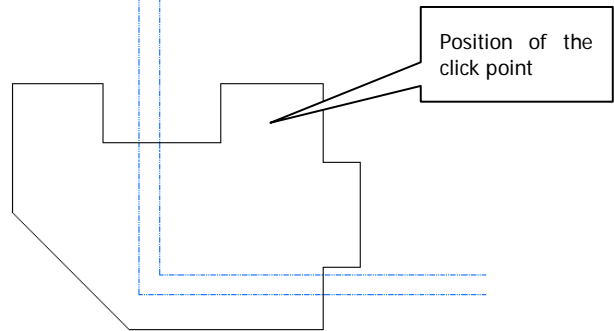
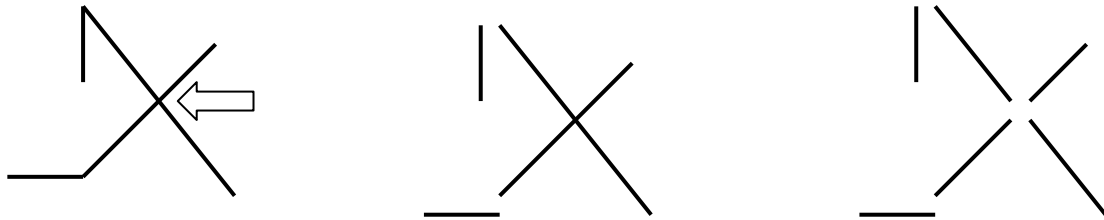


- **Ignore intersection** means that the outline tracing does branch at true X intersections.
- **Branch at intersection** means that the outline tracing will also branch at true X intersections.

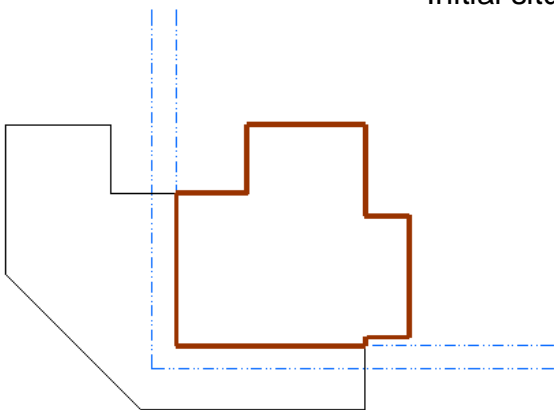
We speak of true X intersection when the AutoCAD elements intersect instead of just meeting.

Intersection situation in the drawing (4 or 6 lines?)	Exploded view	
	true X intersection (4 lines)	pseudo X intersection (6 lines)

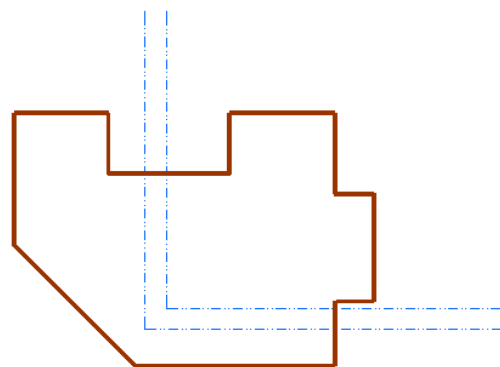
Here an example for the effect of the two modes with the same click point.



Initial situation




Intersections enabled



Intersections disabled

The background of the two modes is, in most CAD plans pseudo X intersections are used for objects that are connected with each other by content (e.g. adjoining walls and windows or doors). Things that overlay the actual plan (e.g. center lines and building grids) are often simply drawn through. Then real X intersections are created. Thus, in most cases it is better to ignore X crossing lines when creating room polygons and the like.


Drawing – Plan analysis - Purge

	Command Name:	PSBHYPURGE
	Ribbon Tab	As-Built Building Plans

This command deletes all AutoCAD objects that As-Built created temporarily. Such objects originate from the As-Built commands for plan analysis (→**Find gaps, find short or double lines**).

Also object markings (dashed line display), as they are created when clicking an object in the area list or in the feature list, are removed.

Drawing – Clipboard – Copy

	Command Name:	PSBHYLASCOPY
	Ribbon Tab	As-Built Building Plans

With this command you can copy any drawing objects to a clipboard. It works similar to the Windows clipboard.

In contrast to the AutoCAD clipboard commands the As-Built clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the As-Built clipboard copies/inserts FARO labels, that means data objects of the feature data management

The command line will read as follows:

Select objects to be copied

Select objects: *(Select any drawing objects)*

You are asked to select any geometry, to click on feature data labels and/or entries of the structure view. By doing so, these objects will be copied into a special clipboard.

You will be prompted for a reference point afterwards. This point is needed for the positioning of the objects in the target drawing. If the objects are to be placed at exactly the same position as in the original drawing please select the option 'Origin'.

Modes: Also copy forward linked objects (On), Also copy linked objects without label (On)

Base point[Origin/Linked/WithoutLabel] <Origin>:

At the same time you may change the modes of the implicit object selection.


Also copy forward linked objects	Additionally to the feature data objects selected manually, all polygons, which are related to the above objects, will be selected.
Also copy linked objects without label	Additionally to the manually selected feature data objects, all objects, which are linked and have no label, will be selected. This option is not relevant for the feature data management.

After stating the reference point you will be informed about the number of the implicitly selected objects if needed and then the command will be ended. You may now change to the target drawing and insert the objects.

See also → **Drawing – Clipboard – Paste**

→ **Drawing – Clipboard – Cut**

Drawing – Clipboard – Cut

	Command Name:	PSBHYLASCLIP
	Ribbon Tab	As-Built Building Plans

With this command it is possible to cut any drawing objects out of AutoCAD drawings. It works similar to the Windows clipboard.

In contrast to the AutoCAD clipboard commands the As-Built clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the As-Built clipboard copies/inserts FARO labels, that means data objects of the feature data management.

The command line will show the following:

Select objects to be cut

Select object: *(Select any drawing objects)*

You are asked to select any geometry, to click on feature data labels and/or entries of the structure view. By doing so, these objects will be moved into a special clipboard.

After that you will be prompted for a reference point. This point is needed to position the objects when inserting them into other drawings. If you want to position the objects at exactly the same position as in the original drawing select 'Origin':

Modes: Clip forward linked objects (On), clip linked objects without label (On)

Base point [Origin/Linked/WithoutLabel] <Origin>:

At the same time you may change the modes of the implicit object selection.

Clip forward linked objects	Additionally to the feature data objects selected manually, all polygons, which are related to the above objects, will be selected.
Clip linked objects without label	Additionally to the manually selected feature data objects, all objects, which are linked to these objects and have no label, will be selected.


After selecting the reference point you might be informed about the number of implicitly selected objects and after that the command is finished. You may now change to the target drawing and insert the objects.

See also → **Drawing – Clipboard – Copy**

→ **Drawing – Clipboard – Paste**

Important remark: The AutoCAD function 'Undo' (undo last command) does not work here. As-Built labels that have been cut cannot be reinserted.

Drawing – Clipboard – Paste

	Command Name:	PSBHYPASPASTE
	Ribbon Tab	As-Built Building Plans

With this command objects from the special As-Built clipboard will be inserted into the current drawing. This works similar to the Windows clipboard.

Previously the objects have to be imported into the clipboard by the commands → **Drawing – Clipboard – Copy** or → **Drawing – Clipboard – Cut**.

In contrast to the AutoCAD clipboard commands the As-Built clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the As-Built clipboard copies/inserts FARO labels, that means data objects of the feature data management.

When starting this command you will be asked for the coordinate system, that should be used:

Paste clipboard objects with their original Wcs or Ucs orientation [Wcs/Ucs] <Wcs>:

Confirm the standard option “WCS” by pressing ENTER, if you want to insert the graphic on the same position as in the original drawing, independent from the current UCS.

After that you are prompted to enter the reference point:

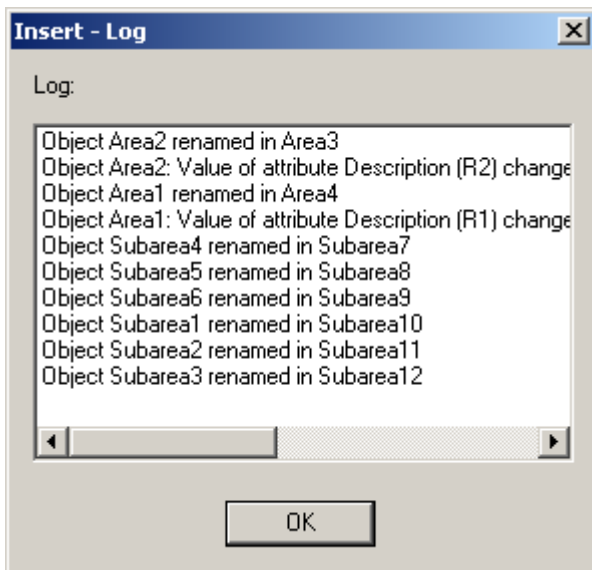
Base point[Origin] <Origin>:

If the objects are to be placed at exactly the same position as in the original drawing please select the option ‘Origin’ .

If there were feature data objects (FARO labels) cut or copied besides the pure geometry As-Built will try to insert them in the new drawing. Therefore the structure definition of the copied or cut objects has to be compatible to the definition of the target file. If necessary, the missing attributes will be created. The IDs of the data objects and polygons as well as the key attributes of the inserted objects will also be adapted where necessary, so the uniqueness will consist. When the copying was finished successfully a protocol will show the changes.

If an structure view has not yet been generated for the target drawing, the minimal structure definition will be created. This definition can include the new objects and where applicable the existing links between those objects.

In case the structure definition of the source drawing is incompatible to the definition of the target drawing an error report will appear, e.g.:



We speak of incompatibility when objects cannot be transferred without losses. That is the case if for example an object of the "Area" class with an integer attribute "RoomNumber"=11 is to be copied into a drawing, which does have a class "Area" with an attribute "RoomNumber", but the attribute has been defined as Text attribute. The structure definition of the target drawing will only be extended, never changed or reduced.

See also ➔ **Drawing – Clipboard – Copy**
 ➔ **Drawing – Clipboard – Cut**

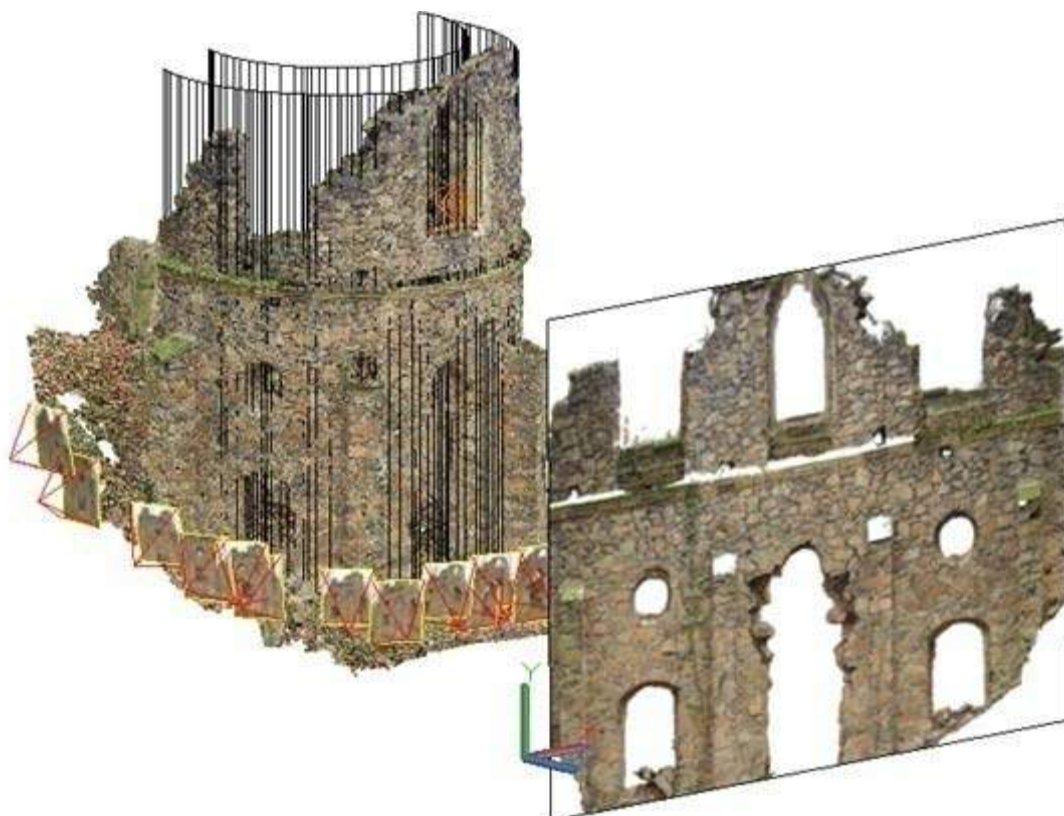
10.6 As-Built Photogrammetry Tools

Photogrammetry Tools - As-Built Photo

	Ribbon Tab:	As-Built Photo

The comprehensive photogrammetry tools provided by As-Built are used for photogrammetric image processing directly in AutoCAD. They enable the true-to-scale rectification of digital photographs, maps and plans as well as their subsequent interpretation and evaluation. Facade photographs, pictures of wall and ceiling paintings or floors can be processed, but also old maps and plan material can be brought back into a true-to-scale form. The results are true-to-scale image plans that combine photographic condition documentation with precise geometric information.

Other tools allow you to orientate photos by means of control points and to use these oriented photos for both 2D and 3D image evaluation. Photos of non-planar facades can be processed into true to length image plans. This creates three-dimensional models or 2D image plans.



Further information about the photogrammetry tools of As-Built can be found in the [As-Built Photo manual](#), which is part of your installation. You may find a link on the As-Built Photo ribbon tab under “Help”.

11 Adaptation and Configuration of As-Built for AutoCAD

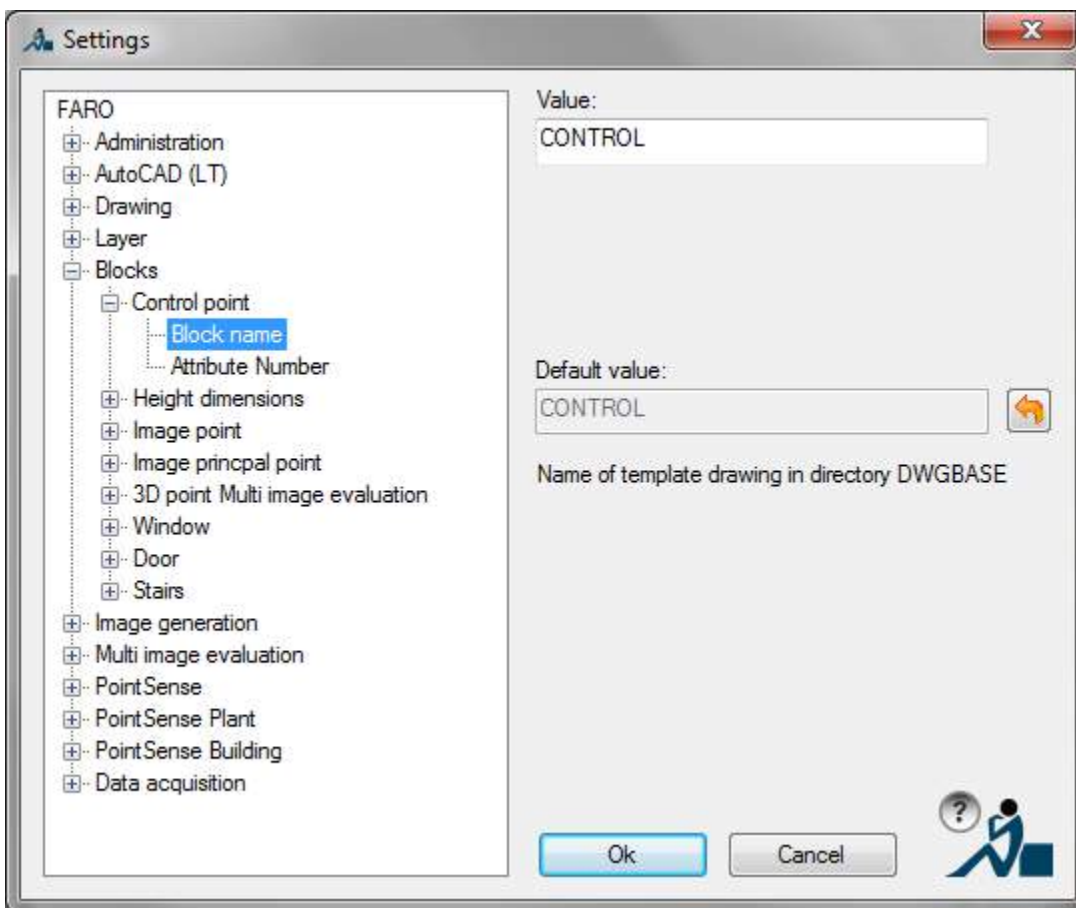
In this section you will learn how you may adapt As-Built for AutoCAD to your specific requirements.

11.1 Predefined blocks

You will find the templates for blocks needed by As-Built for AutoCAD within the installation directory, DWGBASE subdirectory.

These Blocks will be used by As-Built for AutoCAD for specific symbols (e.g. height dimension points, control points) or annotations (e.g. door and window dimensioning)

A complete list of all blocks used by As-Built for AutoCAD you will find in the As-Built for AutoCAD settings dialogue in the section “blocks”.



The block templates can be adapted in their appearance to meet your own requirements.

For some of the blocks it is possible to change the block name and the attribute names. In order to do so please refer to the As-Built for AutoCAD settings dialogue

→ **As-Built Settings**

11.2 Working with template files

An AutoCAD Template file (.DWT) can be used to predefine a whole set of settings, e.g. layer, text and dimension styles, units, as good as layouts with paper formats, stamps etc. .

You can save a big amount of time during the setup of your drawings by predefining your own template files for different types of projects. Also that makes sense for achieving consistent company standards.

Blocks can be predefined in DWT's as well. If the As-Built for AutoCAD blocks are defined in the DWT, the software does not access to the block template directory \DWGBASE anymore.

If you want to modify the As-Built for AutoCAD blocks for specific situations, please follow these steps:

- Generate a new AutoCAD drawing template (DWT)
- Import all block to be edited
- Modify the blocks
- Save the template to the current AutoCAD template directory

You may now derive all new drawings from this template.

11.3 Configuration with company guidelines

Larger companies often want a uniform configuration of all their workstations. For FARO 3D Software products it can be done as follows:

All configuration values like for instance layer names can be given as corporate standard. To do so, you customize the according entries in the "Settings"-dialog and close AutoCAD.

In the Windows Explorer brows to the application data directory:

C:\Users\[user name]\AppData\Roaming\FARO.

There you will find following files:

FaroConfigCurrentUser_en.cfg (language-dependent values)

FaroConfigCurrentUserCommon.cfg (language-independent values)

When opening these files with a text editor (format: UTF-8), it will look like this (here an example of the **FaroConfigCurrentUser_en.cfg**):

```
{
  { "kubit configuration file version 1.1"
  },
  {
    { "Key_Layer_InsertedImages",
      "[corporate layer name]"
    },
    { "Key_Layer_UndistortedImages",
      "[corporate layer name]"
    }
  }
}
```

```
    },  
    { "Key_Layer_RectifiedImages",  
      "[corporate layer name]"  
    },  
    ...  
    { "Key_Hylas_Pathes_Tools",  
      "Tools"  
    }  
  }  
}
```

One single configuration entry consists of one text block in curly brackets, a key and a value:

```
{ "Key_Layer_InsertedImages",  
  "[corporate layer name]"  
},
```

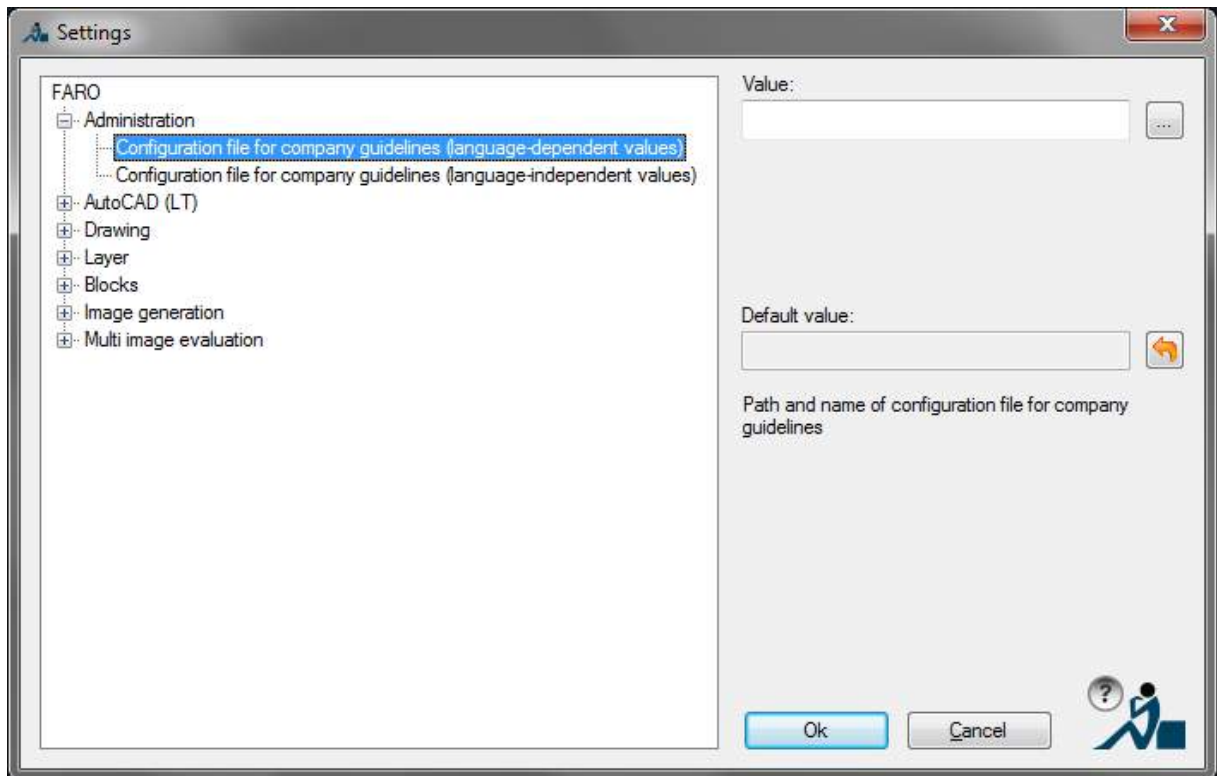
(Please note that the last block of a configuration file must not end with a comma.)

Remove all blocks from the file which you do not want to set as corporate standard. Please contact FARO Support if you are not sure about the keys in the configuration file or if you cannot find them.

In the **FaroConfigCurrentUser_en.cfg** file you will particularly find block and layer names while **FaroConfigCurrentUserCommon.cfg** file contains system settings like for instance the drawing unit.

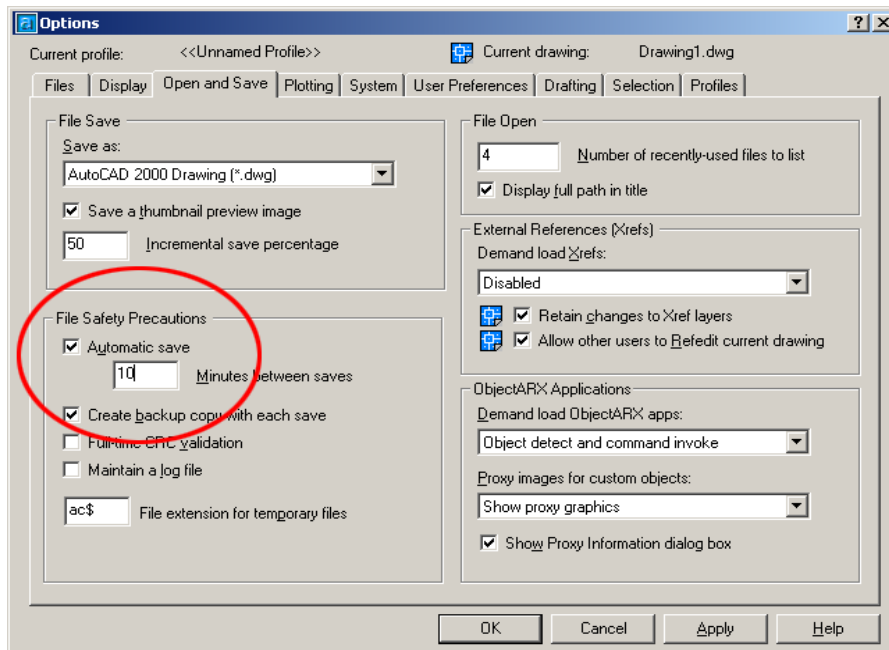
Now save the customized corporate configuration files under a meaningful name on a network drive all colleagues have access to. Set the file's properties to "read only" for all users.

All colleagues who shall work with the corporate settings have to select the new configuration files within the settings dialog of their FARO 3D Software product in the section „Administration/Configuration file for company guidelines“. Then re-start AutoCAD once more. The corporate default settings should now be taken over. Of course each user can change these setting during his AutoCAD session but with every re-start of AutoCAD they will be reset to the corporate defaults.



11.4 Useful AutoCAD Settings

- The automatic save should be set to a very low value (e.g. 10 minutes) in order to keep the loss of data during a possible system crash as low as possible. You may change this setting this way: call up "_config" command within the command line and check settings under "Open and save" → "File Safety Precautions".



- You may use the command "GRAPHICSCONFIG" to check on graphics performance. AutoCAD is automatically searching for graphic cards updates.
- As-Built turns off the Selection Preview (system variable SELECTIONPREVIEW). This is necessary because otherwise AutoCAD tries to highlight the point cloud when moving the mouse over it.
- As-Built turns off the Dynamic Input (system variable DYNMODE). This is necessary because otherwise the automatic regeneration of the point cloud in the "Level of Detail" mode does not function correctly.

12 License Contract

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 softlock or a dongle. The dongle or softlock contains complete licensing information. Therefore your licence is irremovable embedded in your dongle or your computer. 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 comes closest 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.

Supplementary Provisions for Use with AutoCAD LT

FARO Software uses LTX under licence from ManuSoft (<http://www.manusoft.com>) for the required software support, to be able to load and run FARO Software from within AutoCAD LT®. You are hereby granted permission to use for an unlimited period of time, the software LTX as an integral part of the FARO Software. However permission has only been granted exclusively for the use of LTX in connection with the FARO Software or which it has been developed, planned and approved. Any other use of the LTX files is expressly prohibited, and every violation of this Licence Agreement will be prosecuted to the full extent that the law provides. Neither ManuSoft nor any possible heir or legal successor guarantees implicitly or explicitly the applicability or the freedom from defects of LTX when used by the end user or any third party. ManuSoft disclaims all liability for any direct, indirect, special, incidental or consequential damages in any connection with LTX.