As-Built
for AutoCAD® Software

Total Station Interface

Manual
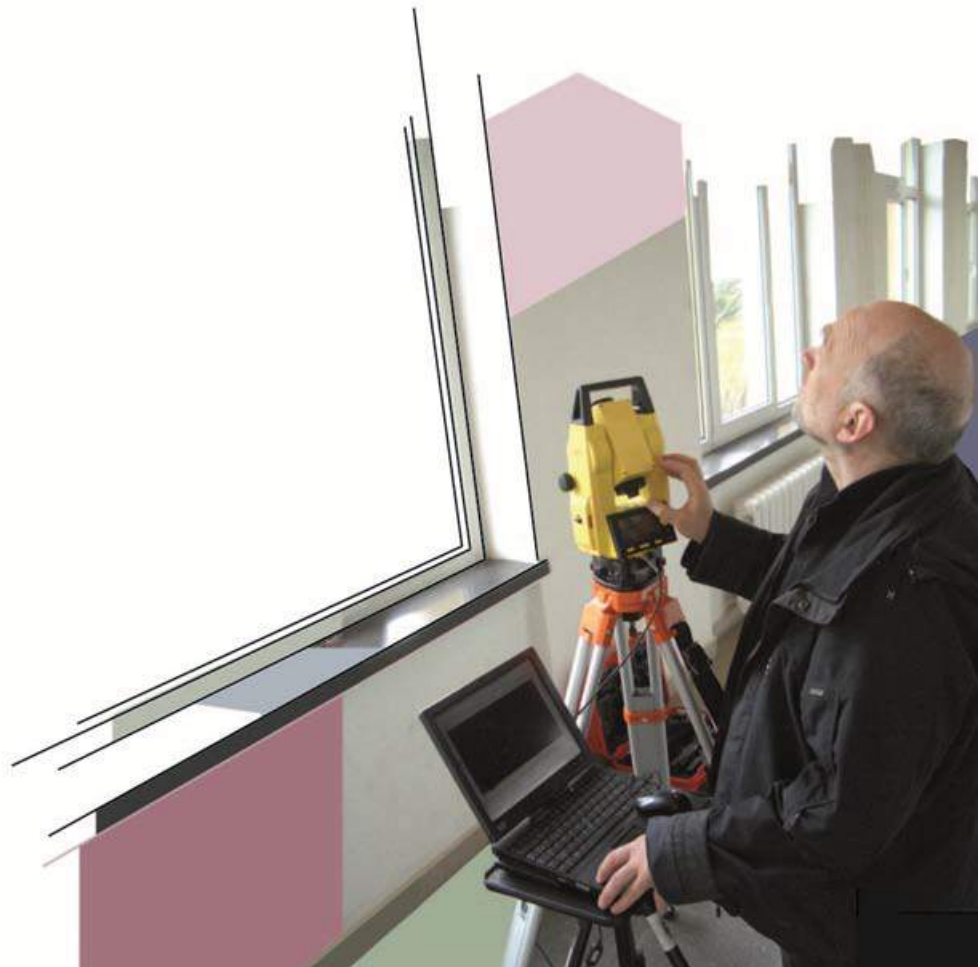
FARO®

SYNERGY
POSITIONING SYSTEMS

As-Built TS 2018.0

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A few Words concerning this Manual

Dear user,

We would like to thank you for the trust you put into us and wish you much success in the daily work with our software.

This manual is part of the As-Built for AutoCAD user documentation. **Please refer to the** As-Built for AutoCAD manual **for installation and system requirements**.

In order to provide an optimal start for the work with the total station interface, this manual contains a tutorial that lead you through typical project runs step by step.

Be sure we at FARO gave our best to develop this piece of software to support you and your work. We are aware that the combination of different components (i.e. total station, notebook, software) has to lead to best results under any circumstances. Nevertheless, we are not perfect. So please do not hesitate to contact our support service if you have questions or in any case of problems. Your questions and suggestions are important to us and are always welcome. Our software gains from the ideas and the creativity of its users.

Content

1 Working with the total station interface – first steps

This unit introduces you to the most important commands of the As-Built total station interface. Thus, we will not explain all commands in detail. Many points will be explained in further detail later on in this manual.

Our experience shows that working with instrument, laptop and As-Built for AutoCAD for the first time is not always easy. Please do not hesitate to ask for our support if something appears illogical or does not work out as you would have expected

1.1 Select type of instrument

We assume that your instrument is already set up and that it is connected to your laptop via data cable or Bluetooth. You have already installed As-Built for AutoCAD**.**

Start AutoCAD and select file TACHYCAD_ENG.DWT as a template.

Your first step is to chose the type of instrument you are working with. Select menu item ”**Select instrument”** on the As-Buil TS Control ribbon tab under Total Station. Following dialogue appears.
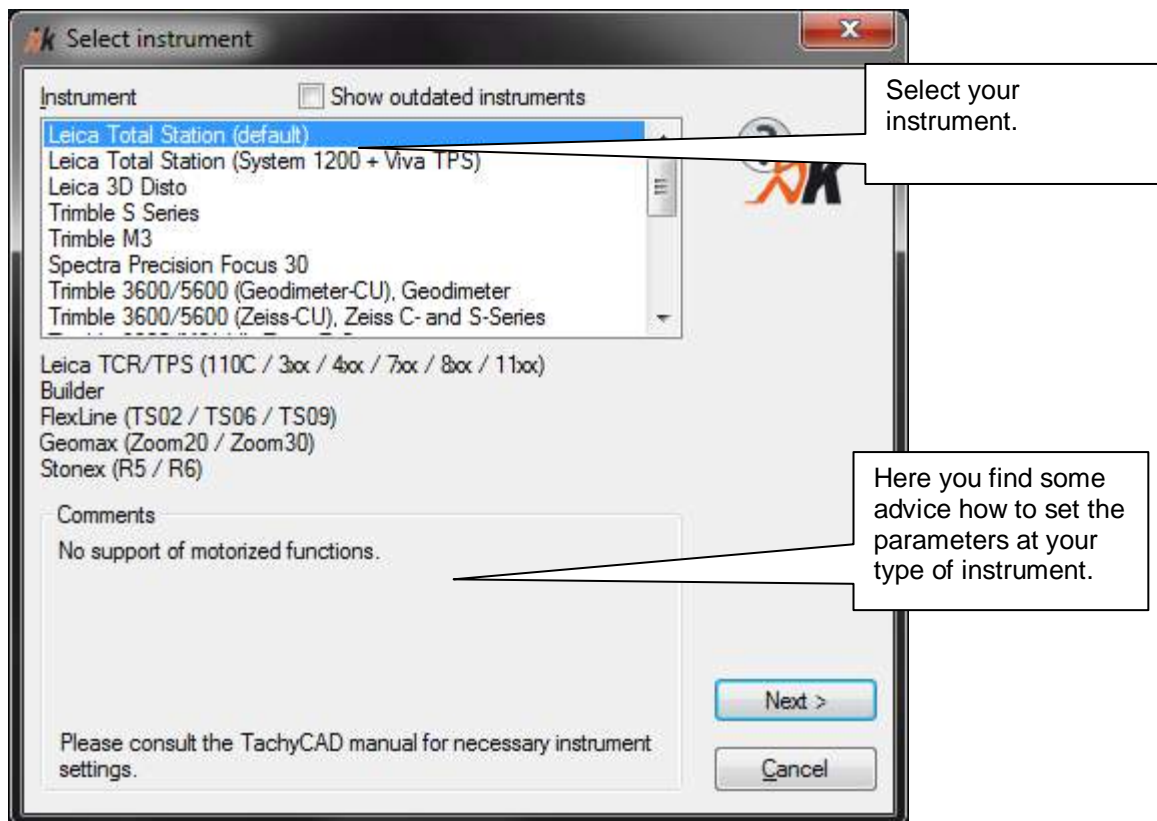
One click on **Next >** leads you to the settings for the communication interface and the units. Depending on the selected connection type the dialog looks as follows:

Bluetooth

You will find instructions for setting up the Bluetooth environment of your operating system in chapter **11.2 Bluetooth Connection**.

Close the settings dialogue box by clicking the **OK** button.

Cable

Under "Serial Port" select the COM interface via which your instrument is connected to the computer. Make sure the interface parameters (baud rate, data bits, stop bit and parity) of your instrument are set identical.

You will find further detailed information regarding supported instruments and necessary settings in chapter **11 Advice regarding supported instruments.**

Close the dialogue with **OK**.

1.2   Quickly define an instrument station

Before your first measurement you have to define an instrument station. Just follow the steps described here. You will find detailed information concerning different possibilities of defining an instrument station in chapter **3.4 Instrument station, control point, orientation** and under →**Orientation and control points - New/Edit orientation…** in the command reference.

We use a local system in our example. That means that the position of the instrument in AutoCAD coincides with the origin of coordinates in the WCS (AutoCAD World Coordinate System). The direction of the Y axis corresponds to the direction 0°00'00" at the instrument.

Select "New/Edit orientation" in the menu "**Orientation**". Then select "Local orientation" in the following dialogue
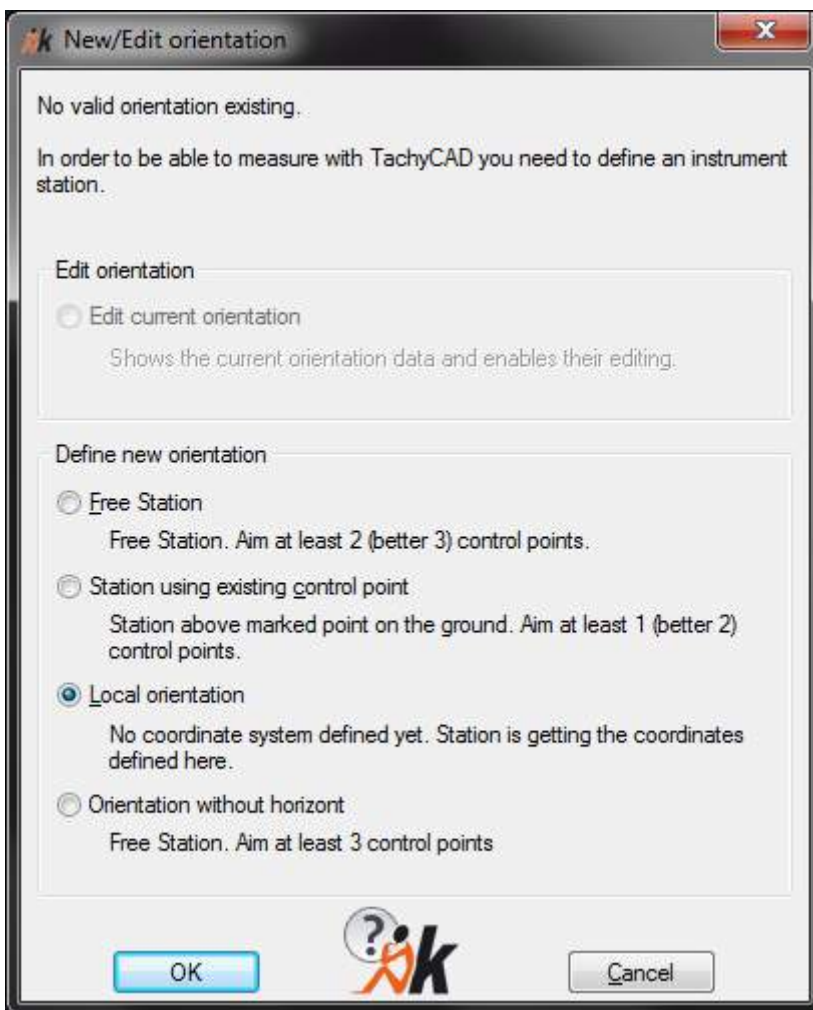
Close the dialogue with **OK**.

Afterwards you can number your instrument station. In our case just adopt the suggested number. To make things easier, we do not enter height data and coordinates at this stage. Just confirm suggested numbers with **OK**.

You find detailed information concerning positioning in the command reference under →**Orientation - New/Edit orientation….**

When you zoom in the origin of coordinates in AutoCAD now, you will see a symbol for the instrument station you have just defined.

1.3   How does the measured point get into AutoCAD?

If you want, you can now start working with the As-Built TS interface. As an example you want to measure the intersection line of a wall. Maybe you want to draw it as a line in AutoCAD. Select the tool icon for the command "Line" or directly insert the command in the command line.
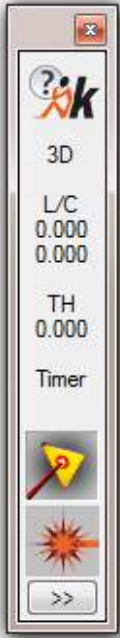
Command: **line**
Specify first point: *(AutoCAD asks you for the start*
                                                                                     point of the line.)

AutoCAD now asks you for the coordinates of the start point of the line. There are numerous possibilities of defining a start point in AutoCAD. For example you can either click a start point with the mouse or insert the coordinates manually.

We want to measure this point with our instrument.

You have probably already noticed the slim dialogue box on the left side of the screen. We call the dialogue "measurement box".

The measurement box symbolizes the connection between instrument and AutoCAD. When you see the measurement box on the screen, you can use the instrument within AutoCAD for measurement.

There will be detailed information concerning the measurement box in chapter 8 Command Reference (As-Built Total Station Interface)**.**

Now aim at the corner of the wall with the instrument and initiate the measurement and registration at the instrument. As-Built for AutoCAD calculates coordinates using the measured direction and the distance to the end point and forwards these to the command line of AutoCAD

Specify first point: **3.889,-2.365,0.261**   *(coordinates of measurement)*
Specify next point or [Undo]:

Now you can measure the next corner of the wall:

Specify next point or [Undo]: **4.819,-1.361,1.263**

As you can see handling As-Built for AutoCAD is quite simple.

Whenever AutoCAD expects the coordinates of a point, you can measure these with your instrument and thus forward them to the command line of AutoCAD.

This means that you are not limited to a certain graphic element e.g. a line or a point in the measurement.

If you look at the line you have just measured, it seems to be aligned in no order within the AutoCAD drawing window since at this time all measurements refer to the coincidentally defined local orientation. You will find information on how to control alignment in **3.2 Ground plan, vertical section, view – everything is possible with UCS**.

If you want, you can now try out a few measurements. For example try to change the layer or the element type.

If you sometimes should be to fast and accidentally initiate a measurement before having run a command As-Built for AutoCAD will warn you by making a special

sound. Besides this **all** measured values are kept in a stack (buffer). Using the command →**Measure – Show instrument values** you may display these values and use them ex post for constructing in the CAD. You find more detailed information within the command reference.

2 Working with the laser distance meter – first steps

By means of the most important commands, this chapter introduces you to the work with a laser distance meter and As-Built. Not all details will be explained though. According to experience the interaction of notebook, Bluetooth and laser distance meter is not always easy. If some things do not seem completely logical to you or if certain things do not work as you expect them to please do not hesitate to contact us for support.

2.1 Establishing the connection between laser distance meter and As-Built

Set up Bluetooth

The Bluetooth conection controls the appropriate driver software. Dependng on the operating system being used the procedure can be a little different.

Roughly, these steps are necessary:

- Switch on the hand held laser and activate the Bluetooth

- Activate the Bluetooth on the computer

- Using the Bluetooth function of the operating system search for devices

→ Detailed instructions can be found in the section "Details for supported hand held lasers".

Configure the interface settings

Start AutoCAD with As-Built and open any drawing.

To establish a connection you must tell As-Built the type of your laser distance meter. To do this open the hand held laser's settings dialogue:

"Hand held laser – Connect hand held laser" or by using [icon] .
In der Hand held laser-Box click on the double angle brackets **>>**.

On the "Connection" tab select the connected device (e.g. "Leica DISTO") and then click on "Connect". If everything is correct, the connection to the hand held laser will be established and the prompt changes to green.

Now take a look on the tab "Sending options". For now select the following settings:
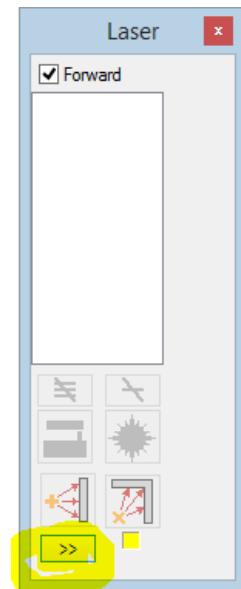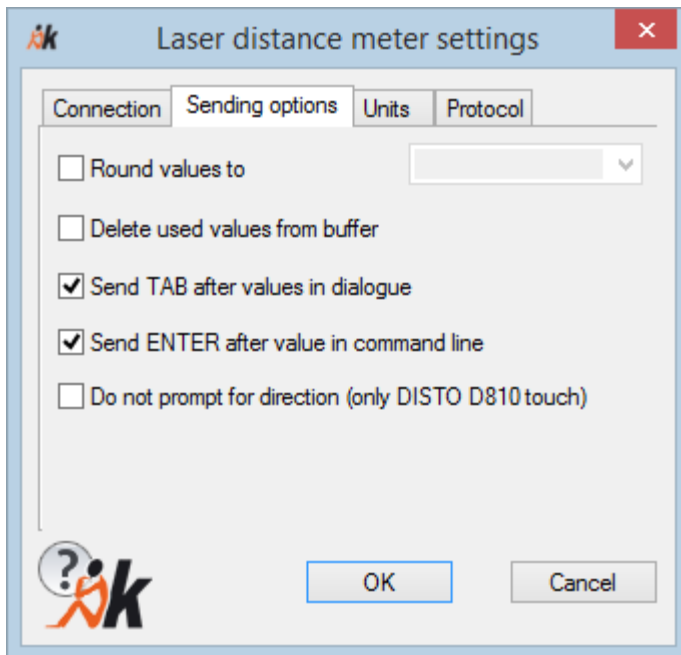
Within the **command reference** of As-Built you will find detailed descriptions of the different options. Now close the settings dialogue.

2.2 How does the measured value get into AutoCAD?

Measure the length of the first wall with the laser distance meter. Then press the "Send" button. When the Bluetooth connection has been established, the value will be displayed within the list of the laser distance meter box. At the same time the computer will send an acoustic signal.

Some laser distance meters (e.g. Leica DISTO) have several send buttons (blue arrows).

The arrow in front of the measurement value corresponds to the used send button of the laser distance meter.
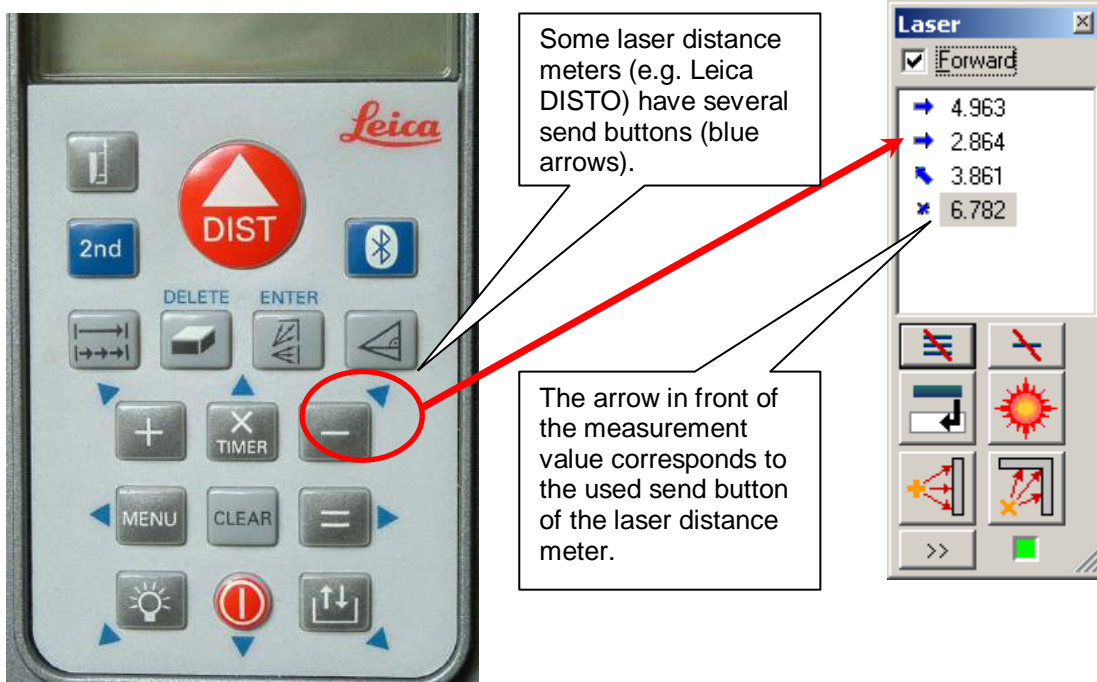
You can now measure the remaining distances in the same way and collect them within the list. You can use the different arrow directions as memory hook in order to relate the different measurement values again later on.

2.3 The mode of operation

As a matter of principle there are two different modes of operation when measuring and drawing with a laser distance meter.

2.3.1 Collect measurements and process them separately afterwards

In the next step you will construct a polyline with these values.

In order to do so select the symbol for the AutoCAD command "Line" from the toolbar or enter the command directly into the command line:

```
Command: line
State first point:                    (AutoCAD will prompt you for the starting point of the line.)
```

Click the starting point of the line with the mouse. Now drag the mouse in the desired direction – the line will follow on a "rubber-band". When the direction is correct select the context menu with the right mouse key and select the "Selected handlaser value" entry.
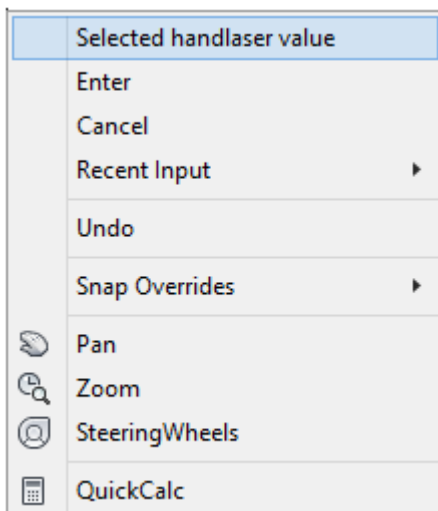
The value selected within the laser distance meter box will now be written into the command line and the started line will be drawn with that length. The used value will be deleted from the list and the selection switches to the next value.

Continue the polyline. Again pull the mouse in the wanted direction and get the measurement value from the list via the context menu.

In combination with the **AutoCAD option ORTHO** it is very easy to construct rectangular rooms this way.
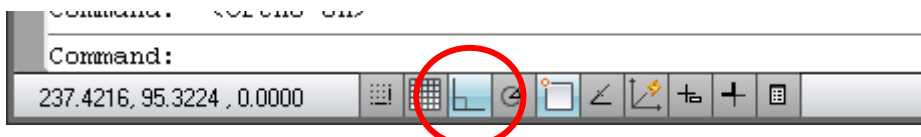
This mode of operation does not only work with AutoCAD lines but also with all other geometrical elements and design aids.

Whenever AutoCAD expects the entry of a distance, you can measure that with the laser distance meter and transfer the measurement value to the command line or into the dialogues of AutoCAD!

Examples would be polylines, arcs and circles with statement of the radius or the AutoCAD command "Offset".

If you want to, you may now just try some more measurements. For example change the layer or the element type.

2.3.2   Process measurements directly

The mode of operation described above is advisable for a one-man/one-woman measurement. If you work in a group of two another mode of operation will make more sense:

Within the laser distance meter box activate the setting "Forward". The measurement value will now directly be forwarded to the current cursor position when it is received in the list. When you are drawing a polyline this would of course be the command line. That way you can design and measure at the same time.

This mode of operation has especially been optimized for the As-Built commands "Measure perpendicular room" / "Measure variable room". There, the arrow direction information of As-Built will be evaluated and implemented graphically. The drawing direction of a polyline can therefore be controlled via the arrow keys on the laser distance meter, less entries at the laptop are necessary.
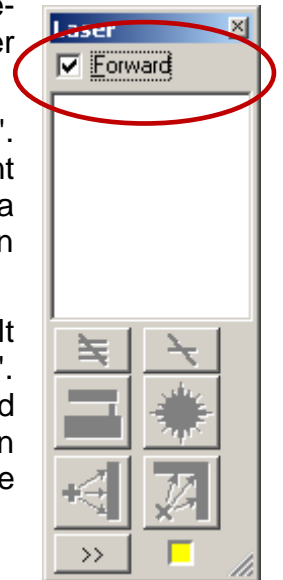
2.3.3   Remote initiation of a measurement

In some situations it may be advisable to initiate a laser distance meter measurement from the laptop. For example when the laser distance meter has been mounted to a connecting rod in order to be able measure from points which cannot be accessed easily.

For this, the As-Built laser distance meter box offers different options:

Initiation of a distance measurement. Since no direction information is available, a blue x appears in the list instead of an arrow.

Start/stop of a minimum measurement. The laser distance meter runs several measurements. The shortest distance will be put out.

This is a reasonable function to work with, when a distance is to be measured perpendicular to a wall. The searched for perpendicular distance is the shortest one.

Start/stop of a maximum measurement. The laser distance meter runs several measurements. The longest distance will be put out.

This is a reasonable function to work with, when a diagonal within the room is to be measured. The searched for distance from corner to corner is the longest one.
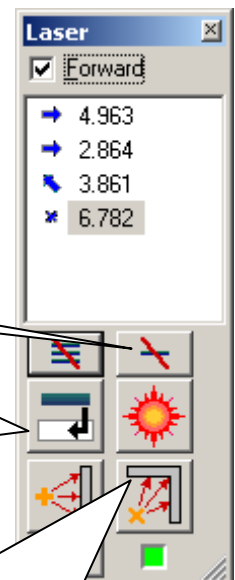
3 Survey projects with total station

After you have received a first overview of the measurement with As-Built for AutoCAD in the previous section, you will learn which steps a typical survey project with the total station interface involves.
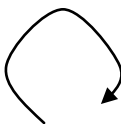
Typically you will obtain results in two dimensional plans (ground plan, vertical section, front view). However, the actual observation does indeed include three dimensional data.

If you create a 3D volume model on the basis of As-Built for AutoCAD volume bodies or on the basis of components of AutoCAD Architecture (ADT), the steps of your projects might vary. In such case for example you would not need to define a UCS (**U**ser **C**oordinate **S**ystem) for a specific sectional or view plane.

3.1 A typical project run

Every survey project possesses individual characteristics. Even though the survey for a project in restoration or archaeology differs from gathering data for facility management, most projects have something essential in common.

Mostly you find following workflow. Please notice the recommendations for the determination of the instrument station and control point in paragraph **3.4 Instrument station, control point, orientation**.

step	Description	Command
1st step	Select the first **instrument station** Orientation **"Local orientation"** (Up to now, there are no control points for "resection".)	New/Edit orientation...
	Determining and observing new **control points**. →Creating a file which will contain all control points (e.g. CONTROLPOINTS.DWG)	Measure new control point
	Observing the geometry which can be seen from the current instrument station	LINE, 3DPOLY, OUTLINEPLAN
2nd,3rd,4th...	Select a new **instrument station** **"Resection"** in CONTROLPOINTS.DWG, Already observed control points will be used.	New/Edit orientation...
	Determining and observing **new control points** in CONTROLPOINTS.DWG.	Measure new control point
	Observing the geometry which can be seen from the current instrument station. Selecting the appropriate drawing if another geometry than the one in the ground plan is observed (e.g. sections). If a new view is created, define a named UCS. Construction and plan drawing	LINE, 3DPOLY, OUTLINEPLAN

Naturally, depending on the task, there are numerous variations of this order.

For buildings with long extensions or complicated structures, we recommend a separate measurement of a traverse first for balanced and exact results (→Net adjustment). If you are only interested in room areas, you do not have to work with control points since a coordination of different instrument stations is not necessary.

3.2 Ground plan, vertical section, view – everything is possible with UCS

3.2.1 WCS and UCS – What is that?

Assuming that you have started your survey project as described in unit **3.1 A typical project run,** you have already defined your first instrument station with the "Local orientation" mode. You might also have measured two control points on the outer wall of the building and connected them through a line.

At first sight, the outer wall represented by the line appears to be arranged at no order. The orientation mode "Local orientation" determines that the instrument station exactly coincides with the origin of coordinates in the WCS (WCS = World Coordinate System). The direction of the Y axis in the WCS results from the (coincidental) direction 0 00'00'' at the instrument.

There is a world coordinate system (WCS) in every drawing and that is, by default, at first always the current system.
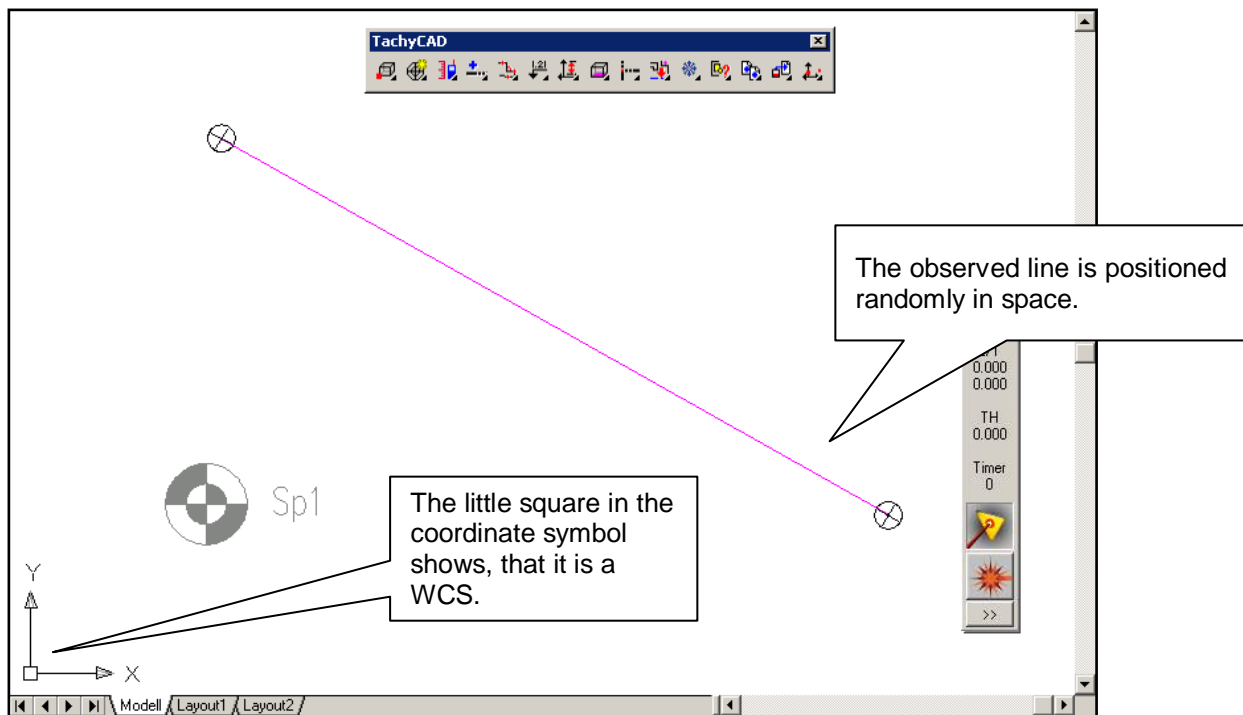
Normally, you aim for aligning e.g. the longest side of the building to be measured parallel to the X axis. There are numerous possibilities to achieve this. The most commonly used and most flexible method is to define a correspondingly oriented UCS (UCS = User Coordinate System). The UCS can be defined freely in space. Though you should define the UCS in such a way that the XY plane corresponds to the desired plane of projection.

Additionally to the UCS a drawing may have any number of user coordinate systems (UCS). But only one can be the current one – either the WCS or one of the UCS.
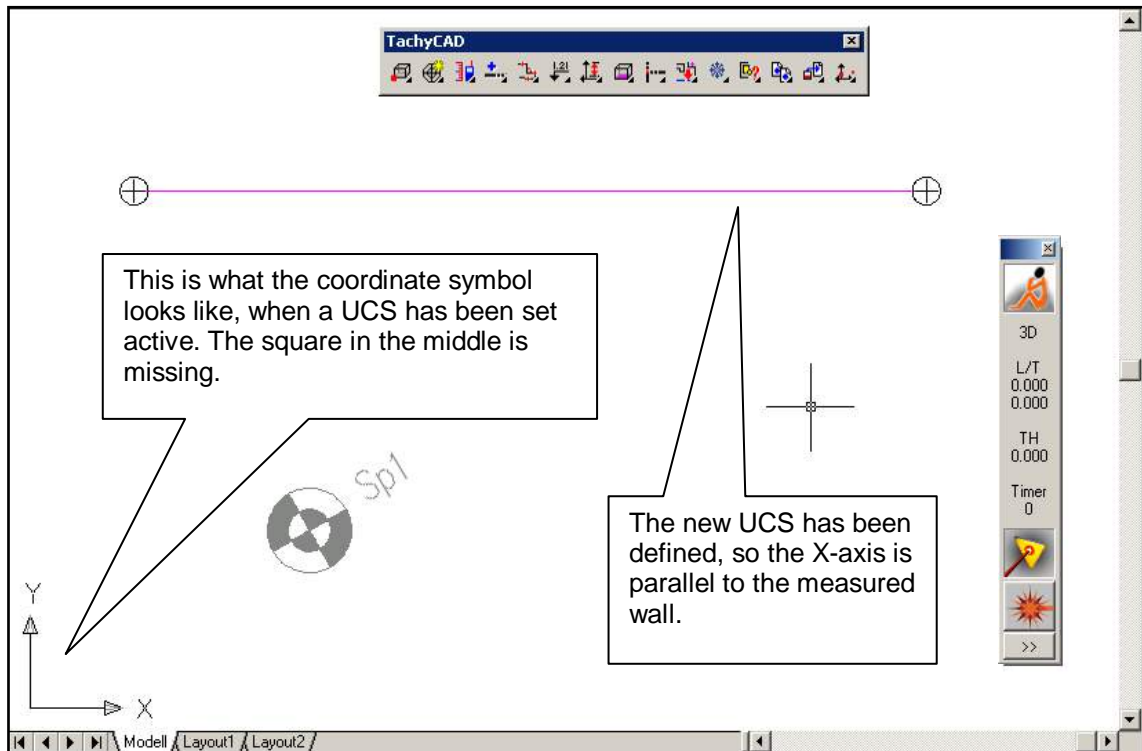
In the above picture a UCS has been defined for the ground plan. Indicator is the missing square in the coordinate symbol. The x-axis has been aligned at the previously measured line. Following several typical applications of WCS and UCS when working with As-Built for AutoCAD will be explained.

3.2.2 Aligning the WCS on the first instrument position

With this simple procedure you can organize that the WCS of the project will have the correct alignment instantly – for example the longest side of the building is to run horizontally within the drawing. By doing so you avoid having to apply user coordinate systems (UCS) afterwards.

When being on the first instrument position of a new project use the orientation method "No orientation"". Within the window the option "Define horizontal direction of the x-axis of the WCS"
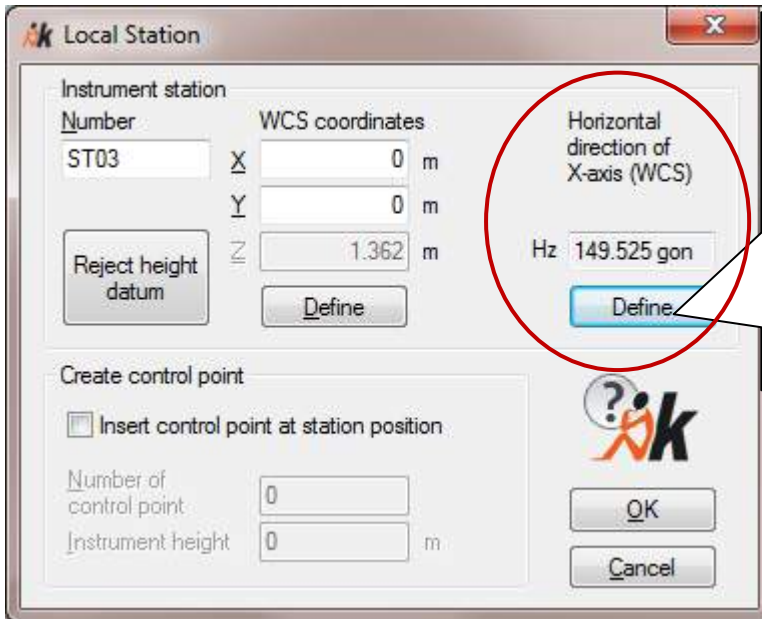
When clicking on Define, you will be prompted by TachyCAD to observe two points for the alignment of the x-axis of the WCS.

Observe the points on a wall, which is to run in x-direction within the drawing (e.g. the longest side of the building).

Observe the points first left right on the wall in order to determine the "viewing direction".

Now the x-axis of the WCS runs parallel to the observed wall.
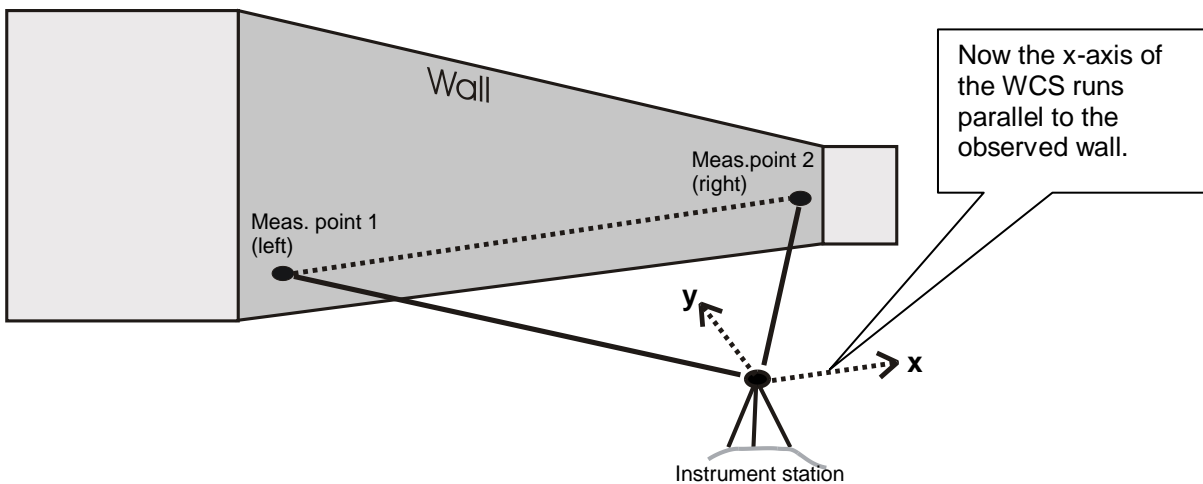
3.2.3   Defining vertical UCS for section or view drawings

If you want to measure and draw a vertical section or a facade additionally to the ground plan, you will have to define a UCS in each case, where the XY plane lies vertically in space and is accordingly aligned. For the following example a ground plan was measured. Along the marked intersection line a length section is to be measured.

First point for the section plane (left) is end point of the intersection line.

Second point for the section plane (right) is the end point of the intersection line.
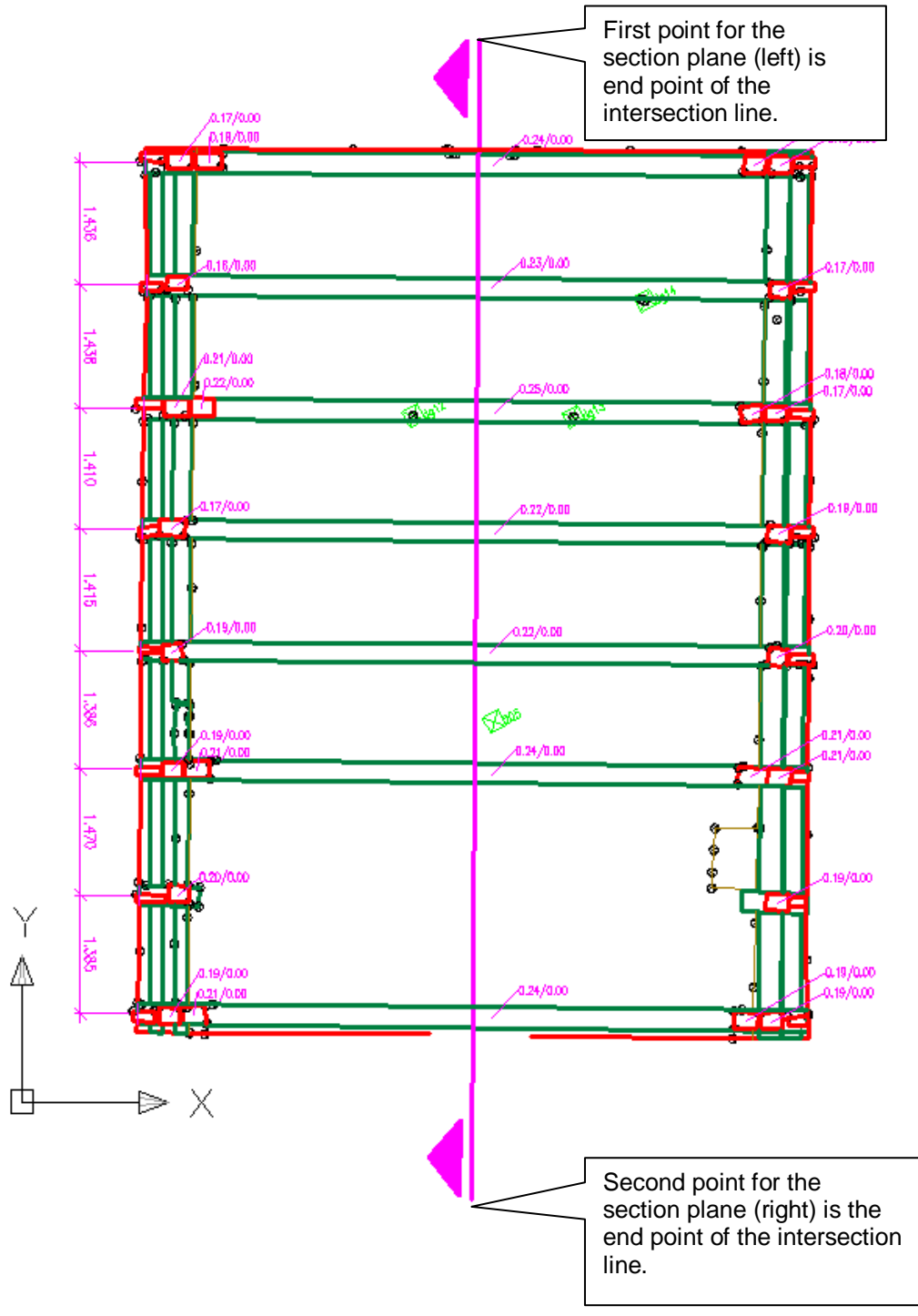
We suggest proceeding as follows:

- Copy the ground plan drawing and save it with another name (e.g. projectname_lengthsection.dwg).

- Start the As-Built command "Vertical view (UCS) trough 2 points". You will be prompted to state the first and the second point of the section plane. Within the drawing click on them or measure them directly with the Tachymeter, if possible. Make sure, you keep the correct order, at first the left point has to be stated, then the right point, in order to get the correct viewing direction.

- Enter a clear name for the UCS (e.g. "LengthSection A")

- The UCS will now be generated and the drawing will automatically be rotated, so you will be looking on the XY-plane of the new UCS. UCS-X now matches the length direction of the building and UCS-Y matches the height. Delete all elements from the drawing that are only ground-plan-relevant and are only interfering with the section view.
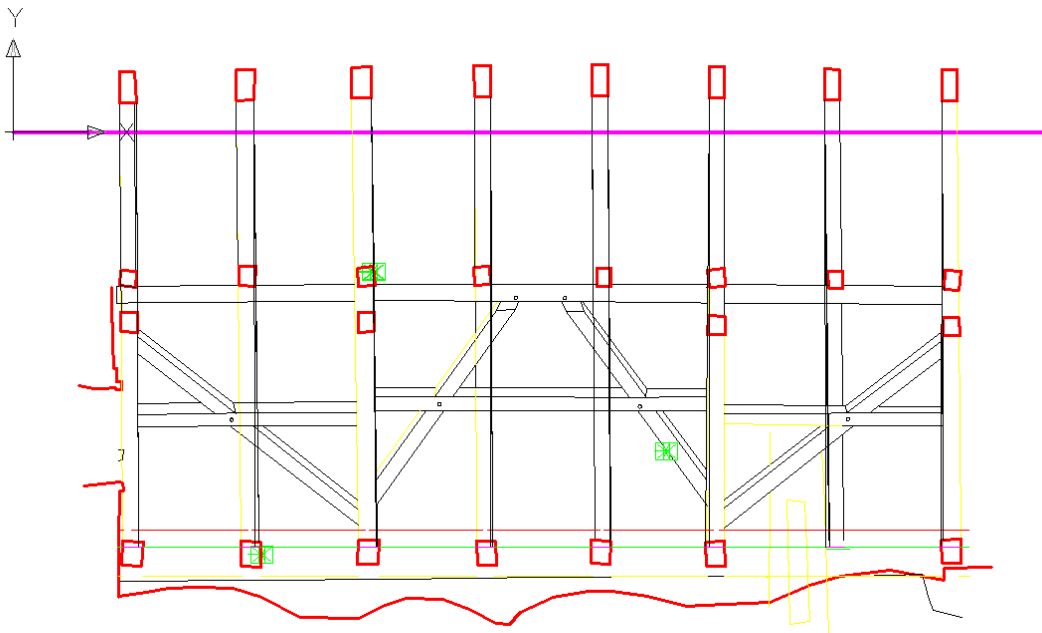
You may now start surveying/drawing. The procedure for the measurement is the same as for the ground plan. All you have to do is to rethink when deciding where to aim to at the object, since the "drawing paper" is now standing vertical.

It is an additional advantage if the XY-plane of the UCS is placed exactly in the section plane. With the help of the Z-value you may check at any time how close you stick to the given intersection line when observing.

3.2.4 Generating and managing UCS with AutoCAD

Within AutoCAD there are all possibilities you can think of for the definition of an optionally aligned UCS. You may, for example define a new UCS by aligning it at an existing line ("Object" option). This procedure is suggested for an UCS for the ground plan. You will find further information regarding this topic in the user documentation of AutoCAD.

Important: Assign a clear name to any defined UCS. That way it will be very easy for you to switch between the different UCS later on. In order to do so use the "Named UCS" command from the AutoCAD menu or simply call up "UCSMAN" within the command line.

You will find detailed information regarding this topic within the user documentation of AutoCAD.
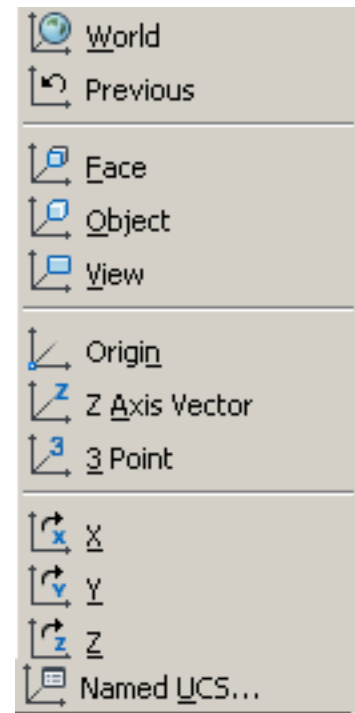
3.3 The measurement box – interface between AutoCAD and instrument

The measurement box symbolizes the connection between As-Built and your instrument. It constantly displays measurement settings and changes of measurement settings.

There are two buttons and a number of control displays:

3D or **2D** observation:
If 2D is chosen, the next observation will be carried out on an x-y-plane of the current user coordinate system (UCS).

Length- and cross eccentricity, which is used for the next observation .

Height of target prism, which will be used in the next observation .

Delay time for remote start

Display of current **observation mode:**

Full observation　　**plane intersection**　　**Horizontal eccentricity**

Remote start of an observation from laptop, if necessary with delay.

Measurement delay

Measure now　　8

Cancel

Extended observation box for configuration of settings

The **fire button** is used for the remote start of a measurement. You can also initiate the measurement directly at the instrument. If a valid instrument station exists (command "NEWORIENT") and the measurement box is visible, a measurement can be started at any time. In this case As-Built inserts the coordinates which were calculated from the measurement in the command line of AutoCAD. If AutoCAD is in a situation which allows the insertion of a point, the coordinates will be interpreted as such.

Example for the measurement of a line:

> Command: **Line**
>
> Specify first point: *(Now you start a measurement at the instrument)*
>
> Specify first point: **0.1234567,-9.8765432,123.4567890**
>
> Specify point or [Undo]:

The three icons show the current measurement mode. There are three different modes in As-Built: **Full measurement**, **Plane intersection** and **Horizontal eccentricity**. For all the result is a 3D coordinate. However, the coordinate is obtained differently:

- In the **Full measurement** mode the coordinate is calculated from directions and distance measurement.

- In the **Plane intersection** mode only the directions are measured and the point is obtained from the point of intersection of a beam with the previously defined plane.

- In the **Horizontal eccentricity** mode the coordinate is calculated from directions and distance measurement, too, but they are measured separately. In the first step the distance is measured, in a second step the correct direction is aimed and measured.

You find further information concerning "Measurement mode" in the As-Built command reference under "Measure - Execute instrument measurement" You can switch between the two modes; the symbol in the measurement box shows which one is currently active.

| **Full observation** | **plane intersection** | **Horizontal eccentricity** |

The button **>>** allows you to set the values for measurements. The dialogue, which opens when the button is clicked, is called "**extended measurement box".**

When you ask As-Built to measure (e.g. when observing a new control point with the command "**defcp**"), the extended measurement box will also open.

Shows the currently set instrument

All observation modes have individual setting possibilities. The frame of the currently active modus is on top.

Permanent: means that all following observations will adopt this value.

3D or **2D** observation ; if 2D is chosen, the next observation will be carried out in the XY plane of the current user coordinate system (UCS).

Insertion of a tolerance in Z direction for the exact observation of sections along a predefined intersection plane.

This button serves for **automatic targeting** of points. It is only displayed if a motorized instrument is attached to the notebook **and** if an approximation for the point to be observed is possible, e.g. when an orientation is over determined or when setting out points.

With every measurement an **Observation point with a number** will be inserted, which logs the measurement values (observations, coordinates, instrument station, additional information).

If the size of the measurement box disturbs you, you can close it. The command TCADMEASBAR allows you to configure the settings.

TachyCAD receives data from the instrument.

The measurement box is visible on the screen.

Commands for opening or closing the communication interface are described in the command reference under → Instrument observation – Connect instrument and → Instrument observation – Disconnect instrument.

3.4 Instrument station, control point, orientation

Almost every survey project requires a number of instrument stations. Normally, you want the observations of different instrument stations to complement one another; e.g. adjoining rooms should be depicted wall to wall in a plan, whereby the wall thickness is not determined by explicit observations but through the reference of different station points. This is also true for the thickness of ceilings.

As-Built offers different functions to achieve this aim. This unit introduces you to these functions.

3.4.1 Terminology

Instrument station

is the point on which the instrument is put up. Mostly it is not marked in the field but appears as a symbol in the plan.

Orientation

is the process during which the current **instrument station** is determined. Through this process, observations from different instrument stations can be included in a joint coordinate system. This means that one and the same point is measured from different station points and appears at exactly the same position in the plan. Orientation requires **control points**.

Control point

Is a point that both in the real world and in the drawing is identifiable by a distinct marking and a number. On an object the control points are mainly marked by adhesive markers[1] and on the ground by survey nails or similar. With a **resection** two, but better three, known control points are observed.

3.4.2 Orientation methods

As-Built for AutoCAD offers four methods of orientation:

- Resection (see figure)
- Orientation using existing control point
- Local orientation
- Orientation without horizon

Usually, you need **"Local orientation"** only for the first instrument station at the beginning of a project. The coordinates of the instrument station are set at any value. In the simplest case, you assign the coordinates of the origin (0,0) to x and y. Often you have to measure a height reference for the definition of the height dimension.

"Orientation using an existing control point", **"Free orientation"** and **"Orientation without horizon"** are usually used except for the first instrument

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[1] A sample for adhesive control point markers can be found in the installation folder of FARO Box for AutoCAD under "…Tools\TachyCAD\Survey point.doc". These can be printed out on a sheet of standard labels.

station. In order to include the new observations into a joint coordinate system with those of the previous instrument stations, you have to measure a certain number of control points.

	Instrument station
	Control point
	Observation during orientation
	Simple observation
	Objects to be observed
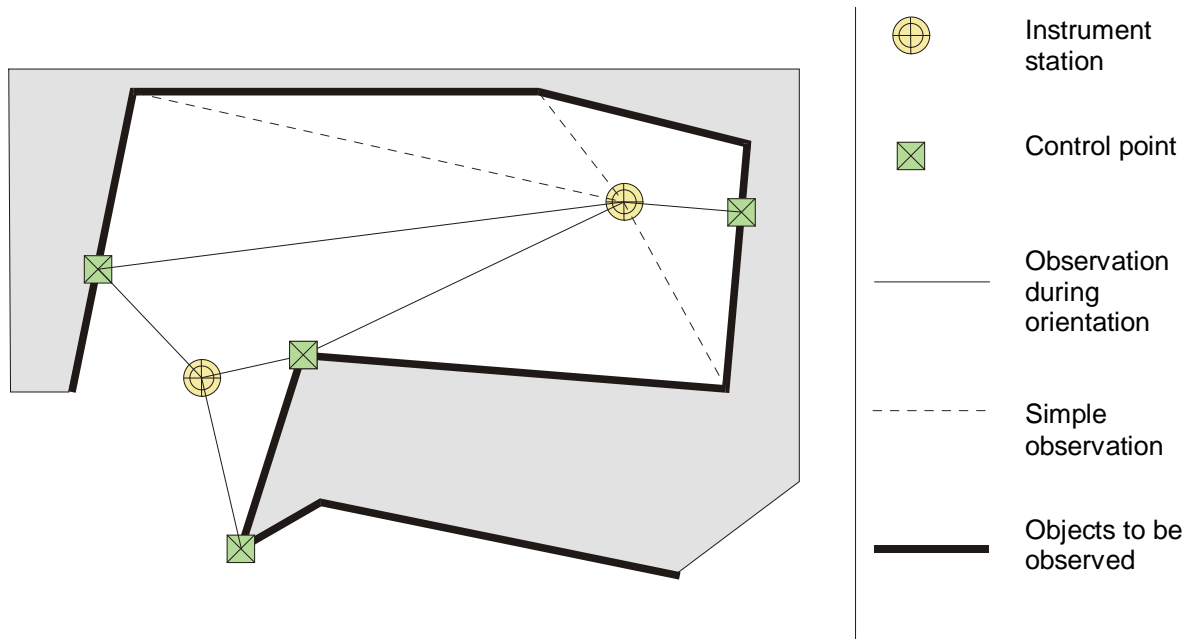
While you can choose any instrument station when you work with "Free orientation" or "Orientation without horizon", you have to place your instrument directly above a control point when you work with "Orientation using an existing control point". "Orientation without horizon" **serves the orientation inside oscillating objects like for instance boats.**

Depending on the number of control points you use for determining a new instrument station, the orientation can be in different states.

The orientation is incomplete

if you have not measured enough control points in order to calculate the instrument station.

The orientation is determined

if a sufficient number of control points has been measured in order to calculate the instrument station. An error check and an evaluation of exactness is not possible.

The orientation is over determined

if more control points have been measured than necessary for the determination of the instrument position an error check and evaluation of accuracy is possible.

You can find further notes concerning the necessary number of control points and different possibilities of orientation in the command reference under ➔**Orientation - New/Edit orientation.**

3.4.3   Accuracy of orientation

The accuracy of orientation substantially influences how well the measurements form different instrument stations match each other. Unfortunately, there is no general rule to guarantee the accuracy for all situations of everyday praxis. Nevertheless, the

following recommendations can help you to achieve a sufficient accuracy for your demands.

- Be very conscientious during orientation. Use the reticule of the telescope for aiming at control points. Avoid only aiming with the red laser point when using a tachymeter that measures reflectorless.

- The first instrument station should be chosen in such a way that as many control points as possible can be seen from that point. Often a central point in or outside the building is suitable (The latter allows to measure control points in the windows of different stories and on neighboring buildings).

- The control points should be equally spread around the instrument station during orientation (ideal is an angle of 120° between the points).

- Use as many control points that have been measured from your first instrument station as possible for the individual orientations during a project.

- Use control points that are far away as often as possible.

- Never forget that a large number of points should be seen from one instrument station. Every additional orientation wastes time.

- Avoid often connecting instrument stations that follow one after another. Such connections mean using control points that have been determined at the previous instrument station only. We recommend observing a separate polygon for complicated building structures first. (Command reference → Net adjustment)

- Do not continue observing when your orientation produces large error values. Try first to find the error sources.

3.5 Building Surveying

In this unit we will explain a few typical measuring situations and how to cope with them in As-Built for AutoCAD. However, since the tasks in building surveying are so numerous, this unit cannot be a complete enumeration of commands. You can find further information concerning these commands in the →command reference.

Measure an outline plan

Use the command →**Measure - Measure outline plan**. Measure two points on each wall around the room. The created line sections are automatically extended so that the wall corners emerge. Different options also allow you to measure corners directly, construct rectangular walls and include manually measured values.

Measure alcoves/recesses

For rectangular recesses use the command → **Measure-Measure recess.** Within a dialogue box you can you can choose between different measurement modes, depending on whether you want to measure the recess with the total station or by hand/Laser distance meter.

Alternatively the command →**Measure - Perpendicular return** is suitable for rectangular alcoves, too. First, you measure the wall without the alcove (e.g. with 'Measure outline plan').With **Perpendicular return** you can drop a perpendicular from the measured point onto a reference line. Select the wall line as reference line

and measure one point on every wall of the alcove. The reference line will be broken up at the **perpendicular point**. You can also choose to insert the (plumb) line.

For irregular alcoves you have to use the command ➔**Measure - Measure outline plan** again. You can also use the AutoCAD command '**Line**'

Measure pillars and columns

Use the command ➔**Measure - Circle through 3 3D points** for circular columns. Measure three points on the surface of the column (also possible in different heights as long as the radius does not change). As-Built calculates the circular arc in the ground plan

Use the command ➔**Measure - Measure rectangle** for rectangular pillars. Depending on how well you can aim at the pillar, choose between different measurement modes e.g. aim directly at corner or determine indirectly, measure edge by hand or with total station, rectangular or square shape.

Curves

If the curves can be described by a circular arc, the command ➔**Measure - Arc through 3 3D points** is suitable. Measure three points on the curve; As-Built will then calculate an arc, which will be projected into the ground plan.

If the curves are irregular, measure a 2D polyline with the AutoCAD command '**PLine**'. Since these polylines are only two-dimensional (i.e. all points have to be of the same height), but you always measure three-dimensional coordinates with the instrument, you have to switch from 3D to 2D in the large As-Built measurement box. Use the command '**PEdit**' (edit Polyline) after the measurement to adjust curves (option 'Fit').

Particularly comfortable is the command ➔**Measure – Polyline with curves**. Using it curved and linear parts of organically shaped polylines can be combined without limitations. (➔command reference)
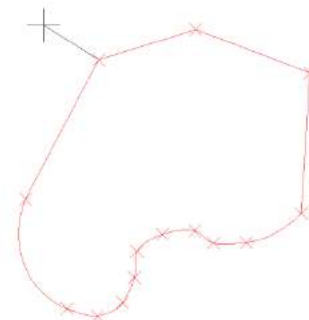
Measure doors and windows

The command ➔**Measure – Measure and insert door** allows to insert a door symbol into a wall line. You can choose between different types of doors and optionally insert the annotation of the clearance . Depending on whether you want to measure the door with the total station or by hand, you can choose between different measurement modes.

The measurement of windows works similar with the command ➔**Measure – Measure and insert window**. Different types of standard window reveals (ordinary window, window with rabbet) are supported. The command ➔**Annotation - Window dimension annotation** allows to enter clear height, clear width and height dimensions.

Both commands have an automatic mode, which allows to insert several doors or windows with the same settings quickly and comfortably

For windows with an irregular geometry or complicated reveals, you have to use the Command ➔**Measure – Measure outline plan** or the AutoCAD command '**Line**'. For breaking up the wall line, the Command ➔**Measure – Perpendicular return** is suitable.

Measure Staircases

For drawing staircases use the Command ➔**Measure – Measure staircase.** Simple straight flights of stairs with parallel and equal steps can be determined by entering number, pitch and rise of steps. As-Built calculates the staircase automatically.

More complex flights of stairs (rounded steps, spiral staircases) you draw in two steps: At first measure every single step using the very efficient Command **Measure – Measure segments.** You can decide how many measurement points you want to take for every step. Once set, the command always starts a new polylinie after that number of measurement points without any inputs on the computer. In the next step the Command ➔**Measure – Measure staircase** helps you to finish the staircase drawing.

Beams

The fastest way of measuring straight beams is to use the AutoCAD command '**Line**'. You can generate the second parallel line with the AutoCAD command '**Offset'** and the option 'Through'. Measure the point through which the first line is to be moved parallel with the total station. Finally, extend the lines with the AutoCAD command '**Extend**' or with the Command ➔**Design – Multiple Trim and Extend.**

If the beams or stair Steps are not straight but have do bend and there are many of them to be measured the Command ➔**Measure – Measure segments** will be very helpful**.**

Set Object heights and dimension room heights

As-Built offers a whole set of commands for dimensioning heights (➔Height dimension tools). Measure the point of which you want to know the height. As-Built calculates the height and inserts a height icon with the height value at the respective place in the drawing. The way the icon is displayed, pre- and suffixes of the value as well as the height reference can be comprehensively configured.

Display vertical arcs in the ground plan

First measure the vertical arc with a 3D polyline. Then the command '**Annotation - Create rotated arch plan**' lays a balanced arc through the polyline and rotates it into the ground plan. If you want to, you can display the height dimensions of the arc at the start points and in the middle of the arc.

Special shapes of buildings

If you have to measure very complex geometry like exceptional roof shapes or something similar the commands for creating, editing and intersecting planes will be very useful. (➔**Design).**

If you measured geometry that lies on a cylindric surface (e.g. a round tower or a vault) and you want to transfer it into a plan you may use the command ➔**Design – Cylinder unrolling**.

Fit geometry into a global coordinate system

The commands ➔**Transform drawing elements** serve this purpose**.**

Commands for manual surveying

You will never measure a whole building only using a total station. There will always be measurements which have to be taken by hand. As-Built offers several useful commands; but also many basic AutoCAD commands prove helpful.

For the manual surveying of rooms there are the two commands ➔**Manual measurement-Perpendicular room** and ➔**Manual measurement-variable room.** These commands directly translate the values from bluetooth laser distance meters into Wall lines. But also tape measured distances can be entered. The ➔**Manual measurement-Perpendicular room** provides an adjustment of measurement imprecision.

The As-Built Measurement tool commands ➔ **Measure door,** ➔ **Measure Window,** ➔ **Measure rectangle,** ➔ **Measure Staircase** and ➔ **Measure Recess** have also

options for manual measurement. Instead of total station points the geometry can be determined by distances for width, depth, length and so on.

Use the command ➔**Measure – Fix points on a line** to set progressively measured values onto a line. Examples are manually measured alcoves, doors and windows.

The command ➔**Measure – Arc Intersection** allows you to determine a new point by (manually) measuring the distance from two known points. The circular arc intersection method is used here.

A second command ➔**Manual measurement-arc intersection (transparent)** delivers a coordinate that can be used inside other drawing commands. For example you can use it in combination with the AutoCAD "line" command to draw a line only with distances from tape or laser distance meter.

There are different commands for extending lines (➔**Extend**). You can either extend a line to a total value or by a certain value. As-Built also supports the rectangular extension of lines.

In connection with the **AutoCAD Ortho mode** the command ➔**Design - Align cursor** offers a very effective method of measuring rectangular rooms. AutoCAD aligns the cursor at a line (e.g. building side); while the Ortho mode only allows to draw lines that run either perpendicular or parallel to the cursor.

Area data / feature data management

As-Built provides several commands for the structured recording of area data and non-geometrical data as well as for fast and comfortable creation of room polygons. A configurable area list is kept along with the drawing. The list is structured like a folder tree. The graphical objects are linked with the non-geometrical information of the area list (structure view). These data can be exported to a table in readable ACSII format, to a HTML file or to an XML file for further processing inside databases or table calculation programs.

3.5.1   Building survey with a laser distance meter – introductive tutorial

Here you will learn which typical steps a survey project by hand with As-Built comprises. The listing makes no claim to be complete, since the measurement situations, which occur during a building survey, are too manifold.

For As-Built a manual measurement for rooms is to be seen as complementary instrument for the tachymetric measurement. In this case there would be already measured parts of the building as initial point. Geometrically easy rooms will be added in a manual measurement.

Note: The described function of directly forwarding measurements from a hand-held laser to AutoCAD is not used in the tutorial for practical reasons. Since this example works with fixed values, it is not useful to connect your instrument. **Whenever you have to enter a measurement value in the example, you would, of course, measure and transfer this value in reality from your laser distance meter**.

Another hint before you get started:

There will often commands of the status line for **AutoCAD layout settings** be used for the tutorial. These commands are valuable aids when constructing and they offer many opportunities. You can find these commands within the status line of AutoCAD (The gray line below the command line.) as button or icon with the descriptions ORTHO, POLAR, OSNAP and OTRACK.
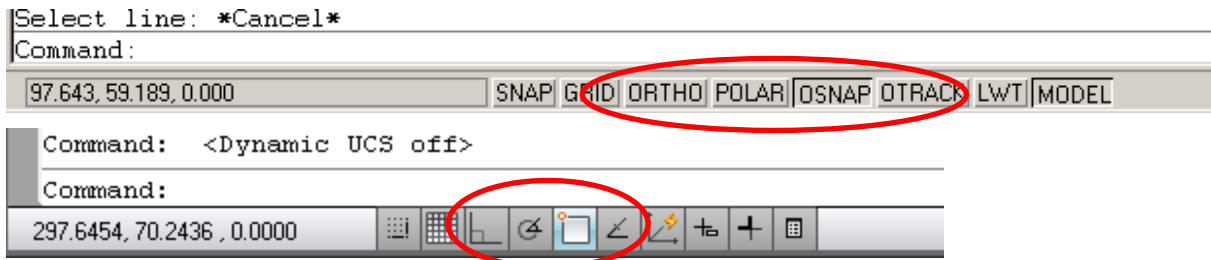
These can be activated and deactivated when needed. The single object snaps may be configured by calling up the context menu (right mouse button) and selection of "Settings…".

- **OSNAP** serves to "catch" certain points in the drawing e.g. the end point of a line, the intersection point of two objects or the plumb on a line. That way you can hit the exact position of a point of an object without having to know the coordinates or having to draw construction lines
- **POLAR** serves for aligning new objects according to certain angles.
- With **ORTHO** you can draw exactly perpendicular to the cross hairs. This option is extremely useful since floor plans are often perpendicular.
- With **OTRACK** you draw objects in a certain angle or in a certain relation to other objects. Temporary orientation paths will be displayed with which you can draw your object in a certain direction.

Within the tutorial these commands are generally written in capital letters, in order to point out that they do not have to be entered into the command line but have to set via the status line.

Please note that blue arrows symbolize AutoCAD commands

and green arrows symbolize As-Built commands.

Example data

Two rooms are measured for this example. At this it will be demonstrated how you could proceed under As-Built with a manual measurement for perpendicular and oblique rooms.
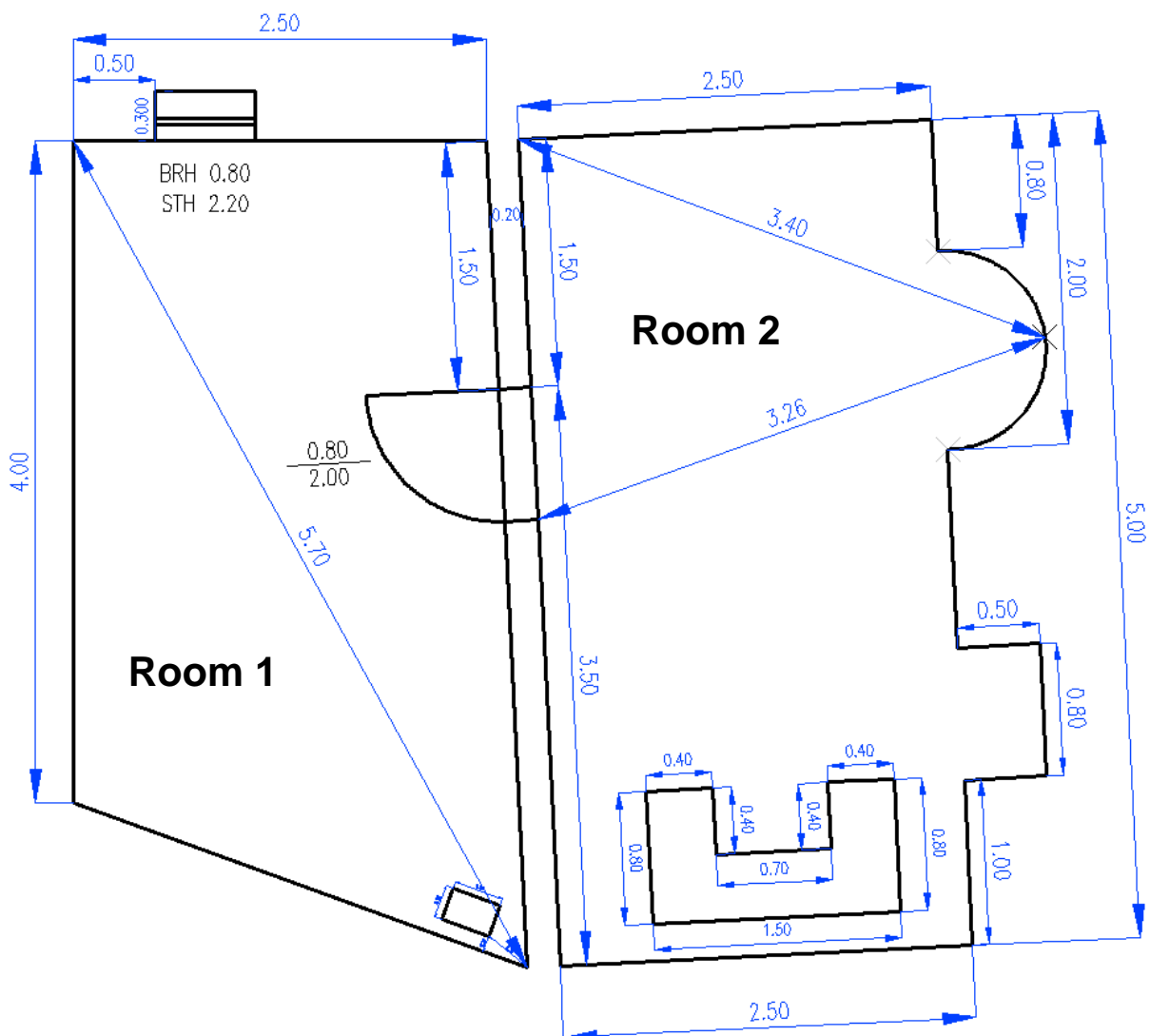
Preparation

Generate a new drawing with the AutoCAD command "new". Here select the "tachycad.dwt" or the "distoplan.dwt" template.

The first room

We will now start with room 1. It has an oblique geometry with an exception of the top left corner, which is perpendicular.

For the survey of this room start the command "Measure variable room".
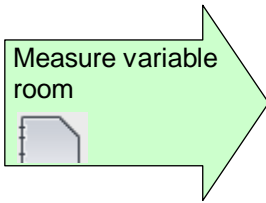
Measure variable room

> Command: _**TCADROOMOUTLINE**
>
> Current mode: UCS
>
> Select initial point or [oPtions]: P

Enter "P" for options. Here you will set, to which object room 1 is to be aligned. We will select that the room is to be aligned parallel to the UCS.
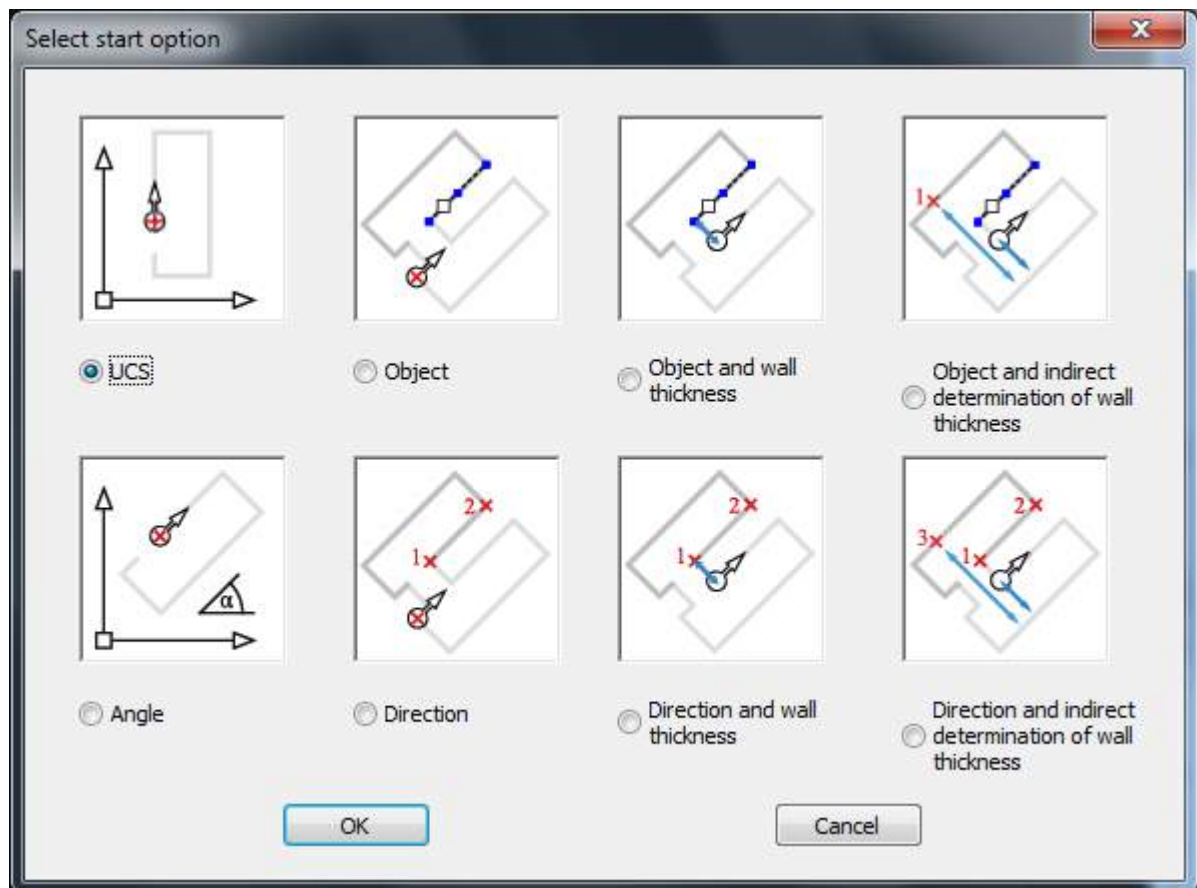
Exit the dialog with "OK". You will now be prompted to enter the initial point:

> Select initial point or [oPtions]: 0,0,0

Click any point in the empty drawing. That point will become the bottom left corner of the room. The following dialog will open:
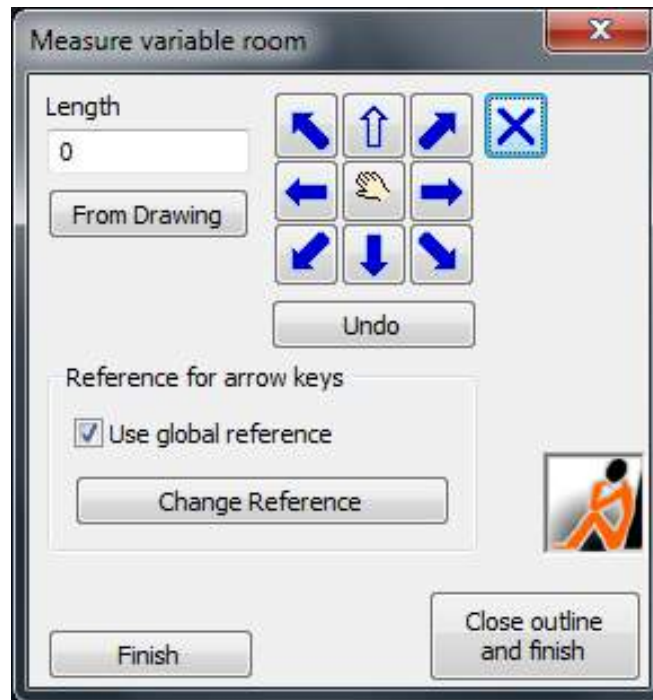
Starting from this point the walls of the room will be measured. We will start with the left wall.

1. For the length enter: 4.0 and then press ⬆️ [2]. Now a 4m line will be drawn vertical up
2. The perpendicular corner of the room will be next. For the length enter 2.5 and then press ➡️. The wall will be drawn to the right.
3. An oblique corner of the room will be next (top right corner). Since we cannot work with the arrow functions (You can only map 90° and 45° angles with those.), we will select the arc intersection function. Additionally to the wall line a space diagonal to the corner point will be measured.

   For the length enter 5.0 and then press ❌ [3] . That is the length of the sloping wall.

   A small dialog will open next, there you will enter the length of the space diagonal leading to the corner to be calculated.

   Enter 5.7 in the length box and then press "OK".

---
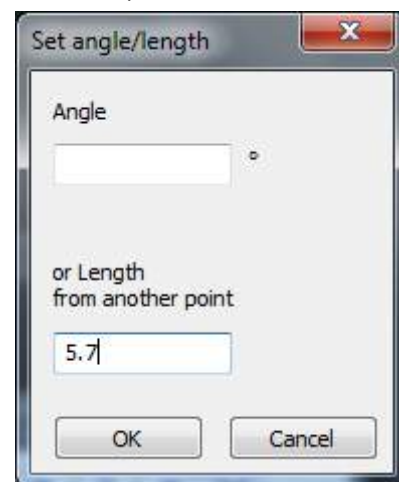
[2] In reality you would, of course, use a laser distance meter for the measurement and then transfer the value with the appropriate arrow key. Then you do not have to press any other key, the line will be drawn directly.

This is the same for all further entries.

[3] In reality you would transfer the measurement reading via the Bluetooth to the computer not with the arrow key.

Now you have to state the point to which the space diagonal has been measured. There will be an according prompt within the command line:

Select reference point or [2Reference points]: *(Select to top left corner of the room.)*

Select side *(Pull the arrow down into the room and then click.)*

4. Now all necessary measurements for the room have been executed. In order to close the room outline click on "**Close outline and finish**" within the dialog.

This is what your drawing should look" like now:
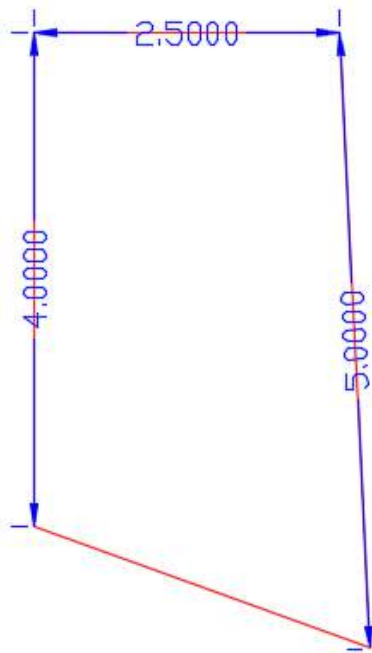
Additionally to the wall, a dimensioning has been generated which is according to the entered or measured values. The protocol of the laser distance meter values will be stored on the "AccuracyReport" layer. To keep a better overview this layer will now be switched off.

In order to do so use the AutoCAD command "layer". Within the layer manager search for the layer in the list and switch it off by clicking on the yellow lamp (lamp will turn blue). The protocol dimensioning will still be generated, but it is not visible.

Measure column

The following dimensions are to be used for the localization of the rectangular column in room 1:

Dimension from the corner of the room.

Perpendicular to the lower wall in a distance of 0.1m.
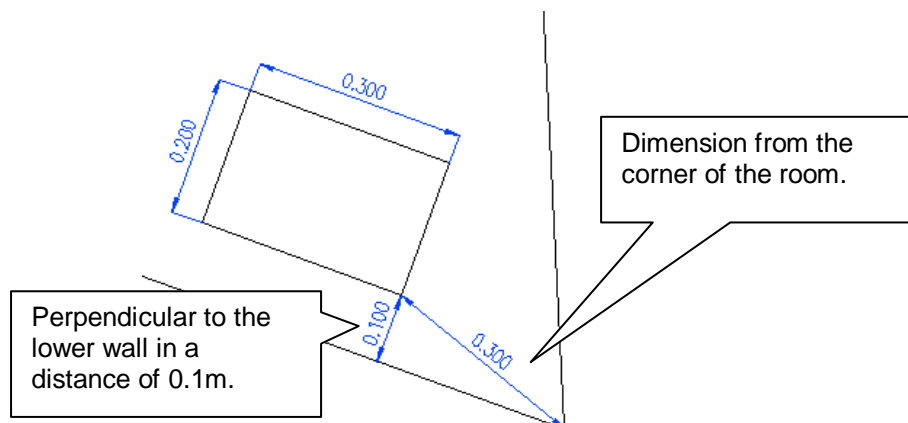
Now call up the command "**Measure rectangle**" in order to generate the column.

Measure rectangle

Command**: tcadrect**
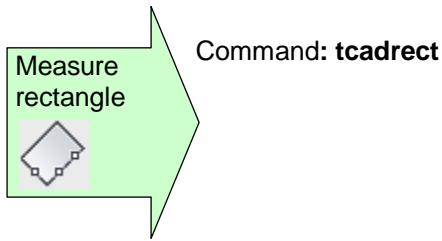
A dialog will appear, where you will have to select the measurement type first:

- "Positioning with distances"
- Picture "One reference line"

After confirmation with "OK" you will be prompted in the command line as follows:

Select reference line or [UNdo]:   (*Select the lowermost inclined wall.)*

Select reference point or [UNdo]:   *(Select the bottom right corner point of the room.)*

distance to reference line or [UNdo]: **0.1**

shortest distance to reference point or [Undo]: **0.3**

Then a dialog for the input of the side lengths of the rectangle will open:

Measure rectangle - define lengths

| Red length | 0.2 | Distance | 0.1 |
| Blue length | 0.3 | Distance | 0.3 |

Aspect ratio
- Rectangle
- Square red length
- Square blue length

Keep settings for next measurements (automatic mode)

OK   Cancel
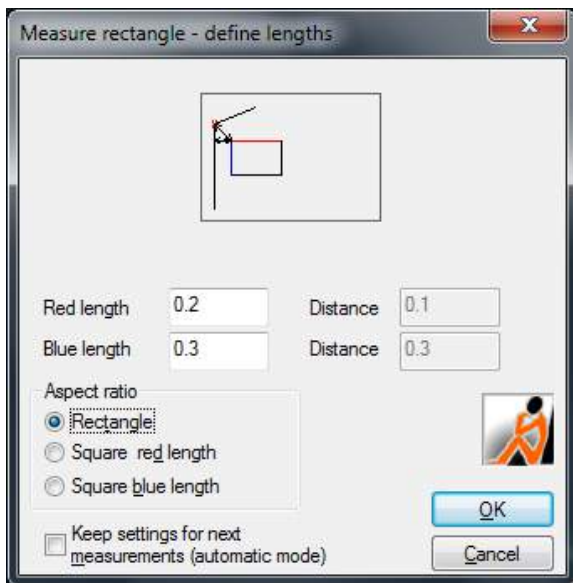
Enter the side length of the column as described above and close the dialog with "OK". Following you will be prompted to click a direction.

Select position: *(Click into room 1)*

Measure window

Now room 1 will get a window at the upper end.

The "Measure and insert window" command will be called up.

Command: **insertwindow**
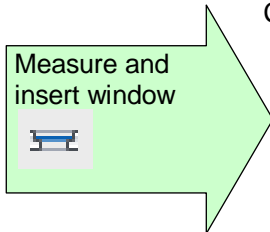
Measure and insert window
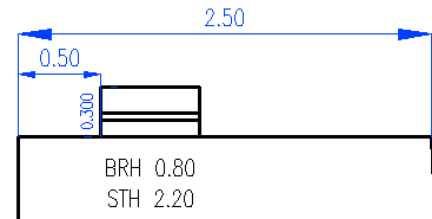
Via the "Change window type" key you will get to the sub dialog "Insert window - type selection". Now select "Window with single reveal" and check "Window with frame" and "Break up of wall lines". Now click "OK". You will get back to the previous dialog that has now the title "insert window with simple reveal". Execute the following settings:
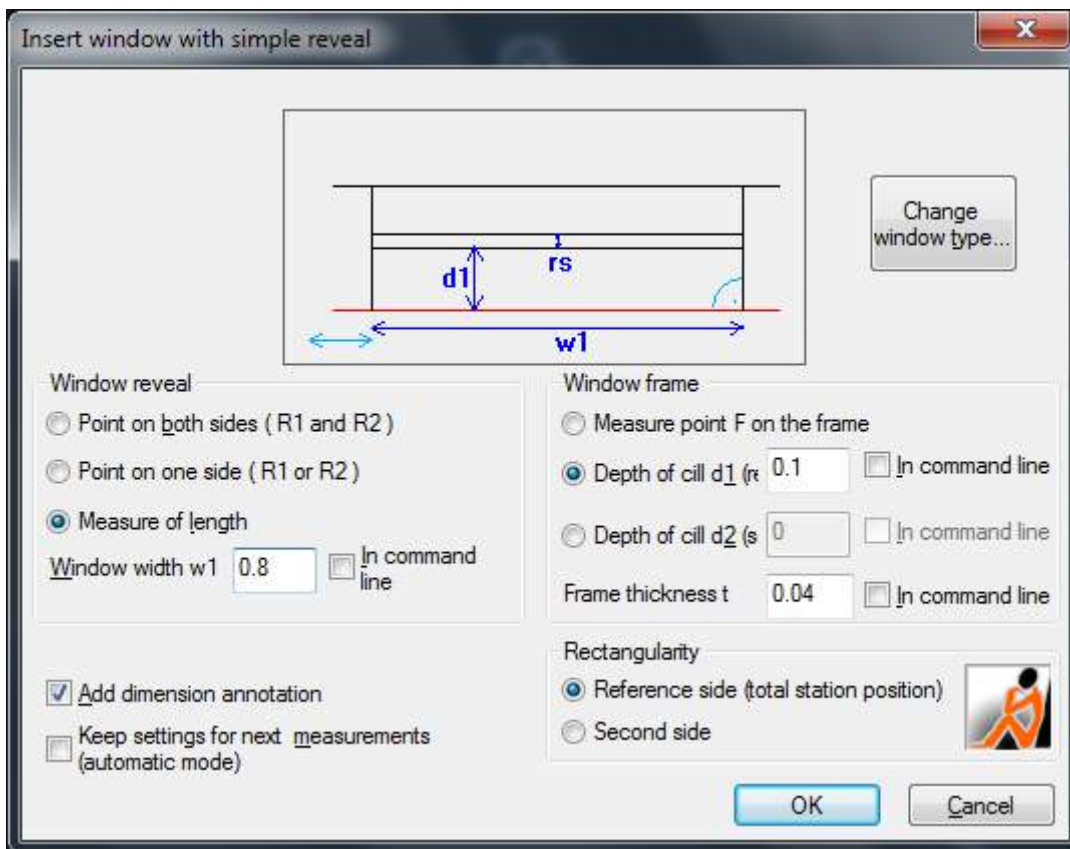
Click on "OK" and the following prompts will appear:

Select reference line: *(Click upper wall of room 1, when doing so touch line at its left end.)*

When selecting the reference line make sure to click the reference line at the end from where the dimension for the beginning of the reveal is to be measured (in this case from the left).

Now enter the wall thickness:

Enter wall thickness or [Line/Point/Back]<Line>: **0.3**

Select side for other wall or [Back]:

(Click point above the wall in order to show where the outer wall is.)

Enter progressive distance for the beginning of the reveal [Back]: **0.5**

(Distance left corner to window)

Afterwards the dialog for the dimensioning of the window will open. Please enter the desired values. Select

- Cill height (Cht): Value with 0.8

- Lintel height (Lht): Value with 1.4

- Activate option: "Value of Lht refer to Cht" (That means the value between cill and lintel has been measured).

- Display heights: Absolute

- Text alignment: Automatic

The clear values will not be displayed, that means do not check "Window opening dimension". After closing the dialog with "OK" the dimensioning numbers will automatically be positioned.

Measure door

Next, the door of room 1 will be measured. In order to do so call up the "Measure and insert door" command:

Command: **insertdoor**

Measure and insert door
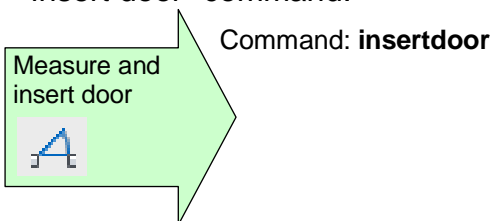
The "Insert door" dialog will open. At first press the "Change door type…" button.

This opens the "Insert door - type selection" dialog. Set the following options:
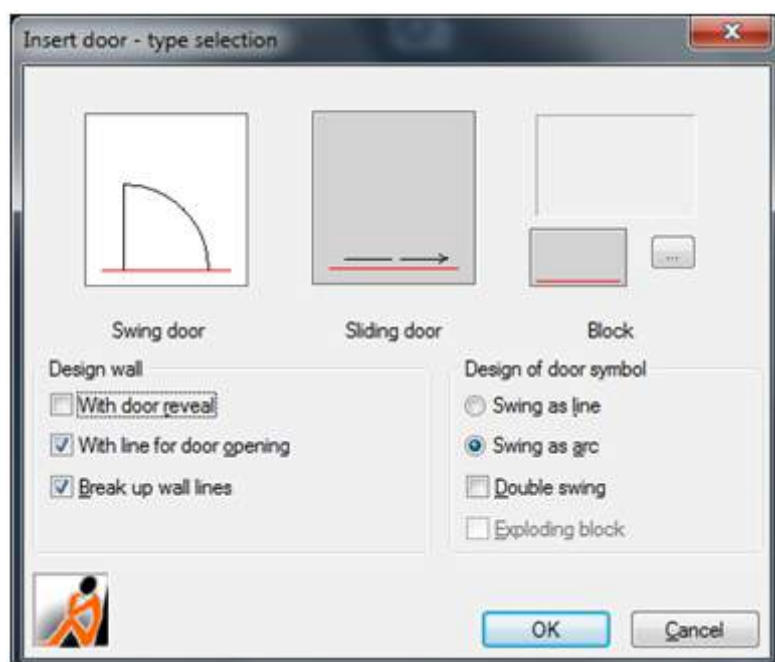
Close the dialog with "OK".

Now you are back at the previous dialog. Please execute the following settings:
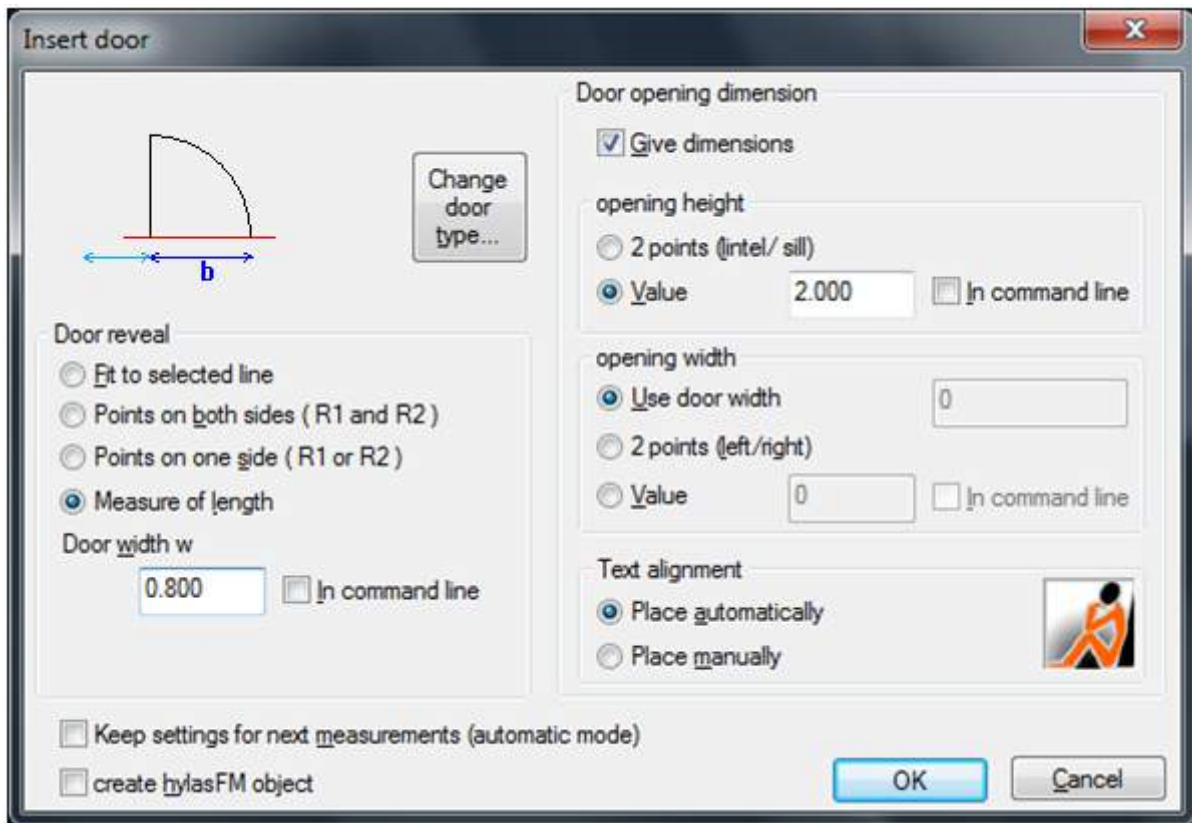
Close the dialog with "OK". Via the command line you will now be prompted:

> Select wall line or [two POints] <two POints>: *(Select the wall of room 1 at the upper end)*
>
> Enter distance from the corner to the reveal or [UNdo]:**1.5**
>
> Select point for direction of door swing or [UNdo]: *(Click into the upper area of room 1)*

When prompted for the alignment of the door the position of the door will be scribed according to where you pull the mouse. If the alignment for the door is correct, simply click. The door symbol will be inserted and the clear measures will automatically be positioned.

The second room

The basic geometry of room 2 is perpendicular. Therefore we will use the command "Measure perpendicular room".

measureperproom

Command: **_TCADROOMOUTLINEPERP**

Current mode: UCS

Select initial point or [oPtions]: P

Enter "P" for options. Here you set from which point the room should start and to what it will be aligned. We select the "Object and wall thickness" option, since the room is to be aligned at the existing wall of room 1 and we want to state a wall thickness.
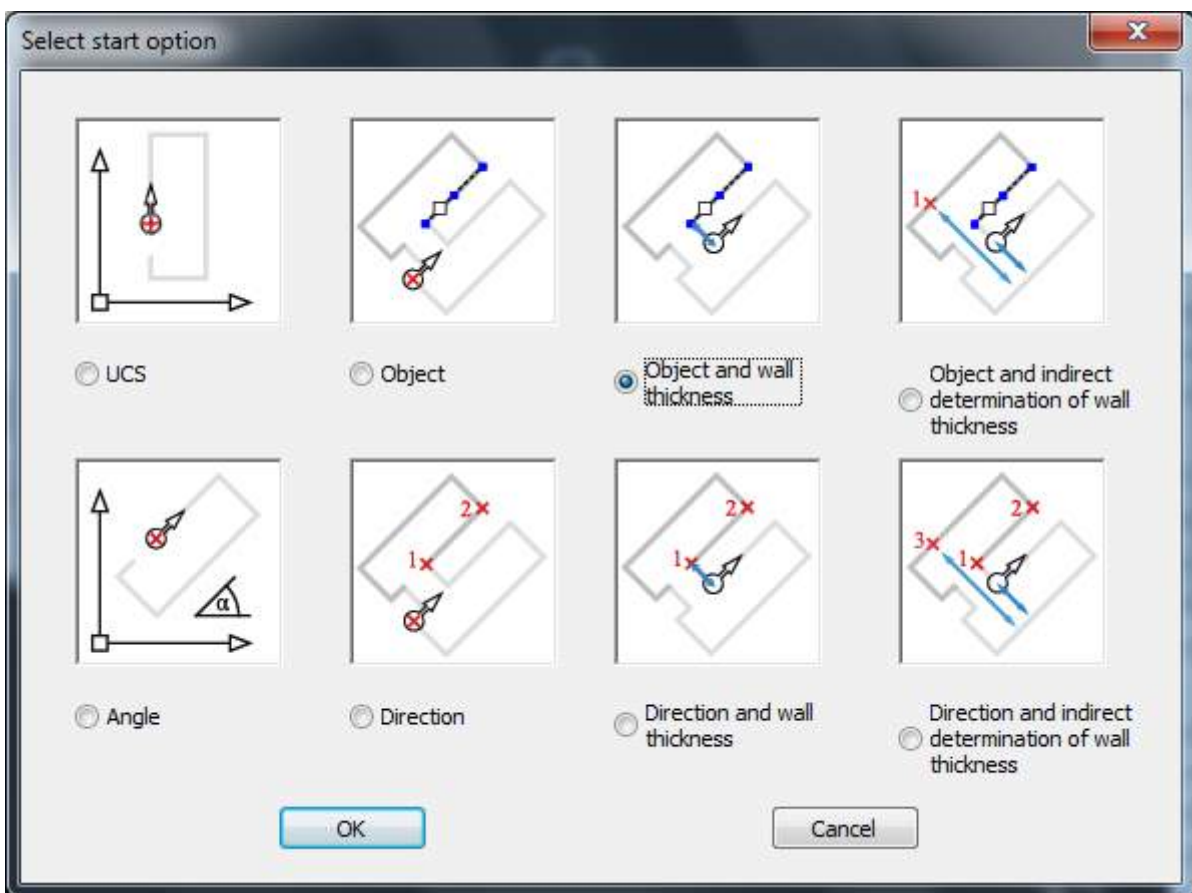
After you have pressed "OK" you will be prompted as follows:

Select line or [oPtions]:   *(Click right wall of room 1 shortly above the door symbol.)*

Enter wall thickness <0>: **0.2**

Select side: *(Pull the scribed direction arrow towards room 2 and click there.)*

After that the following dialog will open:
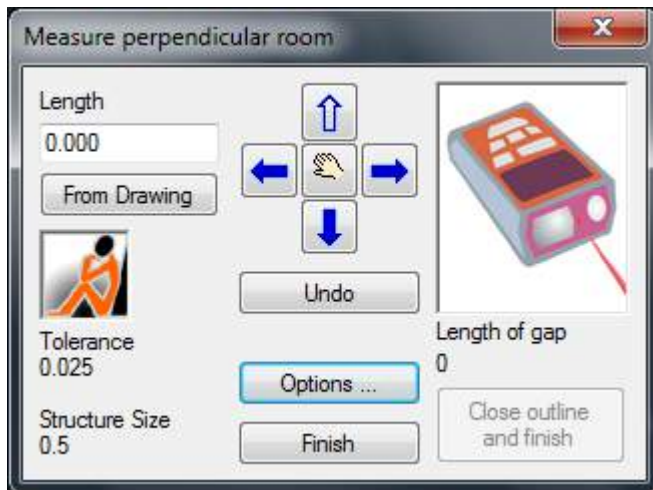
The measuring process is similar to the one of the already known room command.

One after another, the walls will be measured and be drawn into the appropriate direction. But since only right angles are allowed here, there are only four arrow keys and no diagonal function.

Enter the following values and press the arrow keys:

Length: 1.5 ⬆

1. Length : 2.499 ➡

2. Length : 5.001 ⬇

3. Length : 2.5 ⬅

4. Length : 3.498 ⬆

Now we have measured all around once. The dialog shows the following now:
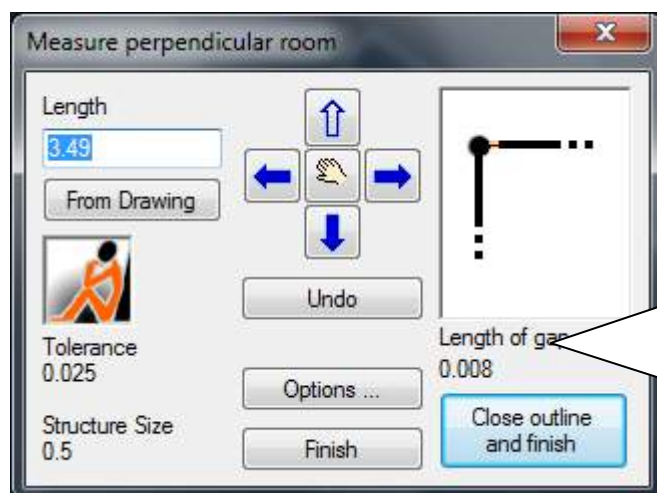
Due to the slightly differing values of opposite walls, in consequence of the error of measurement with the laser distance meter, a small remaining discrepancy will develop at the end.

If this is within the range of tolerance, the room outline will be adjusted.

Click on "Close outline and finish".

Switch on the "AccuracyReport" layer again. Use the AutoCAD command "layer" in order to do so. Now the dimensioning is visible for the room geometry again. The intermediate result looks as follows:
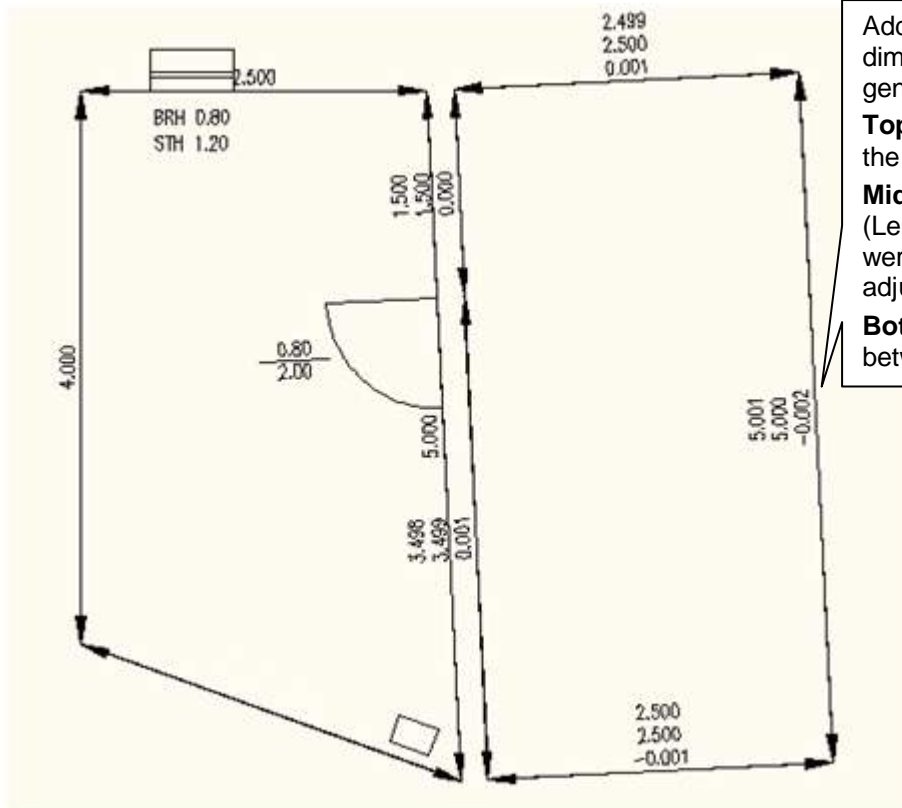
> Additionally to the wall a dimension is being generated:
>
> **Top:** measured values (in the tutorial: entered values)
>
> **Middle**: Adjusted values (Lengths of the walls which were generated after the adjustment.)
>
> **Bottom:** Difference between both values

For a better overview you may now switch the layer to "not visible" again.

Subsequent door reveal

In order to generate the **upper door reveal**, switch the OSNAP at "Endpoint" on and call up the AutoCAD command "Line".

Draw the top reveal line of the door.
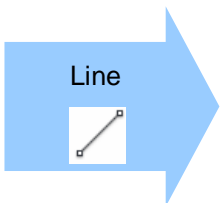
Line

Command: **line**

Enter first point: *(Snap (catch) the top point of the door reveal.)*

Enter next point or [Back]: *(Snap the initial point of room 2 and confirm with ENTER.)*

For the **lower door reveal** we will use the "Plumbing points" command, because since the wall of room 2 has not been split yet the initial point of the object is missing.

> The wall is not split here.

Perpendicular return
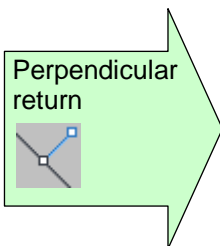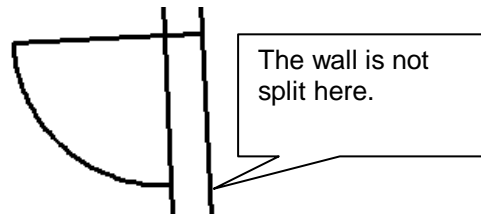
Command: **TCADPERPRETURN**

Select line: *(Part of the wall of room 2 where the door is located.)*

Current modes: Projection=2D Break=On POints=On Line=On

Direction of return or [Projection/Break/POints/Line/UNdo]:

(→Switch off generation of points

→"Break" and "Line" shall stay switched on)

Current settings: Projection=2D, Break=On, POints=Off, Line=On

Direction of return or [Projection/Break/POints/Line/UNdo]:

(Here click on the second point on the reveal of the door in room 1.)

The emerging plumb line is the door reveal. The wall in room 2 will be split at the door reveal as well.

Rectangular niche

Now the rectangular niche is to be measured. Call up the command "measure recess":

Measure recess/spur

Command: **TCADRECESSMEASURE**

Recess measure

Recess reveal
- Distance from corner a `1` ☐ In command line
- Point R1 on reveal

Recess width
- Recess width b `0.8` ☐ In command line
- Point R2 on reveal

Recess depth
- Recess depth t `0.5` ☐ In command line
- Point D in recess

☐ Keep settings for next measurements (automatic mode)

OK   Cancel
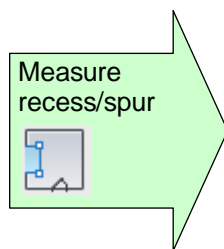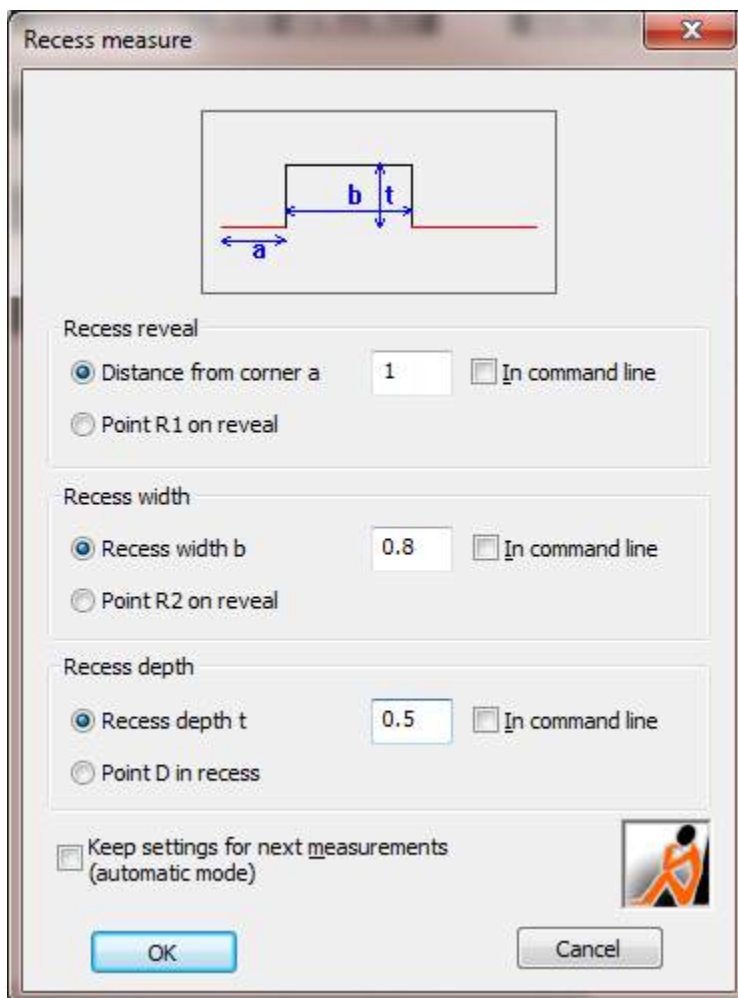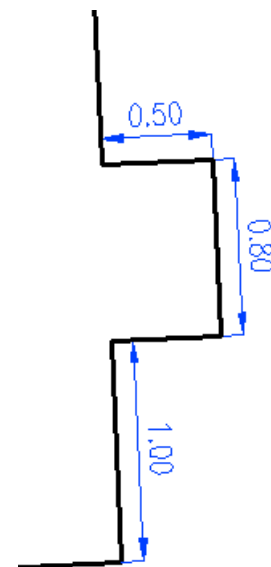
Enter 1.0 for the progressive distance, 0.8 for the width and 0.5 for the depth. Then close the dialog. Now you will be prompted within the command line to enter the wall, where the niche is to be inserted.

Select wall line or [UNdo]:   *(Click right wall at the lower end (close to the corner of the room.)*

Select a side: *(The scribed arrow shows the direction into which the niche will be inserted)*

Arched niche

For the construction of the round niche 3 points are necessary, those are the starting point and end point of the niche as well as a third point which is lying on the arc. During the measuring up a point on the arched has been marked and from that mark the distances to two corners of the room have been measured (arc intersection principle)
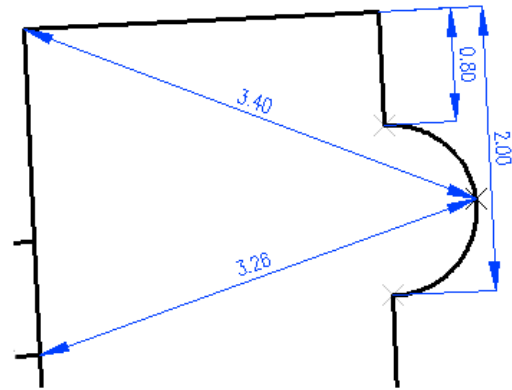
Starting and endpoint of the arched niche will be generated by the "fix point on a line" command. With this command it is possible to settle distances which have been measured along a line and mark them by a point and/or a break in the line.

Command: **FIXPOINTLINE**

Select reference line: *(Select wall at upper end.)*

Current modes: Projection=2D Break=On POints=On

Enter distance from corner or [Projection/Break/POints/UNdo]: **po**

→ *Switch on point generation again.*

→ *"Break" is to stay switched on.*

Current modes: Projection=2D Break=On POints=On

Enter distance from corner or [Projection/Break/POints/UNdo]: **0.8**

(Measured distance from the top right corner of the wall to the beginning of the niche.)

Current modes: Projection=2D Break=On POints=On

Enter distance from corner or [Projection/Break/POints/UNdo]: **2**

(Measured distance from the top right corner of the wall to the end of the niche.)

Current modes: Projection=2D Break=On POints=On

Enter distance from corner or [Projection/Break/POints/UNdo]: **ENTER**

The wall has now been divided into 3 lines. Delete the intermediate piece since that will be replaced by the arc.

We will construct the third point of the arc with the **arc intersection** method.

Command: **arcintersect**

First arc centre: (Top left corner of the room.)

First radius: **3.4**

(Measured distance to the mark of the arc.)

Second arc centre: *(Corner point lower door reveal.)*

Second radius: **3.26**

(Measured distance to the mark of the arc.)

Pick side:

(On which side of the arc intersection is the point to be drawn?)

Plot intersection [polYline/Point/Lines] <Lines>: **p**

(As result an AutoCAD point is to be generated.)

Fix points on a line

Arc intersection

Switch on OSNAP with the settings "End point" and "Node". Now an arc can be placed through the 3 points:
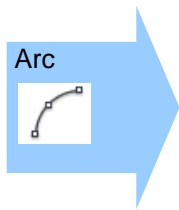
Arc	Command: **arc**	
	Specify start point of arc or [Center]:	*(Starting point)*
	Specify second point of arc or [Center/End]:	*(Point of arc intersection)*
	Specify end point of arc:	*(End point)*

Table

The U shaped table has a rectangular geometry and is aligned to the lower wall. For the rectangular drawing we will use the AutoCAD ORTHO mode.

In order to get a correct alignment we will call up the command "Align cursor". This command aligns the cross hairs towards a reference line:
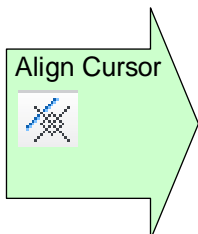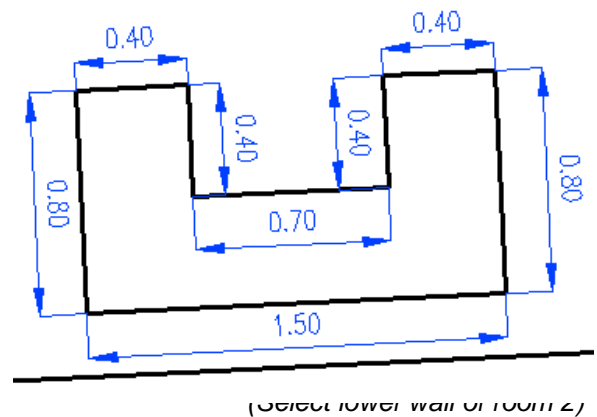
Align Cursor	Command: ALIGNCURSOR	
	Select object or <previousstate>:	*(Select lower wall of room 2)*
	Orthomode switched on.	

The commad activates the ORTHO mode automatically. Call up the AutoCAD command "Line" and click in room 2 to the position where the table is to be placed. You may position the table freely within the room.
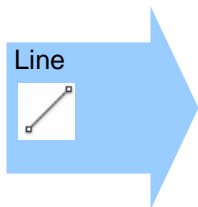
Line	Command: **line**	
	Specify first point:	*(Position for the lower right corner point of the table.)*
	Specify next point or [Undo]:**1.5**	

(Pull the mouse a little to the left, enter the measurement values and confirm with ENTER.)

The "Line" command is still active, so you can continue the polyline directly. Now pull the mouse up, the scribed line is perpendicular to the previous line and to the lower wall. Enter the measurement readings one after the other:

Specify next point or [Close/Undo]: **0.8**

(Pull the mouse a little up, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]:**0.4**

(Pull the mouse a little to the right, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]:**0.4**

(Pull the mouse a little down, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]:**0.7**

(Pull the mouse a little to the right, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]: **0.4**

(Pull the mouse a little up, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]:**0.4**

(Pull the mouse a little to the right, enter the measurement reading and confirm with ENTER.)

Specify next point or [Close/Undo]: **0.8**

(Pull the mouse a little up, enter the measurement reading and confirm with ENTER.)

Enter next point or [Close/Back]: **C** *(Close)*

In reality you would of course have the opportunity to transfer the values via the online connection from your laser distance meter directly instead of having to enter them manually.

Now reset the cross hair cursor.
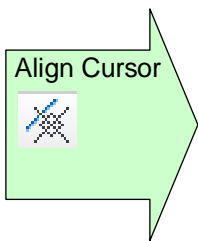
Command: ALIGNCURSOR

Select object or <previousstate>: *(Enter for reset to previous state.)*

Orthomode switched on.

Align Cursor

In the result both rooms should look like this now:
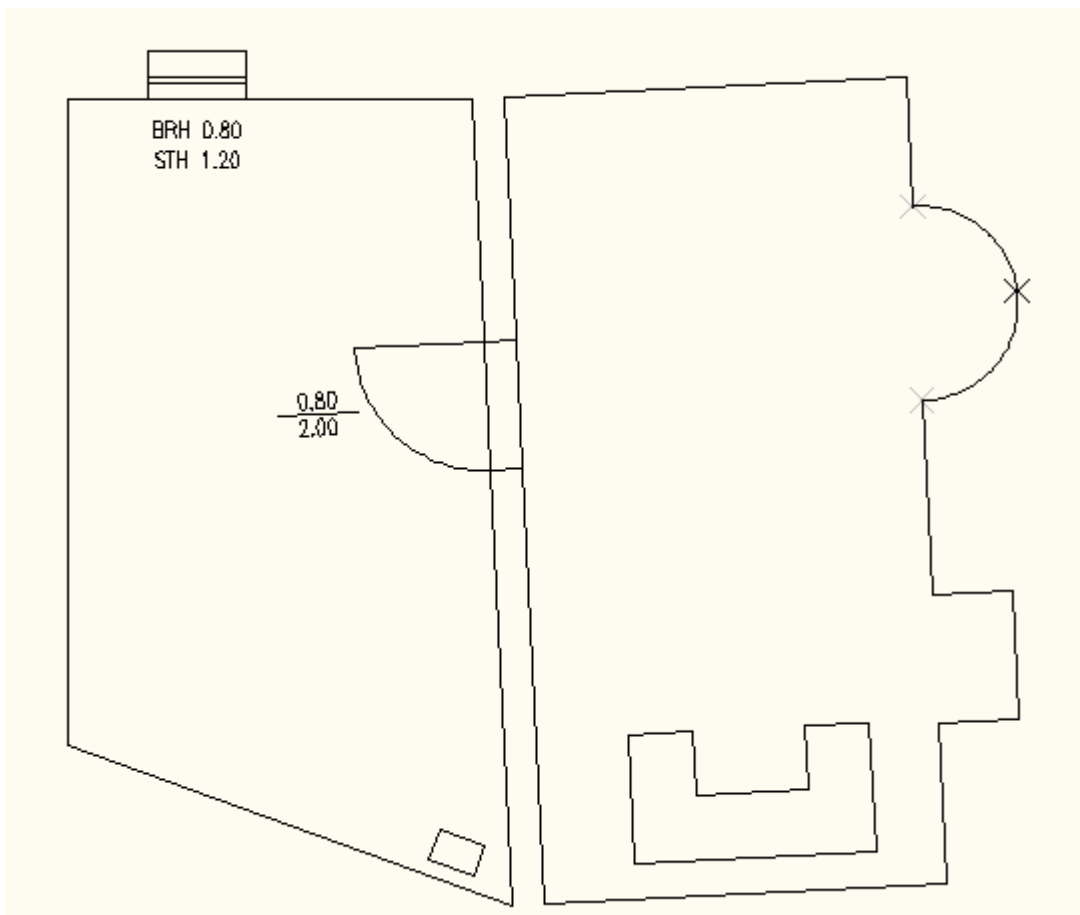
You have now gotten to know some of the most important modes of operation for the manual measurement.

The constructional possibilities and options within manual measurement are manifold. There are always several ways to get to the result, especially with the combination of AutoCAD commands with As-Built functions.

4 Edges and surface modeling

This chapter explains the workflow for the edge and surface modeling of solids.

4.1 Edge model

At first all necessary planes will be generated or measured with the →**Design – Draw plane** or →**Fit plane** commands for an edge model of a solid. In order to get an overview on the position of the planes rotate the drawing into the wanted position with the AutoCAD command "3dorbit". It is also useful to assign a color to the planes that are placed on the "KUBITPLANE" layer by default. It is possible to improve the display of the planes when changing the style from 2D-wire frame model to "Shaded" or "Realistic" with the AutoCAD command "VISUALSTYLES".
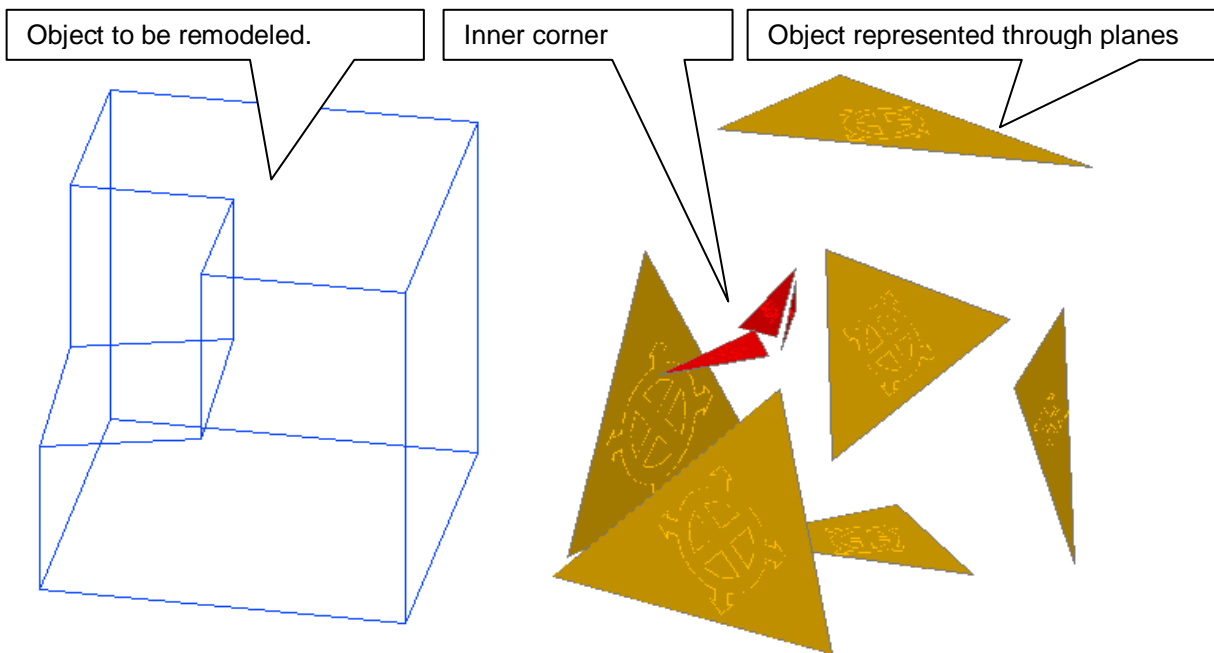
When the necessary planes have been generated, then 3 planes at a time will be spatially joined with the →**Design – Intersect plane – Intersection line (3 planes)** command. A tripod will be generated, which, in each case, represents one corner and the connected edges of the model. The length of the legs will be determined by the expansion of the kubit planes. The plummet of the "most extreme" expansion of the surface will be dropped to the intersection edge. Consequently it may occur that the legs of the tripod will not end, as perhaps expected, in the intersection point (compare left and right display).
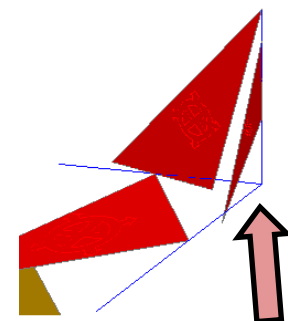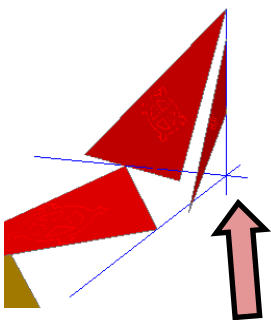
The tripod represents the internal corner.
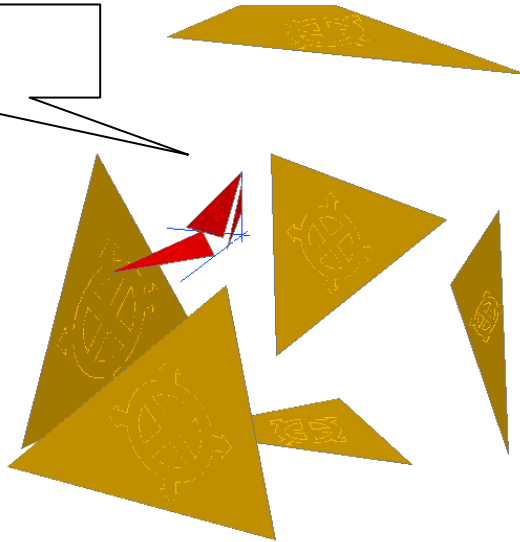
In most cases only a few adjoining corners have to be developed.
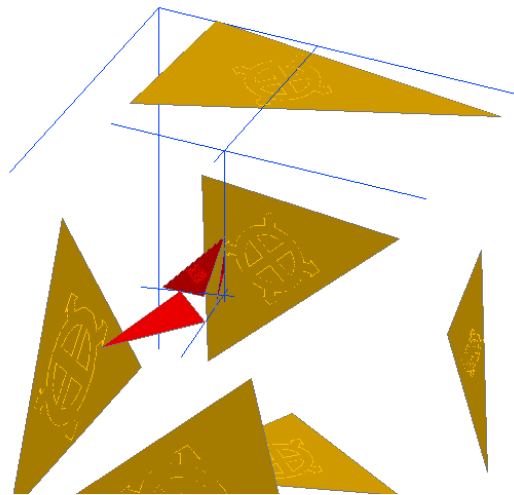
Now you can spatially join the edges with the AutoCAD command "Fillet". When rounding off edges the settings should be for mode = "Trim" and radius = 0. Here you click the edges that are to be preserved. It will then look as follows:

When "rounding off" click these two points, and then a corner will be generated.
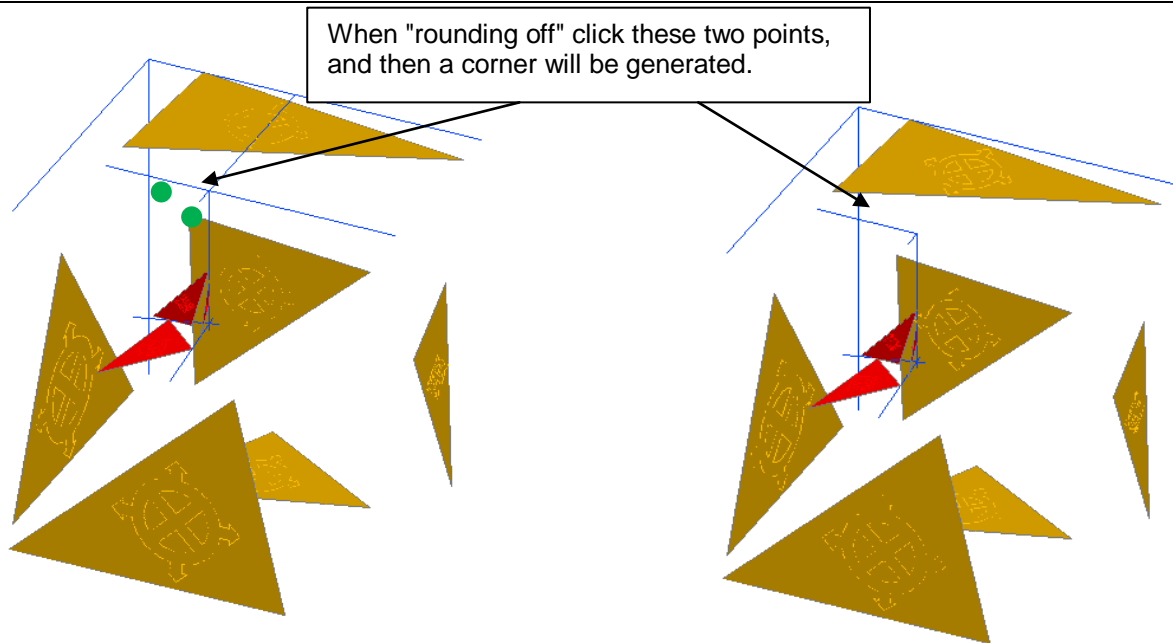
If two legs of two tripods are directed towards another, you may use the AutoCAD command "Join". It will generate one line from two collinear lines. This is also true for lines lying on top of each other that is when they overlap.

When connecting two collinear lines one single line will be generated.
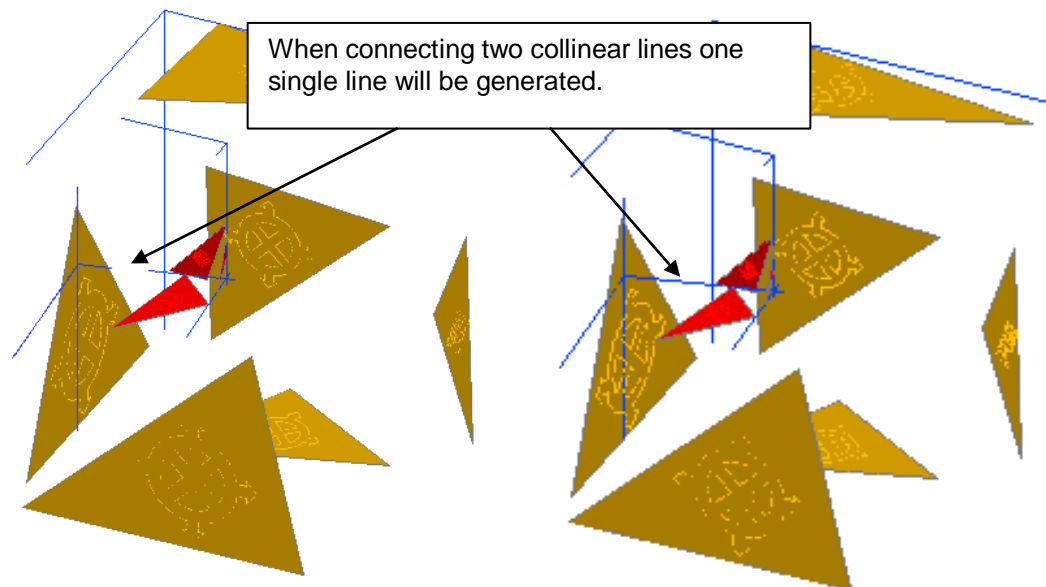
You may repeat these steps now until a complete edge model has been generated.
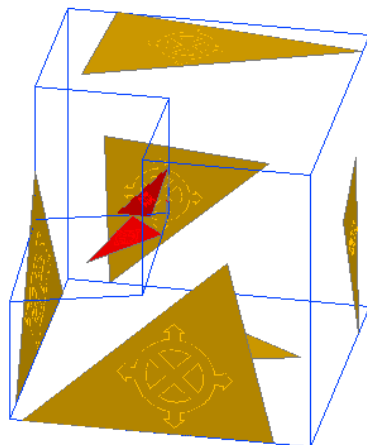
4.2   Surface model

As for an edge model of a solid, all planes will be generated or measured with the
→**Design – Draw plane** command at first for a surface model. Then 3 planes at a
time will be spatially joined with the →**Design – Intersect plane – Intersection line
(3 planes)** command or with →**Design – Intersect plane – Intersection point (3
planes)**. Please refer to the previous chapter "Edge model" for the procedure.
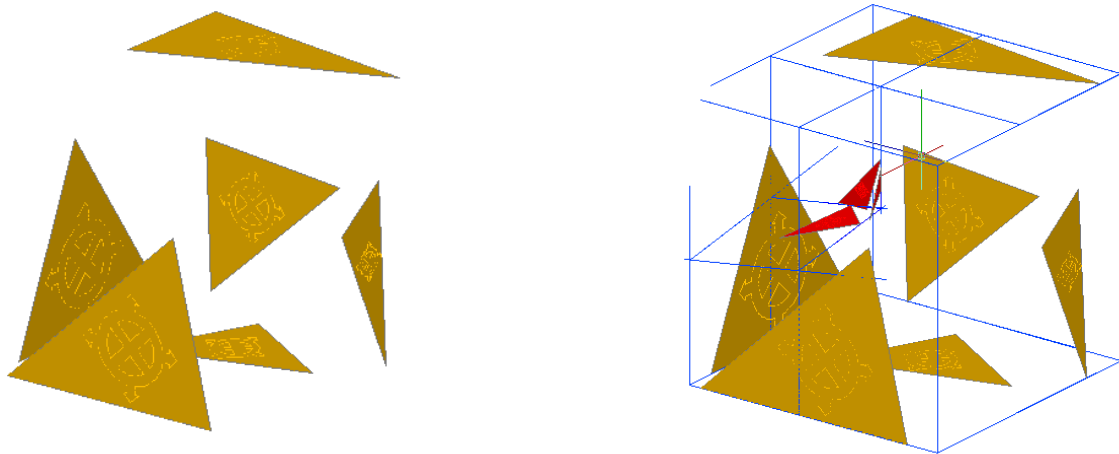
When you have generated all tripods, you may adapt the respective planes with the
→**Design – Modify plane – New rim** command to the correct boundary.

The boundary of the plane will be changed by setting the boundary
on the tripod corner points again.
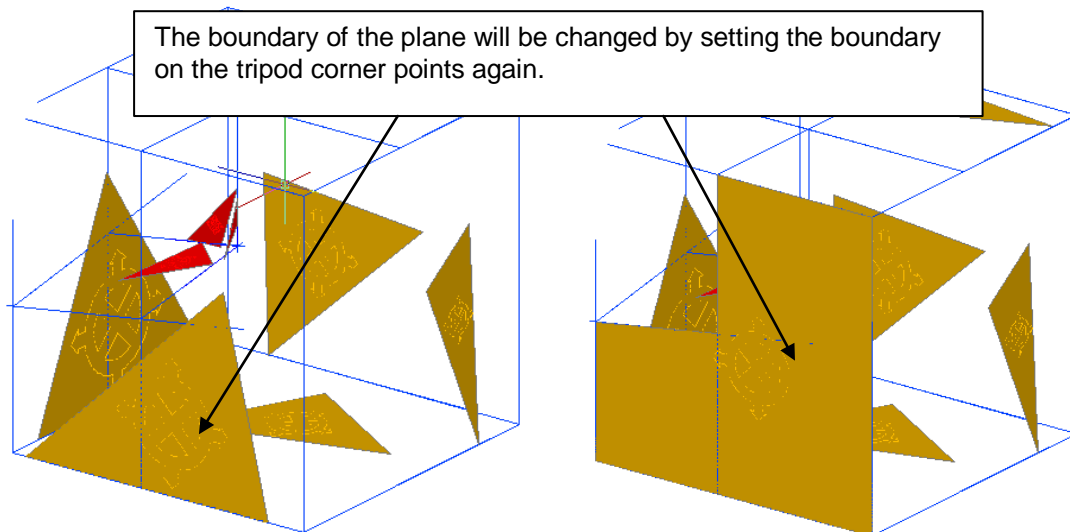
If you don not only want to use the surface model in As-Built for AutoCAD, you need
to convert it to pure AutoCAD objects. You can use EXPLODE to convert the kubit
planes into polylines. A polyline can easily be converted into planar surface
(AutoCAD command _PLANESURF) or a region (AutoCAD command REGION).

5 Adjustment and configuration of the total station interface

In this chapter you will learn how to adjust the As-Built TS interface to your specific demands.

5.1 The template files

The template files AS-BUILT_ENG.DWT and AS-BUILT_ENG_USA.DWT define several settings that are useful for working with As-Built for AutoCAD. These are mainly pre-defined layers and certain user specific settings (Text style, measurement units etc.). The layer structures are only suggestions. You can, of course, also work within As-Built for AutoCAD with your own or a changed layer structure.

All template files contain layout templates for common page formats. A legend, frames and fold marks are already prepared and make printing easier.

5.1.1 Layer structure of template AS-BUILT_ENG.DWT

This pre-defined layer structure only contains layers that As-Built needs for its commands. Else, there are no other pre-defined layers. You do not have to use this template file. If As-Built needs a layer which does not exist, it will be created when necessary.

5.1.2 Layer structure of template AS-BUILT_ENG_USA.DWT

Like the above this pre-defined layer structure only contains layers that As-Built needs for its commands. Else, there are no other pre-defined layers. In addition the AutoCAD drawing units are set to feet and inches. You do not have to use this template file. If As-Built needs a layer which does not exist, it will be created when necessary.

5.2 List of defined blocks

The sub-folder \DWGBASE contains all templates for blocks that are needed by As-Built.

Name of block	discription
ARCH_H	Arc height
ARCH_APEX	Height arc apex
BEAMDIM	Beam Dimensioning
BASICDIM1	Dimensioning clear height (without symbol)
BASICDIM2	Dimensioning clear height (symbol 1)
BASICDIM3	Dimensioning clear height (symbol 3)
CONTROL	Symbol for control point
CURRENT_STATION	Symbol for current station
DD_OW_OH	Door dimension
DD_SYMBOL	Door symbol

DOOR45_SYMBOL	Door symbol swing 45°
HT_GENERAL	Height symbol without special meaning
HT_OUTL_SOFF	Height symbol on lower edge – completed floor
HT_OUTL_SURF	Height symbol on upper edge – completed floor
HT_ROOM_OUTL	Height symbol floor / room height – completed floor
HT_ROOM_SOLID	Height symbol floor/ room height structure measurement
HT_SOLID_SOFF	Height symbol on lower edge – structure measurement
HT_SOLID_SURF	Height symbol on upper edge – structure measurement
INFOBLOCK.DWG	For additional information when importing points
NORTHARROW	North arrow symbol
NOZZLE	Example for block on pipe
NUMPOINT	Numbered point
OBSPOINT	Protocol for observation point
ROOMDATA	Room data
SLEEVE	Example for block on pipe
STAKEOUTPOINT	Documentation of staked out points
STATION	Symbol for station
WD_CHT_LHT	Breast height and lintel height
WD_OW_OH	Opening width and height

You can change the appearance of the templates according to your liking.

For some blocks you can also adjust the name of attributes or the name of the block as you like. In this case you have to adjust the respective value in the settings dialogue of As-Built. (➔**Settings**)

5.3 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 <u>must not</u> end with a comma.)

Remove all blocks from the file which you do not want to set as corporate standard. Please contact our support if you are not sure about the keys in the configuration file or if you cannot find them.

Remove all blocks from the file which you do not want to set as corporate standard. Please contact support@faro-3d-software.com 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 "<u>read only</u>" 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 Software 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 cause 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.
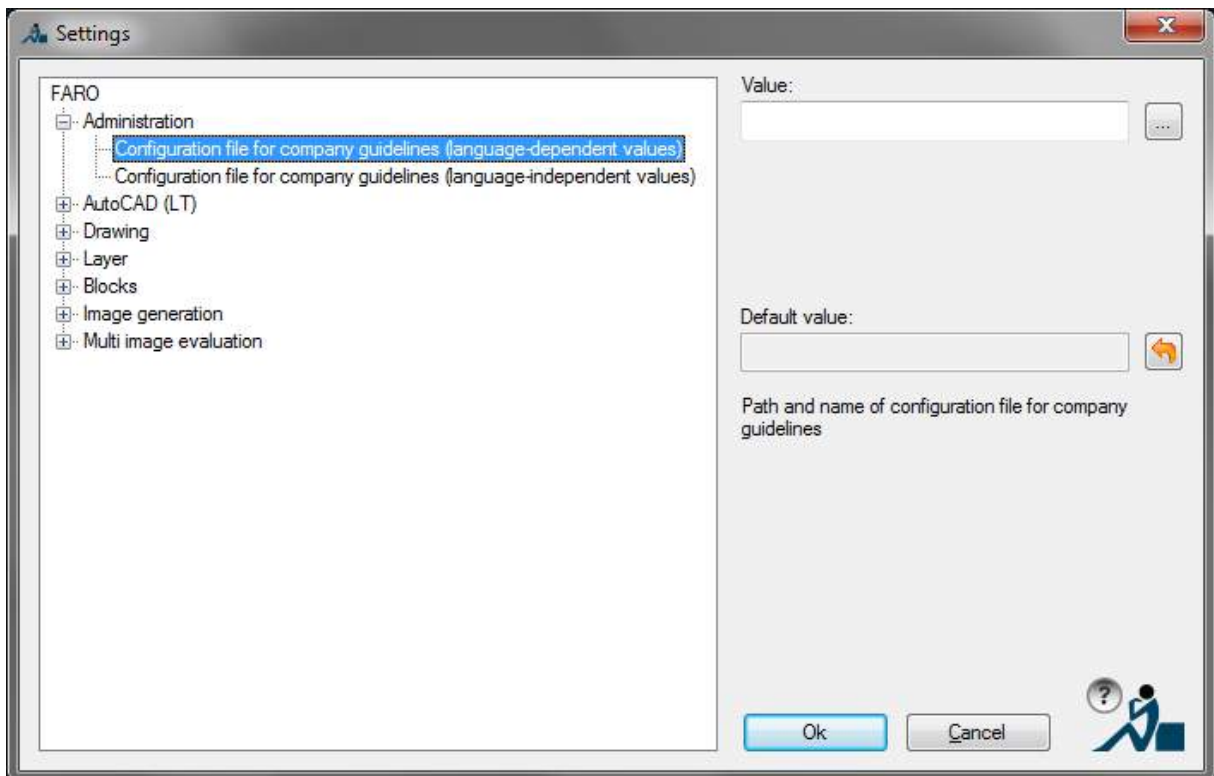
5.4 Usage of the measurement command 'MMT' in own AutoLISP routines.

The following example shows you how to include the measurement functionality of As-Built into your own measurement routines. You can find further information concerning programming with AutoLISP/VisualLISP in the AutoCAD documentation.

As-Built defines an AutoLISP function for you (MMT). The return value of (MMT) is a list of points (X,Y,Z). The function (MMT) can be run with two optional parameters.

- The first parameter contains the query string which appears in the measurement dialogue.

- The second parameter allows the combination of certain measurement modes. The string '-3D' leads to a 3D measurement. With the string '-KP' you can switch off the protocol function. You can also combine both strings.

Example: (MMT 'Measure the insertion point' '-3D-KP')

Let us assume that you want to insert a block TEST at the point to be measured with the command 'Testmeas'. You have to define:

```
(defun C:TESTMESS ( / punkt)
      (setq punkt (MMT))                                      ; MMT is called up
      (command "_INSERT" "TEST" punkt "" "" "")              ;Block is inserted)
```

This simple example only shows you the basic principle of including measurement commands of As-Built. With only a few lines of code you can access the measurement functionality of As-Built in your own AutoLISP routines. This enables you to enhance As-Built by commands that are 'custom made' for your own demands.

5.5   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".
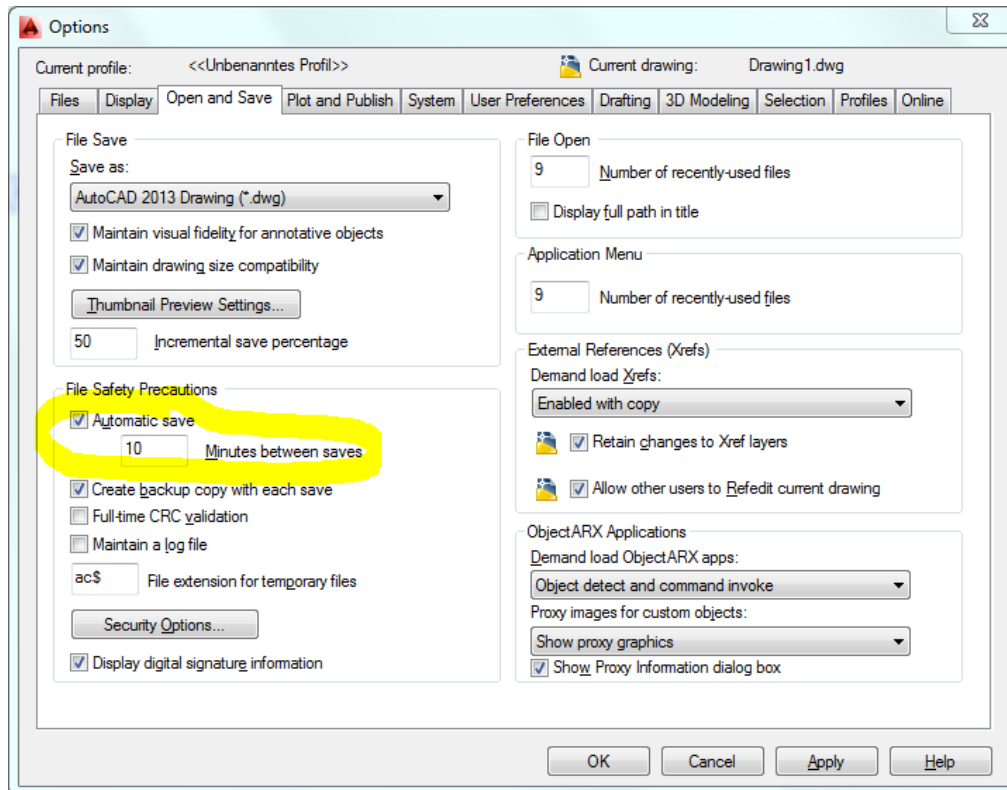
- The forwarding of coordinates calculated by As-Built to the AutoCAD command line is influenced by the current Object Snap Mode. The behaviour of As-Built is the same as when the coordinates are entered manually. In order to ignore the current Object Snaps when measuring, following settings are necessary in the options of AutoCAD:
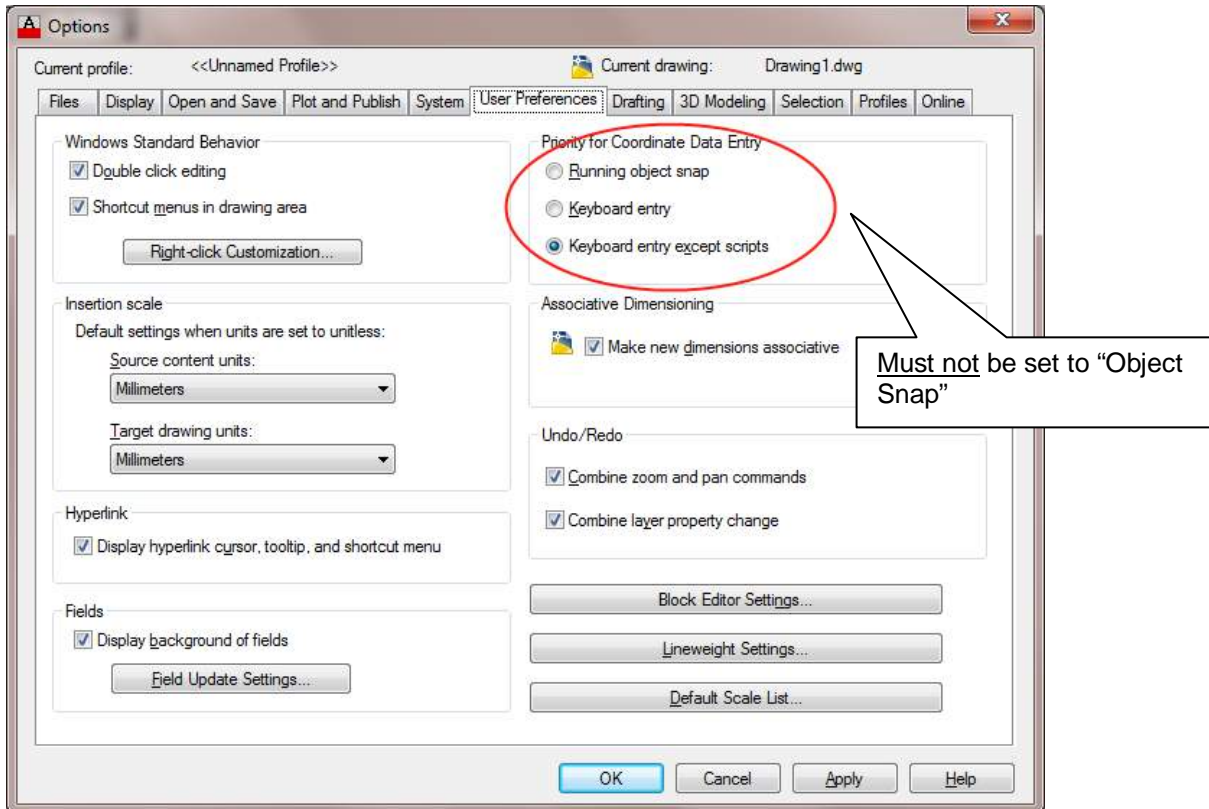
This is done automatically with every start of As-Built (AutoCAD variable OSNAPCOORD=2).

6 Advice regarding supported hand-held lasers

6.1 Bluetooth Connection

To be able to work with a wireless Bluetooth connection you need to have an integrated Bluetooth chip set in your computer. Should your computer not have such a chip set it is possible use a Bluetooth adapter, in the form of a USB stick.

In chapter **7.2** you will find detailed instructions on how to setup a Bluetooth connection for you specific operating system.

6.2 Configure the interface

To establish a connection you must tell As-Built the type of your laser distance meter. Perform the following steps:

- Start AutoCAD with As-Built and open any drawing.

Open the hand held laser's settings dialogue: "Hand held laser – Connect hand held laser" or by using . In the Hand held laser-Box click on the double angled brackets .

- Under the tab "Connection" select the appropriate instrument (e.g. "Leica DISTO") and to the right the discovered device (e.g. Disto D8 with the serial number 592010161)

- Thereafter click on "Connect". If everything is correct, the connection to the hand held laser will be established and the box changes to green.

Further details to the hand held laser settings can be found in the Command Reference under →**"Hand held laser – Connect hand held laser ".**
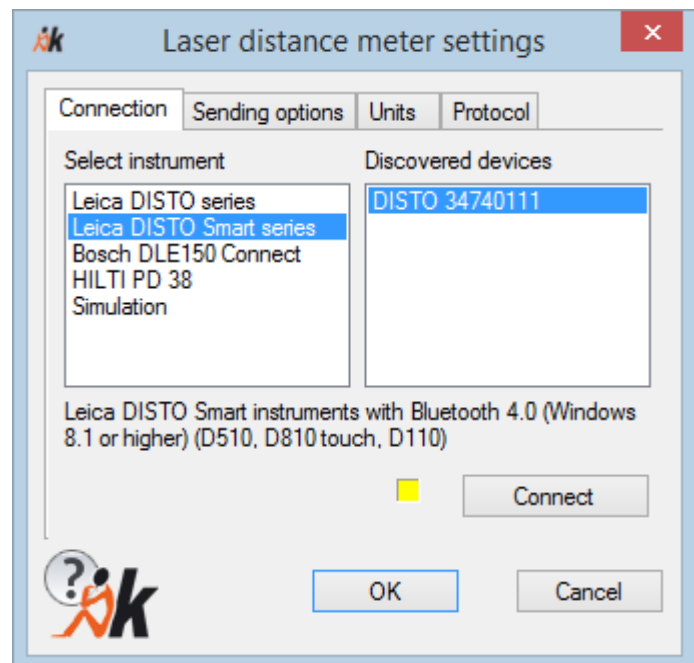
6.3   Leica Disto Plus

Console
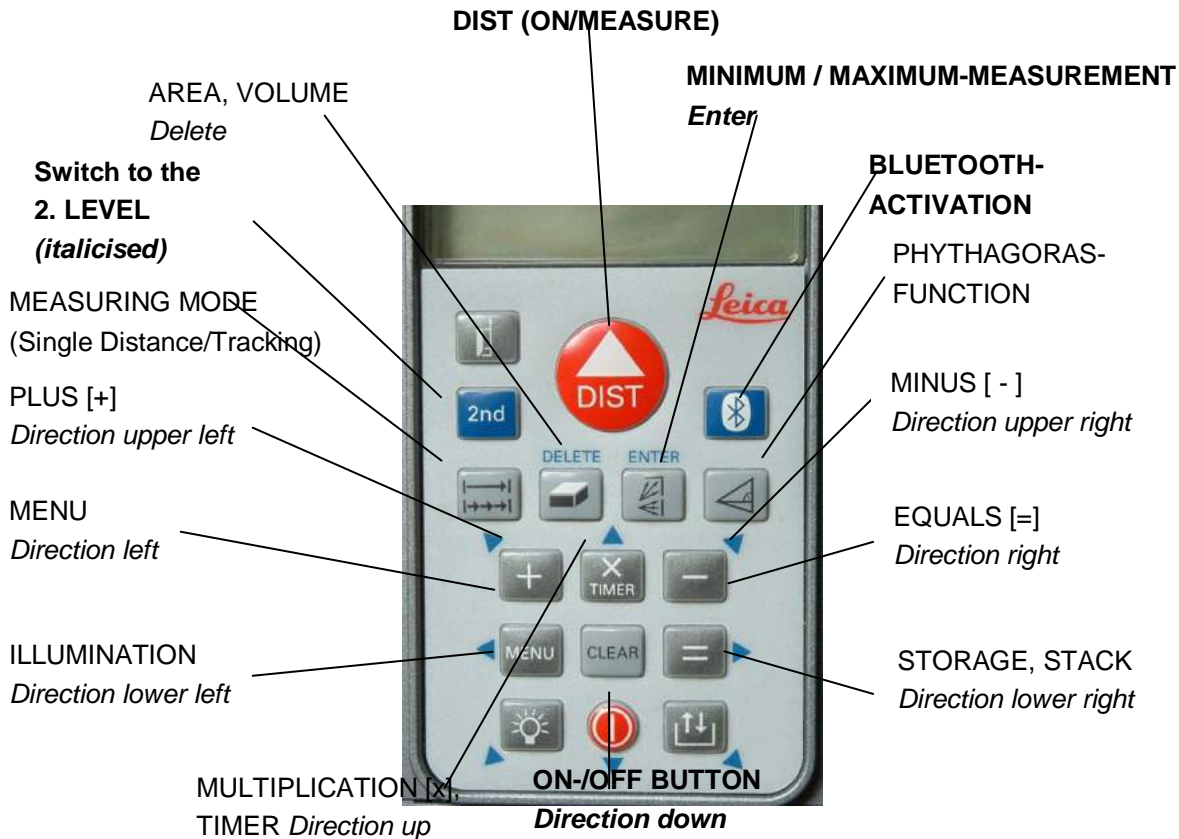
DIST (ON/MEASURE)

AREA, VOLUME
Delete

MINIMUM / MAXIMUM-MEASUREMENT
Enter

**Switch to the
2. LEVEL**
(italicised)

BLUETOOTH-
ACTIVATION

MEASURING MODE
(Single Distance/Tracking)

PHYTHAGORAS-
FUNCTION

PLUS [+]
Direction upper left

MINUS [-]
Direction upper right

MENU
Direction left

EQUALS [=]
Direction right

ILLUMINATION
Direction lower left

STORAGE, STACK
Direction lower right

MULTIPLICATION [x],
TIMER *Direction up*

ON-/OFF BUTTON
Direction down

For detailed information please read the instrument's manual.

Bluetooth communication

To activate the Bluetooth of the hand-held laser press the blue Bluetooth button. The Bluetooth symbol appears in the display. If it blinks it is not yet connected. In order to transfer data from the Disto to As-Built you need a Bluetooth connection between your computer and the Disto. Please refer to chapter .

When the Bluetooth connection is ready the Bluetooth symbol in the Disto's display stops blinking and in the laser box of As-Built the communication indicator switches to green.

Now you can measure by pressing the **DIST** button. By means of the blue arrow buttons which you enable with the **2nd** button you can send the measured values to the computer (As-Built). You may also use the *ENTER* button to transfer data but in the laser box there won't be a direction sign just a blue cross.

Minimum / Maximum function

The Disto Plus provides functionality to measure a minimum or maximum distance from a certain point. Please refer to the Leica manual for the activation of the Minimum- or Maximum-Tracking.
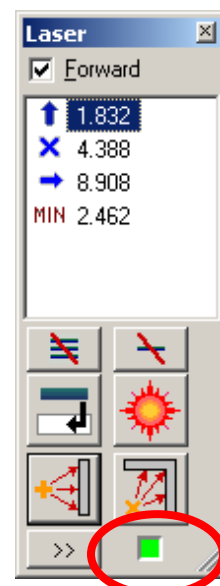
6.4 Leica Disto A6

Console

ON/DIST (ON/MEASURE)

2. FUNCTION LEVEL
(italicised)

BLUETOOTH

PLUS [+]

MINUS [-]

TIMER
Direction upper left

OBLIQUE MEASUREMENT
(Pythagoras)
Direction upper right

EQUALS [=]
Direction up

AREA, VOLUME
Direction left

ILLUMINATION
Direction right

STACK
Direction down

MENU
Direction lower right

MEASURING PLANE
Direction lower left

For detailed information please read the instrument's manual.

Bluetooth communication

To activate the Bluetooth of the hand-held laser press the blue Bluetooth button. The Bluetooth symbol appears in the display. If it blinks it is not yet connected. In order to transfer data from the Disto to As-Built you need a Bluetooth connection between your computer and the Disto. Please refer to chapter **7.2**.

When the Bluetooth connection is ready the Bluetooth symbol in the Disto's display stops blinking and in the laser box of As-Built the communication indicator switches to green.

Measuring and sending

Now you can measure by pressing the **DIST** button. By means of the blue arrow buttons which you enable with the **2nd** button you can send the measured values to the computer (As-Built). You may also use the **ENTER** button to transfer data but in the laser box there won't be a direction sign just a blue cross.

Minimum / Maximum function

The Disto A6 provides functionality to measure a minimum or maximum distance from a certain point. Please refer to the Leica manual for the activation of the Minimum- or Maximum-Tracking.

To transfer data to As-Built you do as follows:

- If you like to transfer the mean value switch to the second function level and press the direction arrow button you wish

- If you like to transfer the **minimum** value switch to the second function level (if not already done) and press on **PLUS [+]** until minimum value starts blinking. By pressing the **BLUETOOTH** button you send the data to As-Built.

- If you like to transfer the **maximum** value switch to the second function level (if not already done) and press two times for a longer while on **PLUS [+]** until maximum value starts blinking. By pressing the **BLUETOOTH** button you send the data to As-Built.

6.5 Leica Disto D8

Key pad
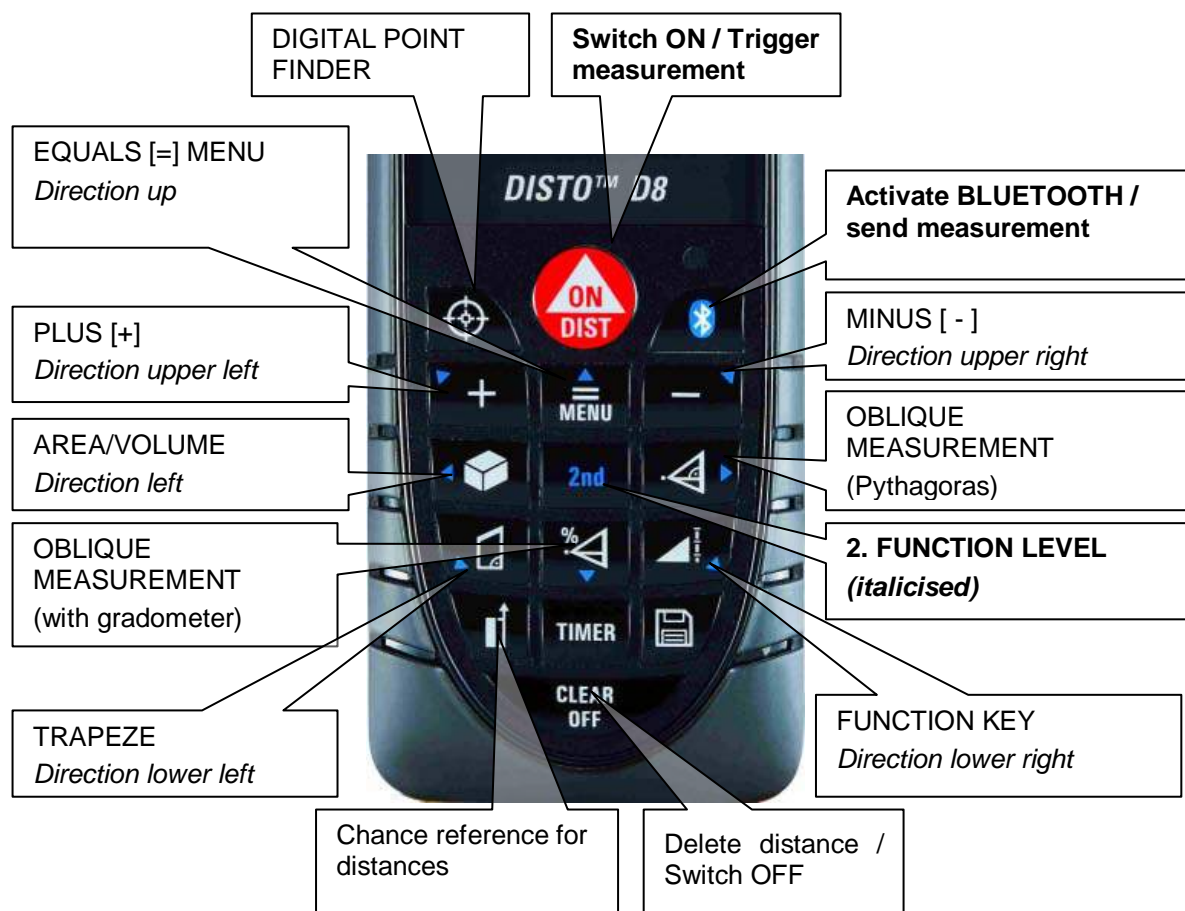
Please read the Leica manual for detailed information.

Bluetooth communication

To activate the Bluetooth of the Disto press the Bluetooth button. The Bluetooth symbol appears in the display. If it is grey it is not yet connected. In order to transfer data from the Disto to As-Built you need a Bluetooth connection between your computer and the Disto. Please refer to chapter **7.2**.

When the Bluetooth connection is ready the Bluetooth symbol in the Disto's display becomes blue and in the laser box of As-Built the communication indicator switches to green.

Measuring and sending

Now you can measure by pressing the **DIST** button. By means of the blue arrow buttons which you enable with the **2nd** button you can send the measured values to the computer (As-Built). (Please note: the second function level is active when the 2nd-symbol is displayed blue). You may also use the **ENTER** button to transfer data but in the laser box there won't be a direction sign just a blue cross. The **BLUETOOTH** button can also be used for sending if the 2nd function level is de-activated. The **BLUETOOTH** button has to be pressed twice for sending.

Minimum / Maximum function

The Disto D8 provides functionality to measure a minimum or maximum distance from a certain point. In order to start the Minimum / Maximum measurement on the laser distance meter, press the DIST button until you hear a beep. Then the Disto will perform continuous measurements until you stop it by pushing the DIST button again. Following that three measurement values will be displayed, the minimum, the maximum and the last measured distance.

To transfer data to As-Built you do as follows:

- Press the **BLUETOOTH** button and hold it until all values in the display are underlined in blue.

- With the buttons Plus [+] and Minus [-] you may select a value to be send. This value then will be underlined.

- By pressing **BLUETOOTH** twice you send the value to As-Built. (Caution: 2nd must not be activated, grey icon in display)
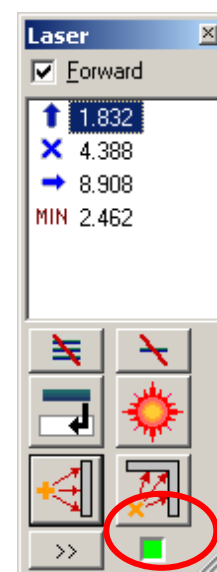
6.6   Leica Disto D3a BT /D330i

Key pad

On / measure distance

Activate Bluetooth / send measurement value

Horizontal mode on/off.
Direction clockwise.

Plus
Direction up left

Minus
Direction up right

Measure area/volume
Direction left

Memory
Direction right

Change reference point for distance measurement
Direction down left

Measurement functions
Direction down right

Self-timer for distance measurement
Direction down

Clear distance / off

Bluetooth communication

To activate the Bluetooth of the Disto press the **Bluetooth symbol** button. The Bluetooth symbol appears in the display. If it is blinking, the connection has not yet been established. In order to transfer data from the Disto to As-Built you need a Bluetooth connection between your computer and the Disto. Please refer to these paragraphs for further information:

- Working with the laser distance meter – first steps

- Advice regarding supported hand-held lasers – Communication via Bluetooth

When the Bluetooth connection has been established, the Bluetooth symbol on the display of the laser distance meter will become permanently visible and the status within the laser distance meter dialogue (Distobox) will switch to green.

Measuring and sending

Now you can measure by pressing the **DIST** button. By means of the blue arrow buttons which you enable with the **2nd** button you can send the measured values to the computer (As-Built). (Please note: the second function level is active when the 2nd-symbol appears in the display). You may also use the **BLUETOOTH** button to transfer data to the Disto but then the direction statement will be missing. Within the laser distance meter box this will be indicated by a blue X. The **BLUETOOTH** button can also be used for sending if the 2nd function level has been de-activated. For this the **BLUETOOTH** button has to be pushed twice.

Minimum / Maximum function

The Disto D3a BT provides the functionality to measure a minimum or maximum distance from a certain point. In order to start the Minimum / Maximum measurement on the laser distance meter, press the DIST button until you hear a beep. Then the Disto will perform continuous measurements until you stop it by pushing the DIST button again. Following that three measurement values will be displayed, the minimum, the maximum and the last measured distance.

Unfortunately the minimum or maximum values cannot specifically be sent to As-Built, since the Disto D3 BT does not have a function to select the desired value on the display. Use the MIN/MAX measurement directly within the As-Built Distobox. Then the desired value will be determined.
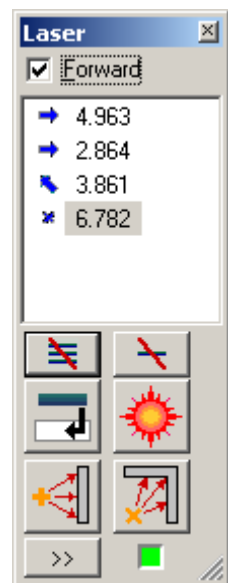
6.7 Leica Disto D110, D510, D810 touch

These instruments belong to new Disto generation with SMART Bluetooth (Bluetooth 4.0).

This technology has specific requirements:

- At least Windows 8.1

- Bluetooth 4.0 is built-in to the computer or it is expanded with a USB stick

Establish a Bluetooth connection

Switch the Disto on. Bluetooth is thereby automatically activated.

Allow the discovery of new devices in the Windows Bluetooth environment. When the Disto has been found, go to "Pair".
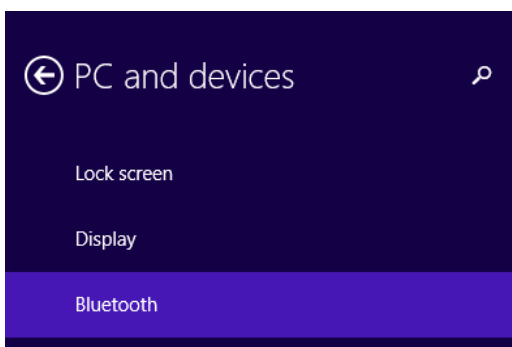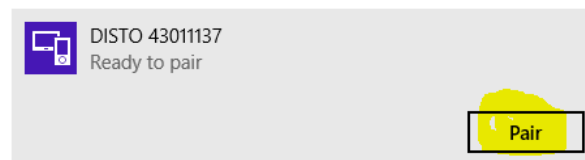
Settings in As-Built

Open the "Laser distance meter settings", tab "Connection" and select "Leica DISTO Smart series". Then click on "Connect". The control light changes from yellow to green and thereby is the connection established. Close the settings dialogue box.
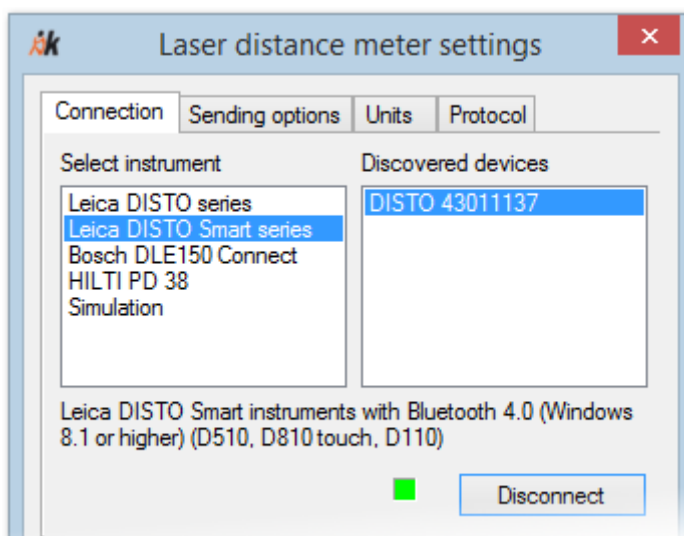
The measuring options available vary between the individual Disto SMART models:

6.7.1 Measuring with the D110
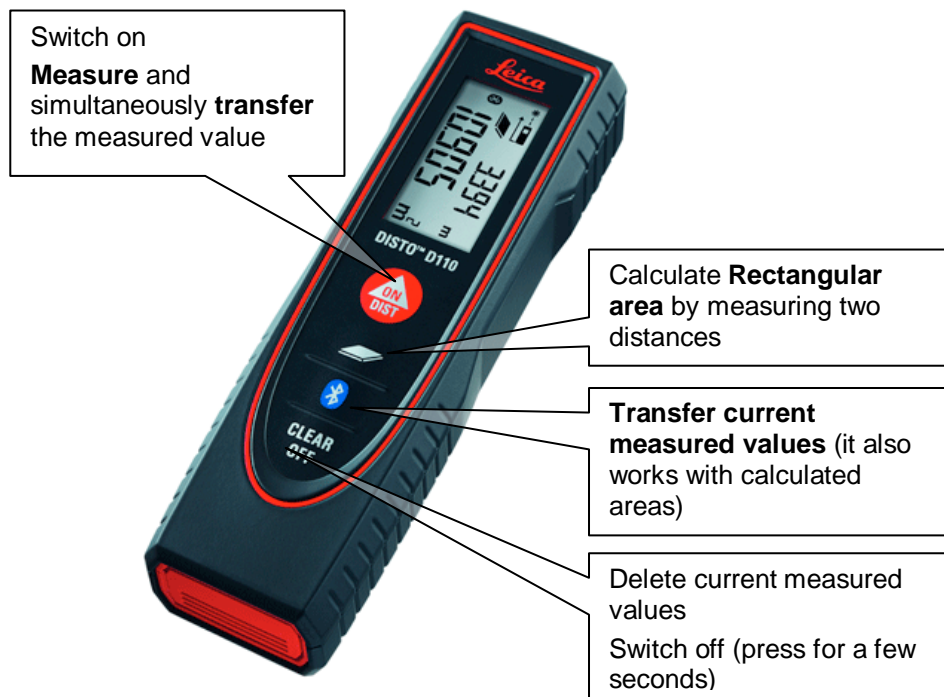
Switch on **Measure** and simultaneously **transfer** the measured value

Calculate **Rectangular area** by measuring two distances

Transfer current measured values (it also works with calculated areas)

Delete current measured values

Switch off (press for a few seconds)

You can either trigger a measurement directly on the Disto or in As-Built (red laser button).

The transfer of directional information (blue arrow) is not possible with this Disto model.

Minimum /Maximum – Function

The Disto D110 has a permanent measuring function. Press the DIST button once, a long beep is emitted. The Disto carries out permanent measuring until you press the DIST key again to stop the measuring.

You must previously start the MIN or MAX function in the hand laser box. After ending the permanent measuring stop the MIN/MAX function and you will be given the minimum or maximum measured value.

Laser

☑ Forward

✳ 1.75 m
✳ 3.25 m
✳ 2.95 m

>>
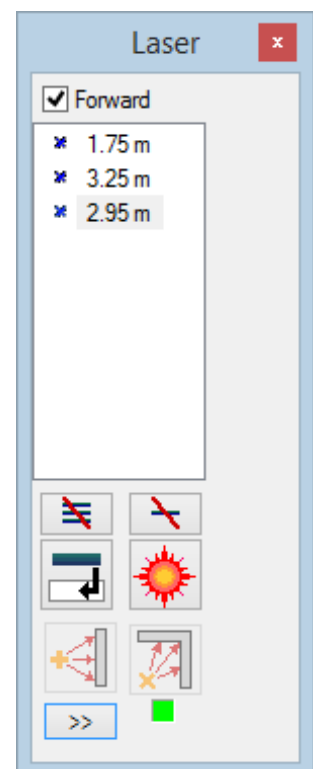
6.7.2   Measuring with the Disto D510 (E7500i)

Switch on **Measure** and simultaneously **transfer** the measured value

"=" Button: **Transfer current measured value** (also functions with values such as areas, volumes, horizontal distances etc.)

The **arrow keys** can unfortunately not be used for transferring directional information.

They are only for navigating through the menus.

Delete current measured values

Switch off (press for a few seconds)

On the D510 a measurement can only be triggered from the Disto. It is unfortunately not possible to trigger it from As-Built

Minimum /Maximum – Function

The Disto D510 has a permanent measuring function. Press the DIST button once, a long beep is emitted. The Disto carries out permanent measuring until you press the DIST key again to stop the measuring.

You must previously start the MIN or MAX function in the hand laser box. After ending the permanent measuring stop the MIN/MAX function and you will be given the minimum or maximum measured value.
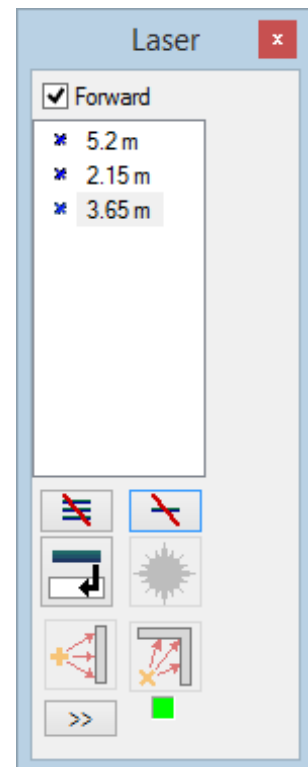
6.7.3 Measuring with the D810 touch

Touch-Display.

 All functions, including triggering a measurement, are also possible using the screen based user interface.

Switch on **Measure** and simultaneously **transfer** the measured value

The **arrow keys** can also be used for transferring directional information (see the instructions below)

Delete current measured values

Switch off (press for a few seconds)

"=" Button: **Transfer current measured value** (also functions with values such as areas, volumes, horizontal distances etc.)

Settings

So that everything functions correctly make the following **settings on the Disto:**

- Switch on the keyboard mode.

If this symbol is displaying **"ON"(green)**, then the cursor on the computer can be controlled by the arrow keys.

Tap this symbol to activate the keyboard mode.
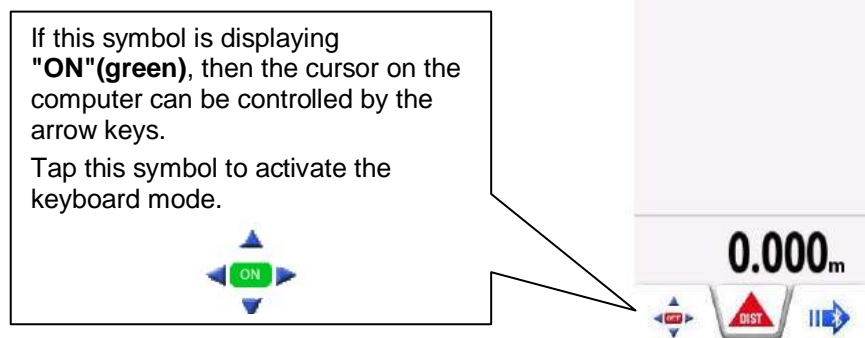
- Bluetooth settings: Select "Send as Text" or "Send as number":

 or
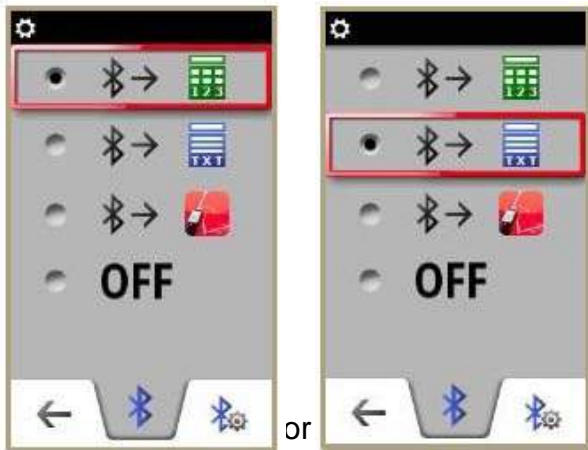
Measuring

You can either trigger a measurement directly on the Disto or in As-Built (red laser button).

The triggering of a minimum or maximum measurement is unfortunately not possible with this model. Therefore the respective buttons are also disabled (grey)

The transfer of **distance plus directional information (blue arrow)** functions in the following manner:

After every measurement the directional arrow dialogue box is displayed in As-Built:

Either click the appropriate arrow key in the dialog field, or press the **arrow key on the Disto, followed by the "=" key.** Thereupon the measured value with the arrow is displayed in the Disto box.

The Disto has only four keys instead of eight. The sloping arrows (45° directions) are missing. To transfer sloping directional information press two keys on the Disto followed by the "=" key.

An example: You want to send a measured value together with directional information .

On the Disto you must press in the following sequence

 or
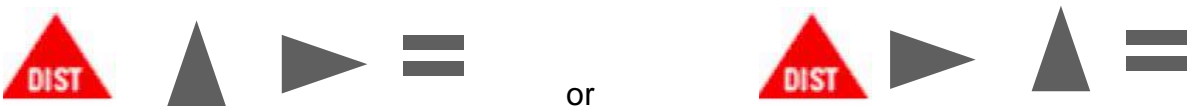
Thereupon in As-Built you see the value and direction displayed in the Disto box:

Should you want to send a **distance without directional information**, then after the measurement press the "=" key (either on the Disto or in the arrow dialogue box). Then in the Disto box the measured value is displayed with an X rather than the arrow.

Should you want to **always work without the arrow key** and only transfer the measured value, then you can switch off the arrow dialogue box, then activate the following option in the advanced Disto box As-Built:
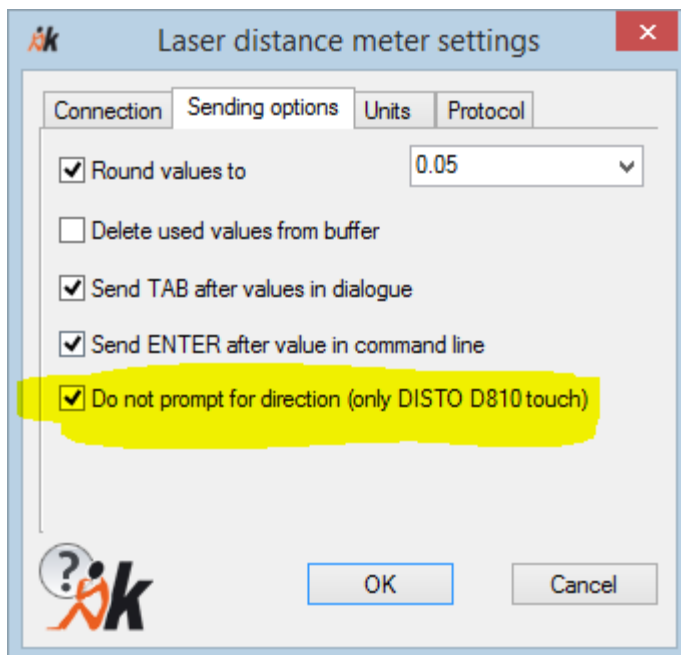
Minimum /Maximum – Function

The Disto D810touch has a permanent measuring function. Press the DIST button once, a long beep is emitted. The Disto carries out permanent measuring until you press the DIST key again to stop the measuring.

You must previously start the MIN or MAX function in the hand laser box. After ending the permanent measuring stop the MIN/MAX function and you will be given the minimum or maximum measured value.

6.8 Bosch DLE 150 Connect

Key pad

PERMANENT POINTER
MODE

ON/OFF

DISPLAY ILLUMINATION

MEASURE/SEND

PERMANENT
MEASUREMENT and
MIN/MAX FUNCTION

OBLIQUE LENGTH

MEASURING MODES:
LENGTH, AREA, VOLUME

DELETE/CANCEL

MEASURED VALUES MEMORY:
ADDITION, SUBTRACTION, SAVE
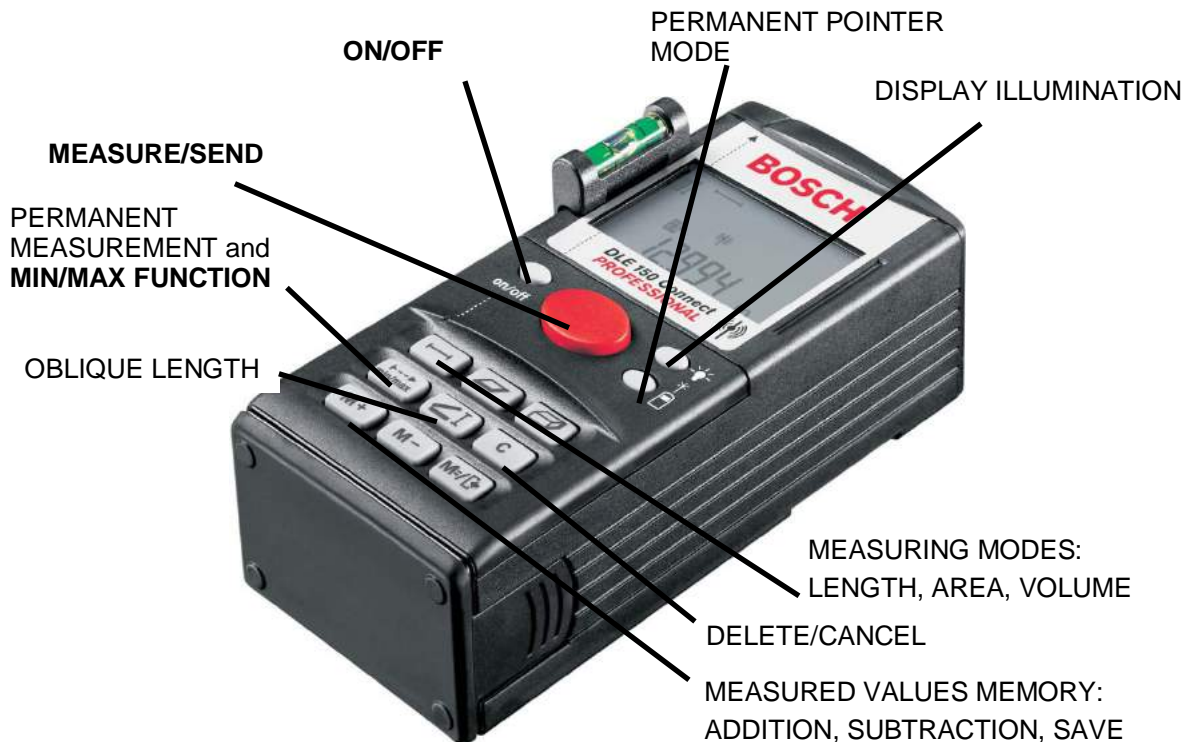
For detailed information please read the instrument's manual.

Bluetooth communication

Using a Bosch DLE 150 Connect hand-held laser you do not have to activate the Bluetooth connection. The Bluetooth is active as soon as you switch on the laser instrument and permanently stays active until you switch the laser off. This is displayed by the following symbol .

The distance measurement is started by pressing the big red button. The resulting data is automatically sent to As-Built if the Bluetooth connection works. Please also read chapter **7.2**.

The Bosch DLE 150 Connect provides functionality to measure a minimum or maximum distance from a certain point. Please refer to the Bosch manual for the activation of the Minimum- or Maximum-Tracking.

6.9   Hilti PD38

Key pad

Optical Target
Finder

Reference switch ──────────── **ON/OFF**

────── Measure

VOLUME function ──────────── Add distance

────── **MEASURE**

Area function ──────────── Subtract distance

Level (horizontal) ──────────

────── Display light

Reference peek ──────────────
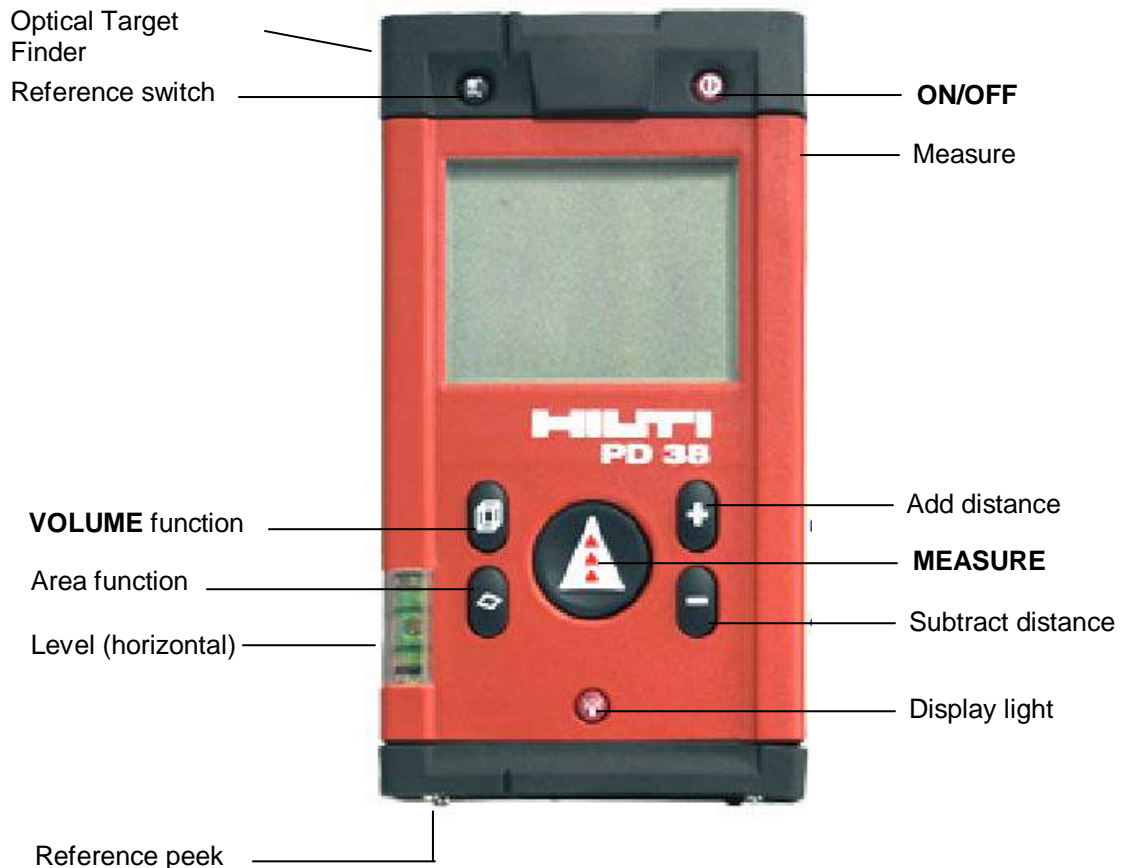
For detailed information please read the instrument's manual.

Bluetooth communication

Ensure the Bluetooth has been activated in the menu of your Hilti PD38 (Antenna symbol 🬀 shown in the display). The menu is activated by pressing the 'ON/OFF' button for ca 2 seconds. The 'Volume' button is now used to switch the Bluetooth on or off.

If in As-Built you click on the button 'Connect' in the hand laser dialogue and the connection was successful, you will hear a beep sound from the Hilti as confirmation. Additionally for ca 2 seconds the 'waves' at the antenna symbol are shown in the display.

Bluetooth active

Bluetooth connection
to external device

While the Bluetooth connection still persists this symbol (•) is not displayed permanently. The connection exists as long as the connection indicator of As-Built shows still green, so measured values can be send. (Exception: when initiating a measurement in As-Built, this symbol (•) is displayed for 2 seconds too.)

The distance measurement is started by pressing the big round button in the middle (or the measure button at the right side). The resulting data is automatically sent to As-Built if the Bluetooth connection works. Please also read chapter **7.2**.

By pressing the **MEASURE** button once, the laser is activated. Pressing it a second time executes the measurement and sends the data to As-Built. The measurement can also be released from As-Built.

The Hilti is switched off by means of the On/Off button or it switches of itself after 10 minutes. The connection to As-Built is then disconnected and the connection indicator in the laser box switches to yellow.

Tracking mode

The tracking mode is activated by pressing and holding the **MEASURE** button for 2 seconds. During the tracking the Hilti permanently sends measures to As-Built (ca every 2 seconds).

Remark

Generally there are two possible settings for the distance units at the Hilti PD38. The unit that is measured and sent (metres) and the unit that is shown in the display. That means independent from the displayed distance units the laser meter sends measured values in metres. If you in As-Built like to work with other units than metres you may set a different unit within the laser box of As-Built (command reference →**Hand-held laser – Connect**)
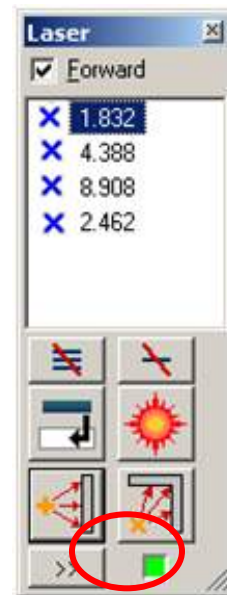
7 Advice regarding supported instruments

Regardless of the instrument type the necessary settings consist of setting the correct interface parameters as well as measuring and registering with the correct recording mask. Usually, As-Built uses the standard recommendations of manufacturers for all settings.

7.1 Refraction correction

Usually the tachymeter corrects the measured distance values regarding the atmospheric conditions by an adjustable factor in ppm. For further information please refer to the manual of your instrument.

As-Built itself corrects the zenith angle (v in rad) regarding refraction and curvature of the earth and in order to do so uses an (adjustable[4]) refraction coefficient of k=0.13.

$$v' = v + (k - 1) * sH / 2R$$

sH = measured horizontal distance (in metres)

R = Earth's radius = 6380000 metres

K=0.13 is a mean value, in reality the refraction coefficient is subject to considerable fluctuation, especially close to ground level and during heavy solar radiation.

Conclusion:

In standard conditions, close to ground level and an accuracy requirement in the range of the measurement accuracy of the instrument (about 5 mm), the justifiable horizontal measurement distance is around **150-200 m**.

Larger level distances cannot be measured reliably with As-Built close to ground level. For those distances simultaneous foresights and backsights are necessary in order to precisely eliminate the current refraction, which are not intended in As-Built.

7.2 Bluetooth Connection

To be able to work with a wireless Bluetooth connection you need to have an integrated Bluetooth chip set in your computer. Should your computer not have such a chip set it is possible use a Bluetooth adapter, in the form of a USB stick.

Preparation

The As-Built Bluetooth communication requires the use of **Windows own Bluetooth driver** (Microsoft Bluetooth Stack). Some Bluetooth devices however use by default their own driver, but to be able to work with As-Built it must be deinstalled. The following Bluetooth drivers are known to us and have concerns:

- Toshiba Bluetooth Stack (driver)
- Broadcom (Widcomm) Bluetooth Stack (driver)

---

[4] Can be set via the "As-Built – Settings" command. You will find further information in the respective section within the command reference.

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Please deinstall these drivers using the Windows Control Panel. Thereafter Windows must be restarted. The Bluetooth device should now be automatically assigned Windows own driver.

7.2.1   Setting up the connection under Windows XP

It is recommended when using Windows XP that the setting up of the Bluetooth connection is done using administrator privileges.

Ensure next that your Bluetooth hardware on your PC is activated. Should you be using an external Bluetooth stick, it should now be plugged in. Open the Windows Control Panel and select the configuration item 🅑 Bluetooth devices. In the dialogue box that opens click on `    Add    ` to start the "Assistant to add Bluetooth devices". Activate the option

> "Device is ready and can be identified".

Before you click on `    Next >    ` ,ensure that the hand held laser is switched on and that the Bluetooth mode is on. When using a Leica Disto press the blue button with the Bluetooth symbol. For all other instruments you will find the necessary information in the instrument manual.

The assistant now searches automatically for all Bluetooth devices within range and displays a list of them. Select your hand held laser from this list and double-click on its symbol.

It opens the window for the primary key. Select here

> "Use the Primary key from the documentation"

and enter four zeros in the field (**0000**). Click on `    Next >    `. The primary key will now be automatically replaced and the Bluetooth device installed. Now click on `    Finish    `.

7.2.2   Setting up the connection under Windows Vista / 7

Ensure next that your Bluetooth hardware on your PC is activated. Should you be using an external Bluetooth stick, it should now be plugged in.

Ensure that the hand held laser is switched on and that the Bluetooth mode is on. You will find out more on how to do this under the relevant chapter for the respective hand held laser models.

Open the Windows Control Panel and navigate to the configuration item:

> Control Panel ► Hardware and Sound ► Add a Bluetooth device
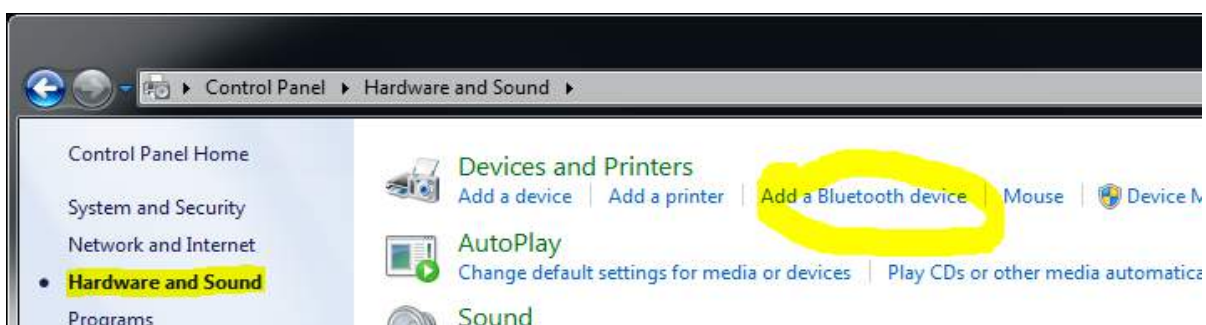
The assistant now searches automatically for all Bluetooth devices within range and displays a list of them. Select your hand held laser from this list and click on   next   .

It opens the window for entering the pairing code. Select here

"Enter the pairing code for the device"

and enter four zeros (**0000**) as the pairing code in the subsequent request. Click on   Next   . The Bluetooth device will now be installed. After the successful installation close the assistant by clicking on   Close   .

7.2.3   Setting up the connection under Windows 8 / 8.1 / 10

Ensure next that your Bluetooth hardware on your PC is activated. Should you be using an external Bluetooth stick, it should now be plugged in.

Ensure that the hand held laser is switched on and that the Bluetooth mode is on. You will find out more on how to do this under the relevant chapter for the respective hand held laser models.

Move your mouse pointer to the lower right of the screen. Thereupon the Windows Charm Bar appears on the right hand edge of the screen. Now navigate through the following configuration items:

with Windows 8

Settings ► Change PC settings ► Devices ► Add device

with Windows 8.1 / 10

Settings ► Change PC settings ► PC and devices ► Bluetooth

The assistant now searches automatically for all Bluetooth devices within range and displays a list of them.
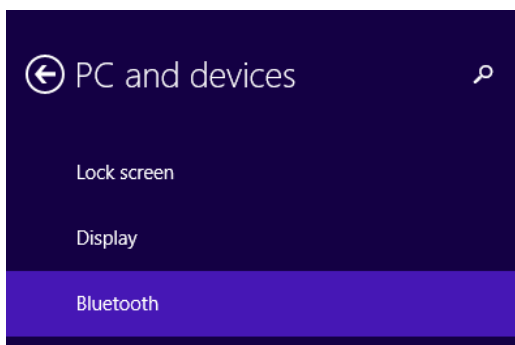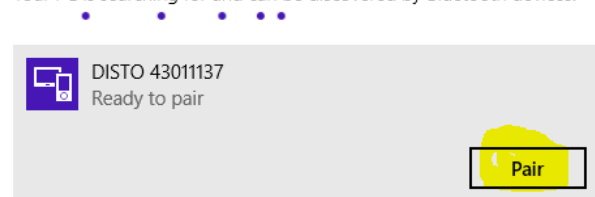
Select your hand held laser from this list and click on   Pair   .

Should you be asked for an identification enter four zeros (**0000**) as the pairing identification. Click on   Next   . The Bluetooth device will now be installed.

7.3 Communication between instrument and As-Built via a connection cable

Only total stations can be connected to the computer by a connection cable. Hand-held laser instruments do not have that option.

For the connection you need a connection cable (a) with one instrument specific and one serial port for the computer.

Many computers do not have a serial interface anymore though. In that case you will need a cable (b) with a USB plug for the computer. If the manufacturer has not provided such a port, you may interconnect a USB-serial adapter (c).

a)   b)   c)

a) instrument specific RS232 plug and serial plug (computer)
b) instrument specific RS232 plug and USB plug (computer)
c) USB – serial adapter

7.3.1 Installation

You do not need a driver for the serial connection cable (a). Just plug it into the serial port of your computer then read off the appropriate COM interface and enter it into As-Built as described in chapter "**Detecting the correct COM interface**".

If you use a USB cable (b) you will have to install the driver provided on the CD first.

If you use a serial connection cable (a) in combination with a USB-serial adapter (c), you will have to install the driver provided for the adapter first.

7.3.2 Connection

Connect the cable with the RS232 interface of the instrument and the USB or serial port of the computer.

If you are using a USB interface, it is advisable to always use the same USB port (e.g. the first one from the top). When connecting the cable, the computer will automatically assign a COM interface to the cable (e.g. COM4). This COM interface will stay the same when you always use the same USB port. You need to memorize this number and enter it in As-Built - Select instrument… (for further information please refer to chapter "**Detecting the correct COM interface**" below). When you insert the RS232-USB cable into another USB port, a new COM interface will be assigned to the cable.

7.3.3 Settings in As-Built

Open As-Built – Instrument measurement – Select instrument… and select your instrument from the list. Set all communication parameters according to the set parameters of your instrument.

You will find a detailed description regarding the configuration of the individual instrument types in the following paragraphs.

Enter the COM interface that has been assigned to the cable when connecting the instrument. Just as described in 1.2.2 **"Connection"** above

7.3.4 Detecting the correct COM interface

Windows XP

Open **Settings – Control Panel from the Windows menu and then select the sub entry System.**
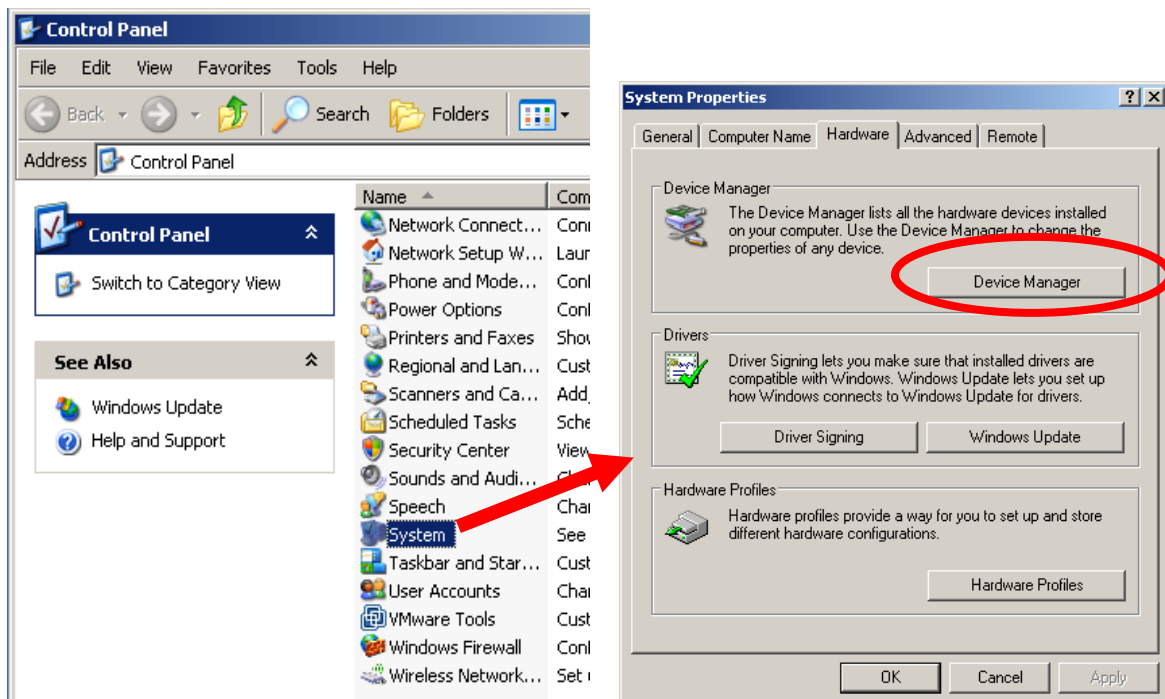
Windows 7 / 8:

From the Windows-Button Open **Control Panel – System und Security – System → Device Manager.** In Windows 8 you go to this point from the right hand side bar - System – Device Manager.

Windows 10:

Enter the words Device Manager into the search box on the taskbar, then select Device Manager from the list of results.

The following window will open.
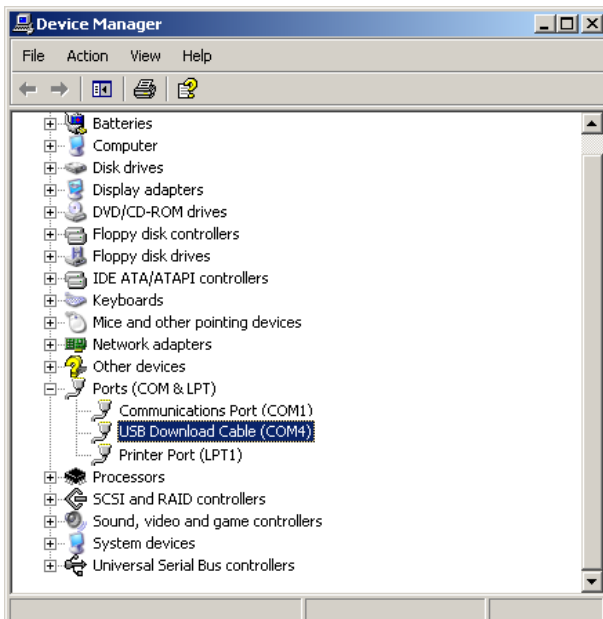
Select the Hardware tab and then press the **Device Manager** button. From the opening window select **Ports (COM & LPT). The port of your cable could be called USB Download Cable or USB to Serial Converter (according to the installed driver) and it has an assigned COM interface (COM 4 in this case).**

This COM interface has to be entered for the "COM Port" in the As-Built command→**Instrument measurement - Select instrument...**

7.3.5   Testing the communication

After you have accomplished all these settings, you are able to use As-Built with your instrument.

Perform a test measurement in order to check whether the communication between the instrument and As-Built works. In order to do so select **As-Built – Instrument settings – COM-Port monitoring**.

If the communication works correctly the results will be shown in the window below.

Instrument test terminal

21.322+38593000 22.322+09404400 31..00+00000712

Leica tachymeter, communication parameters: 9600,N,8,1, COM1, Handshake 0

Delete   Measure   OK
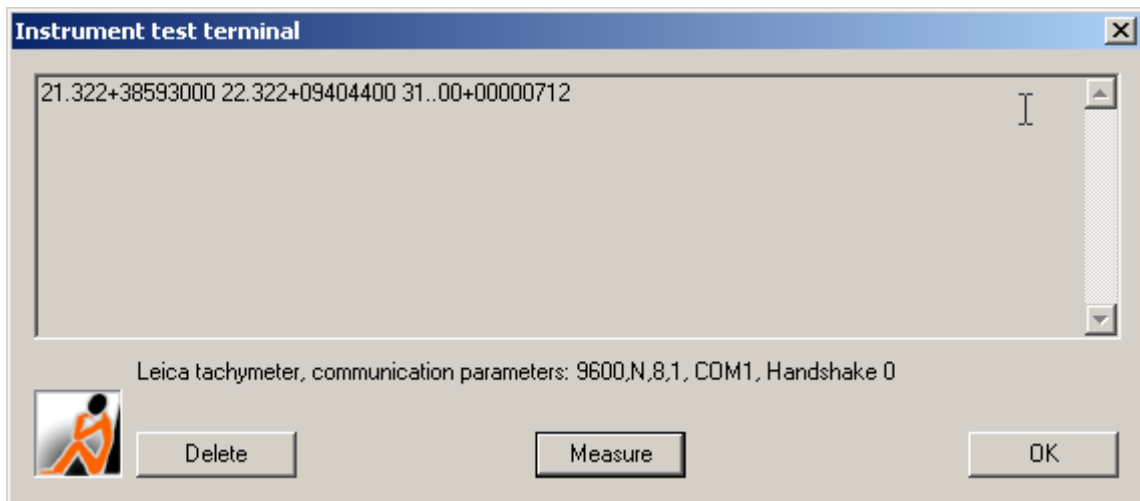
Hint: Don't worry, when your total station outputs a communication error! This message should only appear right here for the interface control, not during a real measurement.

7.4 Configuration of the individual tachymeter types

This section describes the settings, which have to be set at the tachymeter in order to be able to work with As-Built. The most common instrument series have been described. If your instrument series is not among those please try to work according to the description of a similar series or if that fails contact the FARO support. If you own a completely different instrument, please ask FARO if it is possible to realize the support of your instrument short term.

7.4.1 Leica FlexLine (TS02/TS06/TS09)

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The Flexline instruments are available in different firmware versions, that differ in their menu navigation. Here both of the firmware versions will be described, select the appropriate one for your instrument.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

Firmware Version 1 (newer instruments)

Main menu → 5 Settings

You arrive in the Settings menu. The following settings' descriptions are spread over several pages. Flip through the pages using the SCROLL button.

Settings → 3 Data → Page 2/2:

Data Output: Port

GSI Format: GSI 8 or GSI 16 (both work)

GSI mask: Mask1

Exit the menu option using "Cont" (F4)

Settings → 2 Regional → Page 1/3 and 2/3:

Hz Increment: Right

V-Setting: Zenith

Anglular Units.:   gon

Dist Unit: metre

> The units can also be set differently. You must just ensure that the same units are also used in the As-Built instrument settings.

Exit the menu option using "Cont" (F4)

Settings → 5 EDM

Here you set the desired distance measuring mode:

EDM Mode: NP = Non prism (reflectorless)

                          P = with prism

Prism Type: must be set if EDM mode = P

Exit the menu option using "Cont" (F4)

Settings → 6 Interface→ Page 1/2 and 2/2:

Port: Bluetooth → wireless connection

                          RS232 → serial cable with or without a USB adapter

                          USB → USB cable

Baud rate: 9600 (Bluetooth: …)

Data bits: 8

Parity: None

End mark: CR/LF

Stop bits: 1

Acknowledge: OFF

Hint: If using Port=Bluetooth, the other settings will be deactivated. If you need to change them, please switch Port to RS232. Now the parameter settings are activated. After your changes switch back to Port=Bluetooth.

Exit the menu option using "Cont" (F4)

Firmware Version 2 (older instruments)
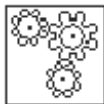
Main menu → 5 Settings" → you arrive in the Settings menu.

The following settings descriptions are spread over several pages. Flip through the pages using the SCROLL button.

Settings → "General" (General settings)

Page 1/5: These setting are not essential for working with As-Built. But by assigning special functionality to the USER keys (e.g. USER key 1 for switching the Laser pointer on and off) you may make your work more efficient. Besides this we recommend that you turn on the Tilt correction and the Hz collimation, since these correct any remaining inaccuracies in the instrument levelling.

Pages 2/5 to 4/5:

Hz Increment.:   Right

V-Setting: Zenith

Angle Unit.:   gon

Dist Unit: Metre

> The units can also be set differently. You must just ensure that the same units are also used in the As-Built instrument settings.

Data Output: Interface (do not select Internal Memory!)

GSI Format: GSI 8 or GSI 16 (both work)

GSI mask: Mask 1

Settings →"Comm" (communication parameters)

Port: RS232

Baud rate: 9600

Data bits: 8

Parity: None

End mark: CR/LF

Stop bits: 1

Settings →„EDM"

Here you set the desired distance measuring mode:

EDM mode: **NP-Standard** (Non Prism, reflectorless)

Laser pointer: **On**

Accept the automatically prompted values for the remaining settings.

Measuring

The procedure for making observations is the same for all Flexline instruments.

Switch to the Main menu (ESC button) and select the option "Q-Survey".

A full measurement (Distance and angles) is initialized by pressing the softkey ALL (F1).

With the softkeys DIST (F2) and REC (F3) it is possible to separate the distance measurement from sending the data. With DIST a distance measurement is started and with REC the data will be sent to As-Built.

For a fast switch between measurement with or without prism you can use the softkey EDM (F2). Use F4 for switching between different assignments for the function keys.

If you have further questions regarding your FlexLine total station please refer to the manufactures manual.

7.4.2 Leica Builder (R100M/R200M/RM Power/300/500)

The settings described here, have only to be done when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away. Please set the bold printed settings. If a menu item is not specified here then the setting is not important for As-Built.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Builder display
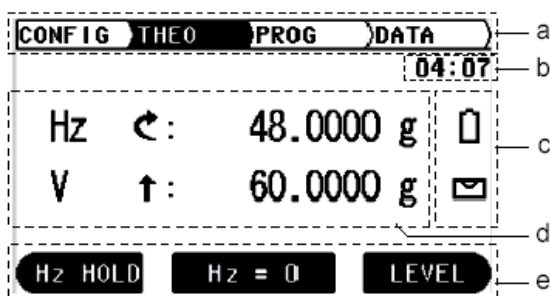
a) Tab bar
The active tab is black.
It shows the selected program mode

b) Time
shows the current time, if set

c) Icons
Battery, current capacity
compensator, active/inactive

d) Screen area

e) Softkeys
currently useable commands
allocation varies depending on program

Main menu

The main menu is located in the header of the display.

With the following button one can switch between the menu items. (tab switch)

Settings under CONFIG:

Laser Pointer: OFF / **ON** / OFF&TRCK / ON&TRCK

Hz Increment: **RIGHT** / LEFT

V-Setting: **ZENITH** / HORIZON/ V(%)

Compensator: **ON** / OFF   (automatic correction of transversal tilt errors)

Measure&Record: MEAS&REC / **ALL-IN-1**

Use the cursor keys ⇧ and ⇩ for navigation and this cursor ⇨ for editing/changing the settings. Afterwards confirm your selection with **OK**.

Continue with further settings in the sub menus of CONFIG:

Sub menu CONFIG/DISPLAY:
- Use the soft-key DISPLAY (by pressing the yellow key below)
- Settings:

- Angle unit: ° ' " / DEC.DEG / **GRD** / MIL   (select angle unit)
- Minimum Reading: PRECISE / **STANDARD** / SIMPLE
- Distance Unit: **METER** / FT-IN(1/16) / US-FT / INT-FEET (select distance unit)

- Confirm with soft-key **OK**

sub menuCOMM. (Builder R200/300/500)

- Select Softkey COMM. (by pressing the yellow key below)
- Settings:
  - Data output: **RS232** / INT MEM / USB / **Bluetooth**
  - Baudrate: 2400/ 4800 / **9600** / 19200
  - Databits: 7/**8**   (Number of data bits for transfer)
  - Parity: **NONE** / EVENT / ODD
  - End mark: CR/**CRLF**
  - Stop bits: **1**
- Confirm with OK

Settings under DATA, sub menu DATA/RS232(Builder R100)

- Select soft-key RS232 (by pressing the yellow key below)
- Settings:
  - Data output: **RS232** / INT MEM
  - Baud rate: 2400/ 4800 / **9600** / 19200
  - Data bits: 7/**8**   (Number of data bits for transfer)
  - Parity: **NONE** / EVENT / ODD
  - End mark: CR/**CRLF**
  - Stop bits: **1**

  In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.
- Confirm with **OK**

Checking the program status

To be able to measure with the Builder the application "Angle & Distance" has to be loaded. The current application is shown in the first line of the screen area. If this is not the case press the **APPL**. key, navigate to "Angle & Distance" and confirm with **OK**.

Start a measurement

In the program "Angle & Distance" one has to press the yellow key below [M&R] to measure and transfer the data to As-Built.

EDM settings for the Leica Builder RM Power

Since you can also execute infra red measurements with this instrument, you will have to check before the measurement, if the settings for the measurement without reflector are correct. In order to do so press the EDM button of the instrument. You may now choose between different settings: the measurement without reflector (red point) or with reflector (fine/fast). The LED of the keypad will show the selected measurement type.

When you have chosen the measurement without reflector there will be no display or options for the prism type or the prism constant. The reflector height should be set to "0".

7.4.3 Leica TCR 110C

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Turn the instrument on with the button ↑(**ON)**.

Press ↑ (Shift) and **MENU** to get into the main menu. Select the menu item 'SETTINGS'. The bottom line of the display shows you how the buttons for the navigation arrows (▾) and the [OK] function are assigned.

The sub menu SYSTEM SETTINGS has three display pages. You can switch between the pages with the button $\boxed{_1{}^2}$. Go through the individual points with the cursor buttons (▾ ▸) and set the following values. The values are changed with the button ▸ .

System settings:

- EDM Modus: RL-Strd (Reflectorless measurement with laser)
- Prism type: RL (no prism is used)
- Prism const.: 34 mm (is automatically set)
- Tilt Corr.: On (automatic tilt compensation)
- angle Gon (Angle unit gon)
- Distance Meter (unit distance measurement)

Finally press the **OK** button to save the new values. You are automatically taken back to the measurement display.

Go to the main menu again with ↑ and **MENU** and select the entry COMM. PARAMETERS. Set the following values with the same method:

COMM.PARAMETERs

- Data output: RS232 (Saves on serial interface)
- GSI Format: 8 (Data output format)
- GSI mask: Mask 1 (Data output format)
- Baudrate: 9600
- Databits: 8
- Parity: none
- End mark: CR/LF
- Stop bits: 1

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

Finally press the **OK** button to save the new values. You are automatically taken back to the measurement display.

A few notes concerning the measurement process itself:

A measurement is released with the button **DIST**.

You can turn the red laser dot on and off with the buttons ⬆ and **LASER** (DIST).

The button ▭ calls up the electronic bubble.

7.4.4 Leica TPS 300 (TCR 303 / 305 / 307) and TPS 700 (702 / 703 / 705)

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Turn the instrument on. First click the button '**SHIFT**' and then '**PROG**' (MENU) to get into the menu display. Select 'ALL SETTINGS' with the cursor keys and confirm with the red Enter button.

SYSTEM SETTINGS:

- Data Output: RS232   (Save on serial interface)
- USER Key: LASER   (Turns the red laser dot on and off with the button '**USER**')
- Trigger key: ALL   (Trigger key with the same function as the ALL key release of a measurement)
- GSI Format: 8   (Data output format)

UNIT SETTINGS:

- angle: gon
- distance: Metre

EDM SETTINGS:

- Dist Mode: RL-Short   (reflectorless, detailed measurement)
- Prism Type: RL   (is automatically set depending on the dist mode)
- Prism Const: 34 mm   (Prism constant, is automatically set depending on the prism type)

COMMUNICATION:

- Baud rate: 9600
- Data bits: 8
- Parity: none
- End mark: CR/LF
- Stop bits: 1

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

The keys '**SHIFT**' and **PgDn** (arrow down) or **PgUp** (arrow up) allow you to move a page forward or backward.

7.4.5 Leica TPS400 (TCR 403 / 405 / 407) and TPS 800 (e.g. TCR 805)

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Turn the instrument on and press the button '**MENU**' on the upper right to get into the menu display. Select the desired menu item with '**F1**' to '**F4**'.

Go through the individual items with the cursor buttons (∀∀) and set the following values. The values are changed with the cursor buttons (◁▷) The settings under each menu item extend over several pages. The button '**F2**' (CONTINUE) allows you to go to the next page. Finally, click '**F4**' (SET) to save the new values; you are automatically taken back to the main menu.

F2 SETTINGS:

- USER key: LASER   (Turn the laser point on and off with the button '**USER**')
- Data output: RS232   (Saves on serial interface)
- GSI Format: 8   (Data output format)
- GSI Mask: Mask 1   (Data output format)

F3 EDM Settings:

- EDM Mode: RL-Short   (Reflectorless measurement with Laser)
- Prism type: RL   (no prism is used)
- Prism const.:   34.4   (is automatically set)
- Laser pointer: on   (red laser dot is visible)

The main menu has two pages. The button '**PAGE**' allows you to switch between page 1 and 2.

F2 COMM parameters:

- Baud rate: 9600
- Data bits: 8
- Parity: none
- End mark: CR/LF
- Stop bits: 1

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

7.4.6   Leica TPS1100 series (e.g. TCR 1103, TCR(A) 1105)

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Use the cursor keys (∇∆) to navigate through the items and make the following settings. For selecting a certain setting you use these cursor keys (◁▷).

1. First of all switch off the instrument's remote control:

Main menu → Configuration → Communication → RCS On/Off → No

2. Set communication parameters:

Main menu → Configuration → Communication → GSI parameters:

- Baudrate: 9600
- Protocol: NONE
- Parity: none
- End mark: CR/LF
- Data bits: 8
- Stop bits: 1
  → continue

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

3. Switch off the GeoCOM mode:

Main menu → Configuration → Communication →GeoCom On/Off → No

4. Switch on the registration by RS232 interface:

FNC→ Job settings   → set measurement job to RS232
                         → set S-Mask to Polar(8)

5. Set the proper registration mask:
Main menu → Configuration → Instrument Config. → Display/Reg.mask → REGMASK (F3) →

1.Word   Hz
2.Word: V
3.Word: SlopeDist

6. Select the units for the measurements:

Main menu → Configuration → Instrument Config.→ Units:

- AngleUnit: 400gon
- AngleDec: 4 Decimals
- Dist.Unit.: Meters [m]
- Dist.Dec: 3 Decimals
- V-Display: Zenith

7.4.7   Leica TPS1200 series, Viva TS 30

Requirement: For unlimited operations with As-Built the "GeoCOM Robotics" module has to be activated within the instrument. Please contact your Leica distributor for the license activation. Without the license there will be the restriction that measurements can only be initiated from the instrument but not from afar within As-Built.

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The following settings are necessary for the communication with As-Built via a serial COM port:

Go to the main menu and select item 5 (Config…). Then select the item interfaces (4) In this menu you have to activate the GSI output and the GeoCOM mode. You must also deactivate the remote control RCS.

Activate GSI output:

Go to "**GSI output**" and press **Edit** (**F3)for Edit**. Adjust following settings:

Use interface:	yes
Port:	Port 1 (data transfer by cable)
	Port 3 (data transfer by integrated Bluetooth)
Protocol:	none
GSI format:	GSI 8 Polar & Kart or GSI 16 polar

Then go to **Instrument settings** with **F5** and select the predefined instrument "**RS232 GeoCom**" or define a new one (e.g. "As-Built"). Following communication parameters should be set at the instrument:

Baud Rate:	19200
Data bits:	8
Parity:	none
Stop bit:	1

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

You can check these settings by pressing **Edit (F3)** which takes you to the instrument configuration settings where you can change deviations.

Press **Continue** (**F1)twice to save these settings and go back to the previous menu**.

Activate GeoCOM mode:

Go to "**GeoCOM Mode**" and press **Edit** (**F3) for Edit.** Adjust following settings:

Use interface:	yes
Port:	Port 1 (data transfer by cable)
	Port 3 (data transfer by integrated Bluetooth)
Protocol:	RS232 GeoCOM

Then go to **Instrument settings** with **F5** and select the predefined instrument "**RS232 GeoCOM**". It has to be the same instrument you have configured in GSI output.

Deactivate Remote Control RCS:

Go to "**RCS mode**" and press **Edit (F3). A**djust following settings:

Use interface: no

Set the units

In the main menu switch to „**Config**" (5), go further to the item **"General settings"** (3) and „**Units & Formats**" (3). Select the tab „**Units**" and check the following settings:

- Distance unit
- Angle unit

It is up to you which units you want to use. You just have to make sure that you choose the same units in the As-Built instrument settings.

Set the measuring mode

To switch between reflectorless and prism mode go to the main menu and open the item **Config...** (5) –I**nstrument settings** (2) –**EDM and ATR Settings** (1).

Measuring

For the measuring process go to the main menu and open the item "**Measure**".

A full measurement (Distance and angles) is initialized by pressing the key ALL (F1).

With the keys DIST and REC it is possible to separate the distance measurement from sending the data. With DIST a distance measurement is started and with REC the data will be sent to As-Built.

Hint: If you use the integrated Bluetooth-interface it might only be possible to start a measurement from the instrument. According to the used Bluetooth driver of the notebook the instrument will not show any reaction when starting a measurement within As-Built.

Motorized stake out

If you apply a robotic total station it is possible to utilize the motorization for the As-Built stake out function. The instrument will move toward the point to be staked out automatically. A detailed description of this feature you may find in the command reference.

Use as one-man station

If the motorized total station has functions for the prism search and for the prism tracking (ATR, PowerSearch), it may also be used as automated one-man station.

When doing so the person is not standing at the tachymeter but walking from measurement point to measurement point with the reflector (reflector pole). The instrument tracks the prism automatically and aims at it. By pushing a button within As-Built the measurement will be initiated.

This mode of operation is advisable for measurement projects where topographic objects have to be measured outdoors. It is not suited for reflectorless measurements.

An appropriate notebook (suitable for the outdoors, small and light) as well as a stable wireless connection between the tachymeter and the notebook are required for the above operation.

Additionally to the conventional As-Built measurement functions a number of special commands for the use as one-man-station are available. You will find further information on this topic within the command reference.

- ➔ Start / Stop Prism search (left/right)
- ➔ Enable / Disable prism tracking
- ➔ Battery state of the instrument
- ➔ Change face

7.4.8 Leica Viva TS15

Requirements:

For the use with As-Built you need to have the firmware version≥ 3.01. installed on your instrument. Please check the version from the main menu under **User**(4) →**System properties** (5) → Tab **SmartWorx Viva**. You will find the version number under **SmartWorx Viva.**

Furthermore the "GeoCOM" module has to be activated in the instrument. You can check the activation from the main menu under **General** (4) →**System properties** (5) → Tab **Total station(TS)** and then under the option **GeoCOM Robotics**. Please contact your Leica distributor for a firmware update or license activation. Without the license there will be the restriction that measurements can only be initiated from the instrument but not from afar within As-Built.

The described settings for the work with As-Built have only to be set when you use it for the first time. They will be saved in the instrument and the next time you turn on the instrument, you can start with the measurement right away.

The units for the angles and distances may differ from the provided. But you have to make sure that you use the same units within the As-Built instrument settings.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The following settings have to be set for the communication with As-Built via the serial interface:

Activate GSI output:

Within the main menu change to **Instrument** (3) →**Connect to instrument** (3) →**All other connections** (3). Select **GSI output** and press **EDIT** (F3). Perform the following settings:
- [x] Output GSI data to device
- Connect using: Cable (data transfer via cable)
  TS Bluetooth 1 (data transfer via Bluetooth)
  RadioHandle (data transfer via Leica wireless radio)

Select **Devices** (F5) in order to get to the additional device settings. Select the connection mode you have chosen before and press **Edit** (F3). Set the communication parameters as follows and **Save** (F1) them:
- Baud rate: 19200 (for Radio Handle use 115200)
- Parity: None
- Data Bits: 8
- Stop Bit: 1
- Flow Control: None

By pushing **Store** (F1) and **OK** you will get back to the GSI output settings. Please make sure, that the communication parameters of the instrument and within the instrument settings of As-Built are the same.

Activate GeoCOM mode:

Within the main menu change to **Instrument** (3) →**Connect to instrument** (3) →**All other connections** (3). Select **GeoCOM**and press **Edit** (F3). Perform the following settings:

- ☑ Allow GeoCOM communication
- Connect using: Please select the same device port that you have selected for the GSI output!

Set measurement mode:

You may set a fast switch between reflector measurement and reflectorless measurement within the main menu in the upper symbol bar under the target surface symbol. Select your prism type or "Reflectorless" in order to measure with the red laser. According to the selected option the symbol changes.

You may alternatively use the **Favorite button**★→**Measure any surface / Measure to prism** (1).

Set units

From the main menu select the option **User** (4) →**System settings** (3) →**Regional settings** (1).

You can check the settings for the distance in the **Distance** tab. Now change to the **Angle** tab. Please also check the unit settings . Additionally the following values have to be set:

- Reference direction: North azimuth
- V angle display: Zenith angle

Hint: You may decide yourself which units to use. But you have to make sure that you use the same units within the As-Built instrument settings.

Measuring

In order to measure change to the main menu and select the point **Go to work**(1) →**Survey**(2). A full measurement will be initiated by pushing the **Meas**(F1) button.

With the buttons **Dist**(F2) and **Store**(F3) it is possible to have the distance measurement and the sending of the data run separately. At first you measure the distance with **Dist**(F2), then the measurement data will be sent to As-Built with **Store**(F3).

Motorized stake out

If you use a total station with motorization, you may use that for the As-Built function →**"Stakeout point"**. The point to be staked out may be approached automatically. You will find further information regarding the above within the command reference under →**"Stake out point"**.

Use as one-man station

If the motorized total station has functions for the prism search and for the prism tracking (ATR, PowerSearch), it may also be used as automated one-man station.

When doing so the person is not standing at the tachymeter but walking from measurement point to measurement point with the reflector (reflector pole). The instrument tracks the prism automatically and aims at it. By pushing a button within As-Built the measurement will be initiated.

This mode of operation is advisable for measurement projects where topographic objects have to be measured outdoors. It is not suited for reflectorless measurements.

An appropriate notebook (suitable for the outdoors, small and light) as well as a stable wireless connection between the tachymeter and the notebook are required for the above operation.

Additionally to the conventional As-Built measurement functions a number of special commands for the use as one-man-station are available. You will find further information on this topic within the command reference.

➔ Start / Stop Prism search (left/right)

➔ Enable / Disable prism tracking

➔ Battery state of the instrument

➔ Change face

7.4.9   Leica Viva TS16

Requirements:

For the use with As-Built the "GeoCOM" module has to be activated in the instrument. You can check the activation from the main menu under

Leica Captivate – Home – Settings – About Leica Captivate Tab Total station

and then under the option **GeoCOM Robotics**.

Please also check the Option P**owerSearch**. If this is not activated, please go to As-Built settings and switch this value: **As-Built – Instrument – Leica prism search mode = "2"** Default is "1", which means "PowerSearch", "2" stands for Simple Search.

Please contact your Leica distributor for a firmware update or license activation. Without the GeoCOM Robotics license there will be the restriction that measurements can only be initiated from the instrument but not from afar within As-Built.

The described settings for the work with As-Built have only to be set when you use it for the first time. They will be saved in the instrument and the next time you turn on the instrument, you can start with the measurement right away.

The units for the angles and distances may differ from the provided. But you have to make sure that you use the same units within the As-Built instrument settings.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The following settings have to be set for the communication with As-Built via the serial interface:

Activate GSI output:

Within the Leica Captivate main menu change to **Settings → Connections**(1) **→All other connections**(3). Select **GSI output** and press **EDIT** (F3). Perform the following settings:
- ☑ Output GSI data to device
- Connect using: Cable (data transfer via cable)
                TS Bluetooth 1 (data transfer via Bluetooth)
                RadioHandle (data transfer via Leica wireless radio)

Select **Device** (F5) in order to get to the additional device settings. Select the connection mode you have chosen before and press **Edit** (F3). Set the communication parameters as follows and **Save** (F1) them:
- Baud rate: 19200 (for Radio Handle use 115200)
- Parity: None
- Data Bits: 8
- Stop Bit: 1
- Flow Control: None

By pushing **Store** (F1) and **OK** you will get back to the GSI output settings. Please make sure, that the communication parameters of the instrument and within the instrument settings of As-Built are the same.

Activate GeoCOM mode:

Within the Leica Captivate main menu change to **Settings → Connections**(1) **→All other connections**(3). Select **GeoCOM** and press **Edit** (F3). Perform the following settings:
- ☑ Allow GeoCOM communication
- Connect using: Please select the same device port that you have selected for the GSI output!

Press **OK** (F1) to get back to GeoCOM output settings.

Set measurement mode:

You may set a fast switch between reflector measurement and reflectorless measurement within the main menu in the upper symbol bar under the target surface symbol. Select your prism type or "Reflectorless" in order to measure with the red laser. According to the selected option the symbol changes.

Set units

From the main menu select the option **Settings** (4) →**System** (5) →**Regional** (3).

You can check the settings for the distance in the **Distance** tab. Now change to the **Angle** tab. Please also check the unit settings . Additionally the following values have to be set:
- HZ angle display: North azimuth
- V angle display: Zenith angle

Hint: You may decide yourself which units to use. But you have to make sure that you use the same units within the As-Built instrument settings.

Measuring

In order to measure change to the main menu and select the point **Measure**. A full measurement will be initiated by pushing the **Meas**(F1) button.

With the buttons **Dist**(F2) and **Store**(F3) it is possible to have the distance measurement and the sending of the data run separately. At first you measure the distance with **Dist**(F2), then the measurement data will be sent to As-Built with **Store**(F3).
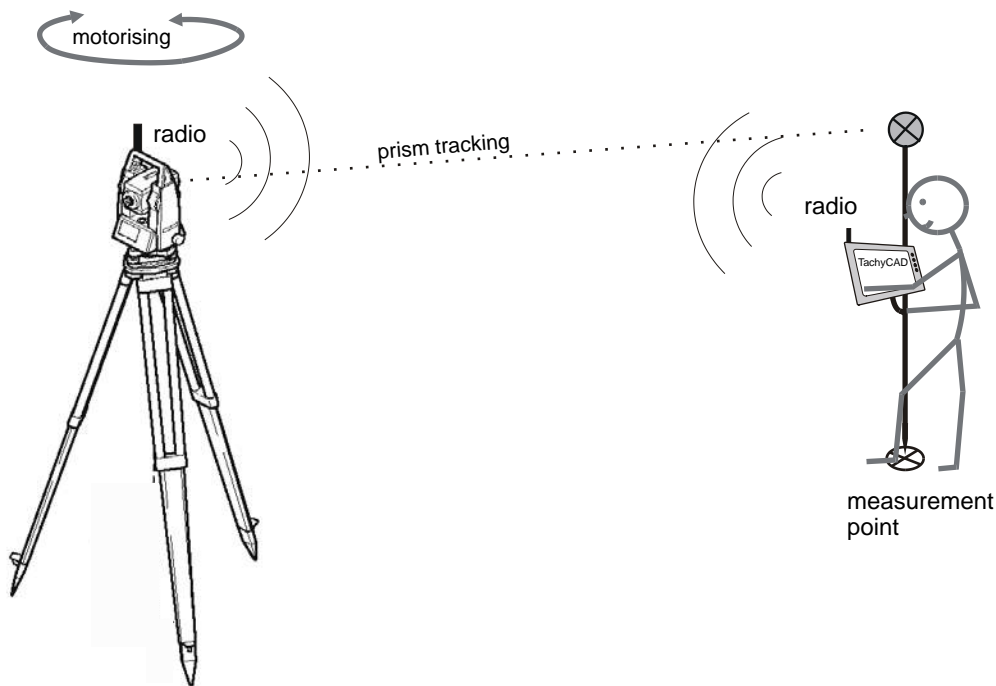
Motorized stake out

If you use a total station with motorization, you may use that for the As-Built function →**"Stakeout point"**. The point to be staked out may be approached automatically. You will find further information regarding the above within the command reference under →**"Stake out point"**.

Use as one-man station

If the motorized total station has functions for the prism search and for the prism tracking (ATR, PowerSearch), it may also be used as automated one-man station.

When doing so the person is not standing at the tachymeter but walking from measurement point to measurement point with the reflector (reflector pole). The instrument tracks the prism automatically and aims at it. By pushing a button within As-Built the measurement will be initiated.

This mode of operation is advisable for measurement projects where topographic objects have to be measured outdoors. It is not suited for reflectorless measurements.

An appropriate notebook (suitable for the outdoors, small and light) as well as a stable wireless connection between the tachymeter and the notebook are required for the above operation.

Additionally to the conventional As-Built measurement functions a number of special commands for the use as one-man-station are available. You will find further information on this topic within the command reference.

→ Start / Stop Prism search (left/right)

→ Enable / Disable prism tracking

→ Battery state of the instrument

→ Change face

7.4.10 Leica Nova Series

These instruments are identical in their technical control with the **Leica Viva TS15**. Please read the section → **Leica Viva TS15**

7.4.11   Leica 3D Disto

Requirements:

For the use with As-Built you will need:

- As-Built, starting from version 9.0.10.0

Installation and activation of the Leica 3D Disto controller software for windows (version 1.1.0.6 or higher). Without the "3D Disto for windows tablet"-license the communication with the instrument will not be possible. Please contact your dealer in order to purchase the licence for FARO 3D Software GmbH or your Leica 3D Disto dealer. The Setup for die control software can be found on http://service.faro3dsoftware.com/supportfiles/.

Technical implementation:

The communication between notebook and measurement device is assumed by the 3D Disto controller software. The connection may either be established via WLAN (wireless) or the provided USB cable.

As-Built does not communicate directly with the measurement device but it establishes a connection to the 3D Disto controller software. From the software, As-Built gets the measurement values and processes them.
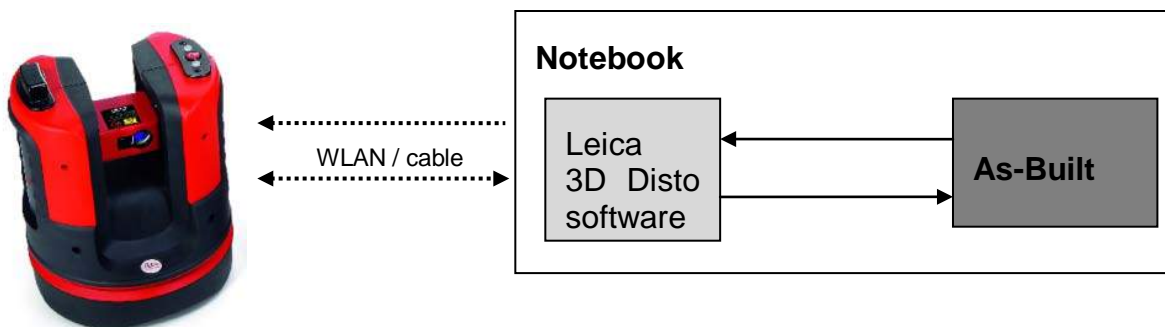
Setting up the system

You may either connect to the instrument via

- cable or
- WLAN

    Important: When you use the 3D Disto for the first time with your computer you must make the first connection with a cable. Should you switch between several 3D Disto instruments the first connection after switching instruments must be made via a cable

1) Establish connection to the measurement device

    a. **Cable**: Plug in the cable at the measurement device and the laptop. Switch ON the measurement device.

    b. **WLAN**: Switch ON the measurement device. Activate WLAN on the notebook. Set up and establish the connection within the WLAN software ("Wireless network connection").

Important: To connect the 3D Disto via WLAN you have to manually change the TCP/IP address of the WLAN device to the fixed value **192.168.87.82** .
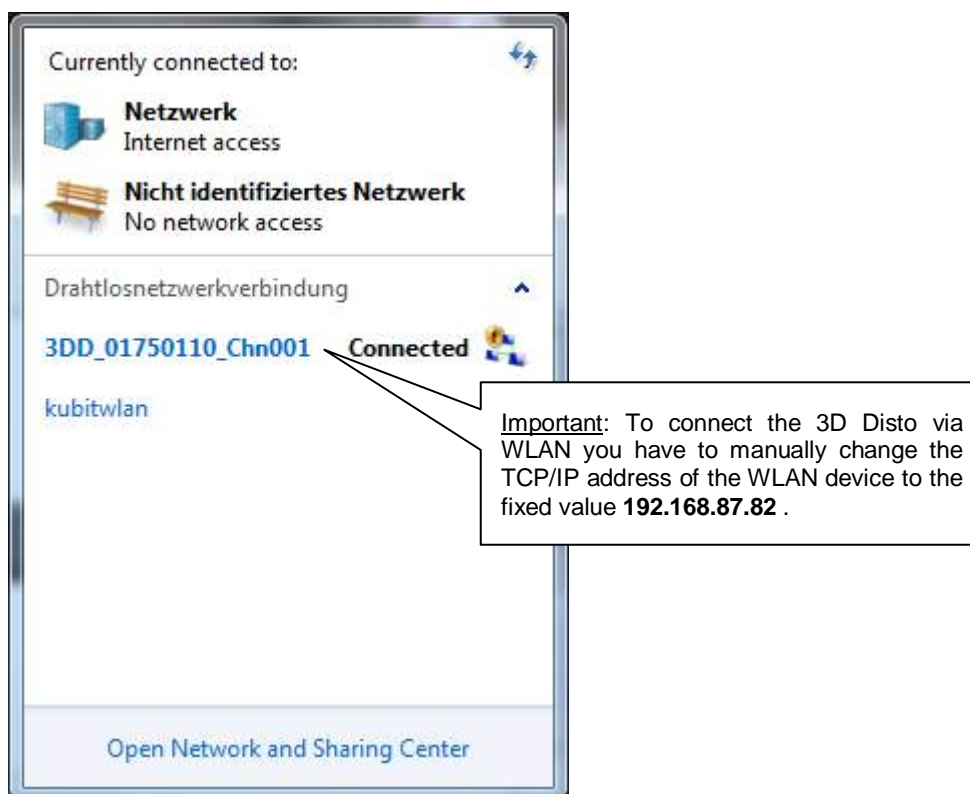
2) Starting 3D Disto software

Click here for the cable connection.

Click here for WLAN.

When the connection to the instrument has been established successfully confirm the appearing hint with OK. If you encounter any problems connecting the Leica 3D Disto device to the Leica 3D Disto control software, please contact Leica Support.

3) Starting AutoCAD with As-Built and execute the following settings:

- Start the As-Built command "Select instrument". Select "Leica 3D Disto" as instrument. The following parameters are valid for the communication with the 3D Disto controller software via the serial interface:

    - Connection: COM 70

      (This connection number will be predetermined by Leica by default and should work in most cases. A different number will automatically be used, if the above-mentioned has been assigned otherwise by the operating system. The correct number will be displayed within a dialogue window at the end of the installation process of the Leica 3D Disto software.

      Afterwards it can only be retrieved within the somewhat hidden configuration file of the

      Virtual COM port for GSI client: COM70

      OK
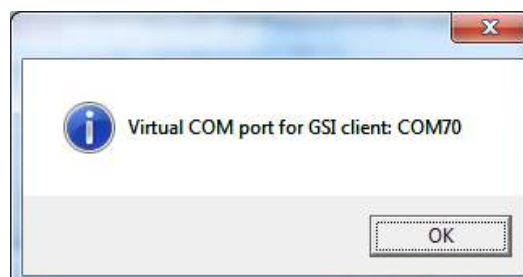
software. Please contact FARO in case this should become necessary.)

- Baud rate: 115200

- Data bits: 8

- Stop bits: 1

- Parity: none

- Flow control: none

- The **distance and angle units** within As-Built have to match the set units within the 3D Disto controller software. Within the 3D Disto controller software you may change the units within the **Menu** under **Settings→Units**.

- As soon as the instrument settings have been configured completely, As-Built connects to the controller software. The search window and the cross hairs will then be visible within the software.

The 3D Disto controller software

The 3D Disto has a camera which provides a search-window picture. This search window is used as pointing help when approaching the measurement points.
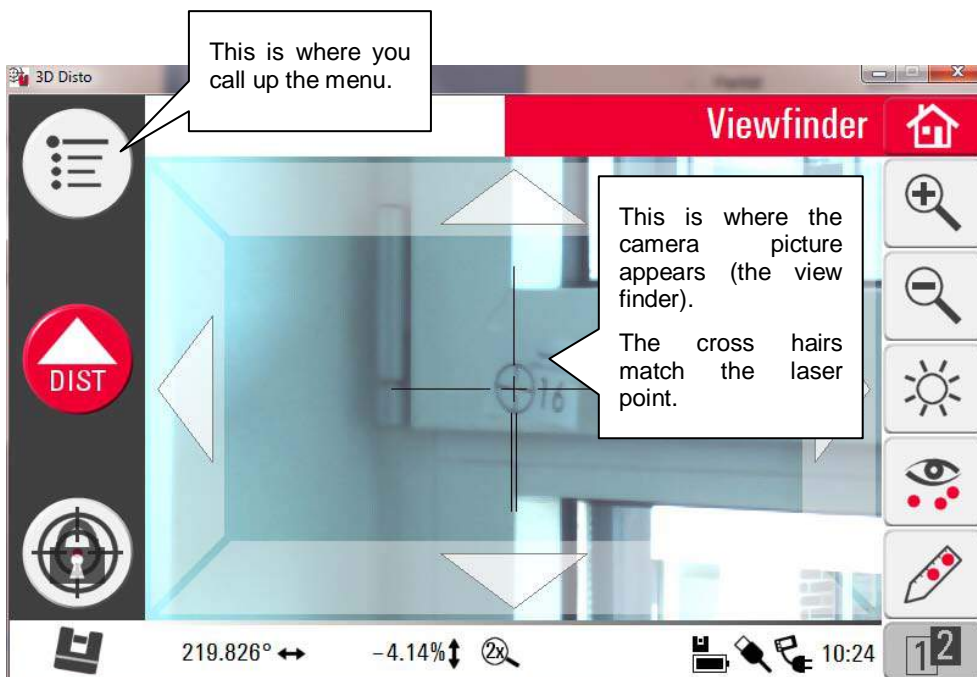
With the help of motorization it is possible, to operate the device by remote control. The measurement device does not have to be turned manually towards the target point.

The controller software provides according functions in order to move towards the target point:

- **Rough aiming**: Click on the middle of the cross hairs, hold the key down and aim into the wanted turning direction. A red arrow will appear. The 3D Disto will follow this movement. Small arrow means slow movement, large arrow means fast movement. That way you can navigate fast towards the wider target area.
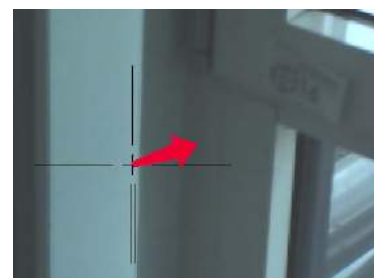
- **Precision aiming**: As soon as the target point is visible within the camera picture you can click on it with the mouse. The laser point of the 3D Disto (and therefore the cross hairs within the picture as well) will immediately move to that point. Use the zoom key in order to see the target better.
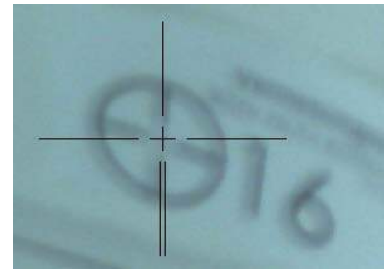
- **Alternative:** You may also use the direction arrows at the border of the window for the target search. But this is not very comfortable for bigger movements.

- If you cannot localize the point well enough at the screen, it is possible to walk towards the target point. You don't have to take the laptop in order to do so. Use the provided remote control for the navigation and look at the laser point.

Explanation of the keys:

	Menu
	Trigger measurement
	Point finder on / on+lock / off
	Back / Close project
	Camera zoom + / -
	Camera brightness
	Visualize measured points within the camera picture
	Measure offset points (channel ruler)

	Switch tool bar – further functions
	Take picture
	Negate picture for a better view of the edges
	Several turning options

Measuring:

A measurement may be triggered in three different ways:

- **3D Disto controller software**: Press the **DIST** key
- **As-Built/AutoCAD**: Press the **Star key** within the measurement box.
- **Remote control**: Press the **red button**.

Motorized stake out

You may use the motorized control of the 3D Disto with the As-Built function
➔"**Stake out**". In order to do so you may select a point from your AutoCAD drawing and have it pointed at with the laser point automatically. You will find further information in the command reference under ➔ "**Stake out** ".

7.4.12 Other Leica instruments

These settings are valid for all current as well as for older Leica total stations, Leica theodolites and Leica theodolites with mounted Disto. Among these are for example instruments of the TPS 1100, TPS 1000-2000, T1600/1610 and TC 1600/1610 series. Instruments of the TPS100/TPS300/TPS400/TPS700 series were explained in separate paragraphs.

Whether you work with a total station or an older theodolite with mounted DATADISTO, the standard settings for the communication via the RS232 interface are defined by following parameters:

- Baud rate: 9600
- Data bits: 8
- Parity: none
- Stop bits: 1
- End mark: CRLF
- Protocol: GSI

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

After setting the above parameters, you might have to configure a recording mask (REC mask):

- WI21   Horizontal angle
- WI22   Vertical angle
- WI31   Slope distance

Newer instruments allow raising the GSI data record length from 8 to 16. As-Built now also supports GSI 16 format.

As-Built always uses uncorrected measurement values. Thus, make sure to turn off all correction functions of instruments with mounted DATADISTO. The correction of the EDM offset is run directly in As-Built with the selection of the instrument type.

All changes are saved, even after the instrument has been turned off.

7.4.13   Instruments of type Geomax Zoom35 pro

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

The units for angles and distances are allowed to differ from those proposed here. But you have to make sure that you use the same units within the As-Built instrument settings.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

Total station settings

Please take the following setting. All other values are not relevant for As-Built and may be kept with their default values.

Open the **main menu** by pressing ESC several times.

By choosing the menu point **3 – Settings** you enter the settings menu.

The settings described afterwards are splitted on several pages. Press the page key to change between pages.

1 – General

page 1/4:   - Hz Inc.: Right
             - V Display: Zenith
             - Auto-Off: deactivated

page 2/4:   - Angle Unit: GON
             - Angle Dec.: 0,1 mgon
             - Dist. Unit: Metre
             - Dist. Dec: 3

page 3/4:   - Data output: RS232 interface
             - GSI Format: GSI 8 or GSI 16

page 4/4:   - GSI Mask: Mask 1

2 – EDM
- EDM Mode: NP-Standard (non prism) / prism (select Type)
- EDM Type :different types are available depending on the selected EDM mode

3 – COMM
- Port: Bluetooth (Bluetooth ON) → for wireless communication

             RS232 → for communication via cable

The following settings are only available for Port = RS232. If you want to work with Bluetooth, please temporary switch to port=RS232, then take the settings and after that switch back to port=Bluetooth.

- Baudrate: 9600 … 115200
- Data bits: 8
- Parity: None
- End mark: CR/LF

Settings on the computer

Set the As-Built Instrument selection to the "Leica total station (default) " profile (→ **Total station – Select instrument**).

Measurement

Please go back to the main menu by pressing ESC several times.

Choose the point "**4 – Measure**"

A full measurement (Distance and angles) is initialized by pressing the soft key **ALL** (F1).

With the soft keys **DIST** (F2) and **REC** (F3) it is possible to separate the distance measurement from sending the data. With DIST a distance measurement is started and with REC the data will be sent to As-Built.

If you have further questions regarding your total station please refer to the manufactures manual.

7.4.14 Instruments of type Geomax Zoom 80

On the computer

Set the As-Built Instrument selection **(→ Select instrument**) to the profile "**Leica total station (System 1200 + Viva TPS)**".

On the Total station

The settings on the instrument are nearly identical to Leica TPS1200 instruments. Please proceed as described in Chapter → **7.4.7 Leica TPS1200 series, Viva TS 30.**

7.4.15　Instruments of type Geomax Zipp 20

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

On the Total station

The Zipp20 workes with the operations system Windows CE. By default several apps are installed on the instrument. For the work with As-Built you need the app "Zipp20".

Therefore, please close all other running apps. Most likely other apps opened automatically by starting the instrument. (X-Pad) .

Open the app „Zipp20"

You will come to the main menu:
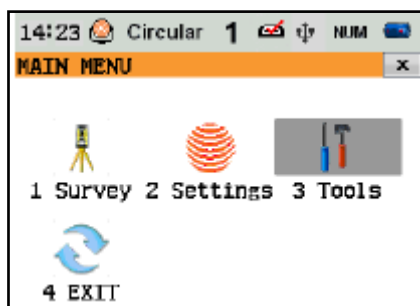
Settings

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

All other values are not relevant for As-Built and may be kept with their default values.

- 2 Settings – 4 EDM – 　Mode: RL-Default (reflectorless)

　　　　　　　　　　　　　　　　or IR-Default (with prism)

　　　　　　　　　　　　Laser Beaml:On

- 2 Settings – 5 Comm:　Port:　　　　Bluetooth (only Bluetooth is supported)

　　　　　　　　　　　　Bluetooth:　On

　　　　　　　　　　　　PinCode:　　Note down for connecting with your computer

Measure

Select „Survey" from the main menu. The Survey page opens:
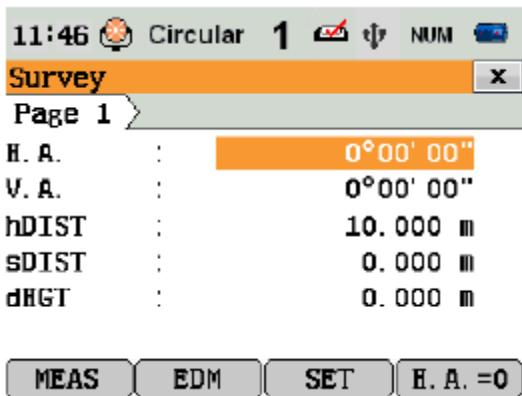
In the upper status line you can check the current states of the instrument (battery, telescope position, Bluetooth, tilt sensor, EDM mode). By tipping on the icons you can open the appropriate settings dialogues.

In the main part of the page you find the current measurement values so as angles and distance. Leave this dialogue open during your work. You don't have to do anything more on the total station.

On the computer

Settings

Open the Bluetooth environment on your computer. Search for new devices and peer with the Zipp20. You will need the PIN code that you noted down from the instrument.

Open AutoCAD with As-Built. Start the command „Select instrument" and set to „ „Geomax Zipp Serie". In the next tep you will be asked for the serial communication parameters:

- Serial port: Set to COM number, that will be shown in Bluetooth environment for the Zipp 20.
- Data bits: 8
- Parity: keine
- Baud rate: 115200
- Stop pbits: 1
- Flow control: keine

Measure

To initiate a measurement use As-Built measure button. With a Geomax Zipp 20 it is not possible to initiate the measurement from the instrument side.

7.4.16   Instruments of type Trimble 3300

The settings described here, have only to be set when working with As-Built for the first time. These settings will be saved in the instrument and when you turn on the instrument the next time, you can start working right away.

Total station settings

Press the buttons **SHIFT and MENU** simultaneously to get to the main menu.

Go to the interface settings (item **6)** and confirm with '**YES**' (same button as **MENU**). Set the individual points to following values. The values can be changed with '**MOD**' (same button as **MENU**).

- **0 Registr**.:   V24/1 (Sends raw measurements to interface)
- **2 Format:**   M5 (Data format)
- **3 Parity**:   none
- **4 Baud rate**: 9600
- **5 Protocol**:   XON/XOFF

In principle this communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

Leave the menu by pressing '**ESC**' twice (same button as '**OFF**').
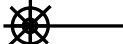
Measure

Following settings have to be set every time the instrument has been turned on:

To switch to the reflectorless measurement mode, keep the '**SHIFT**' button pressed and push '**DR**'. A blinking symbol will appear in the upper right corner of the display.

It has to have following shape for the reflectorless measurement: ⊕

This shape symbolizes infrared measurement against a reflector: ⊗

To switch on the visible laser point, keep the 'SHIFT' button pressed and push this button: ⊛

Check that the values for the sloped distance, horizontal angle and vertical angle are shown in the measurement display:

```
SD              0.000 m
Hz          378.5425gon
V           130.5720 gon
HzV  Hz=0  V%   SCHN  →2
```

If this is not the case, press the '**OFF**' button until this display appears.

Measurements can be released with '**MEAS**'.

Settings on the computer

Set the As-Built Instrument selection to the "Trimble 3300 / M3(old) " profile.

7.4.17 Instruments of type Trimble 3600, 5600

These instruments are equipped with a variable control unit. Depending on the control unit you use, the settings differ.

Trimble with Zeiss control unit:

Using Trimble instruments with this control unit the measurement can only be initialized from the instrument side. Initialization of measurement at the notebook (by As-Built) is not possible.

1. Go to the main menu and choose the setting for registration of the target data. You may find it under the menu item *Configuration*. Switch to *External* registration with the space bar.

2. Switch the registration format to *REC E* in the same sub menu. You find a function key assignment in the bottom line. Select the item *Config* with FCN + according to the displayed button. Here you can check the communication parameters and adjust them if necessary. Following parameters are standard for the communication:

   Speed: 9600

   Data bits: 8

   Parity: none

   Stop bit: 1

   In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same.

3. Change into the measurement program via the main menu. This program remains active throughout the measurement. Make sure that the registration mask that shows slope distance, horizontal direction, vertical direction is activated. You can adjust this setting with **FNC** (function key) + *Mode*. The registration mode has to be set to *R-M.* in order to display only the measurement values. This setting can be changed with **FNC** + the referring displayed button.

The visibility of the laser beam can be controlled with '**Shift**' + **Symbol key 'Laser beam'**.

All settings are saved permanently.

Trimble with Geodimeter control unit

- Set the communication parameter 1.8.0.9600 under menu item (MNU) 4-1-2. Set *Table 0* and *RS232* to '1 = on' in the same sub menu in order to send all measurement data to the interface.

- Confirm the query for the REG button with '*YES*'.

- The function key **F79** has to be set to the predefined value 62. This corresponds with the terminator.

- Switch to the reflectorless measurement mode by setting *RL-measurement* (DR mode) in the menu item (MNU) 7-2. (MNU 7-2-1 → R = with reflector, MNU 7-2-2 → DR = without reflector).

- Change to menu item 7-4 and set the *Pointer* to 'on' in order to make the laser visible.

- A measurement is released with the button '**A/M**'. To send the measurement, you have to press '**REG**' afterwards.

Trimble with ACU control unit

This device configuration unfortunately <u>cannot</u> be used together with As-Built.

The ACU is not able to send the measured data to the serial port.

7.4.18   Instruments of type Trimble S3; S6, S7, S8, VX

With the **Trimble Precision SDK** (TPSDK), Trimble offers new possibilities for the communication of their tachymeter with third software like As-Built. All newer Trimble series can be triggered with it.

- S series

- VX

- M3

- Spectra Precision Focus 30

The communication with the instrument is managed by the TPSDK.
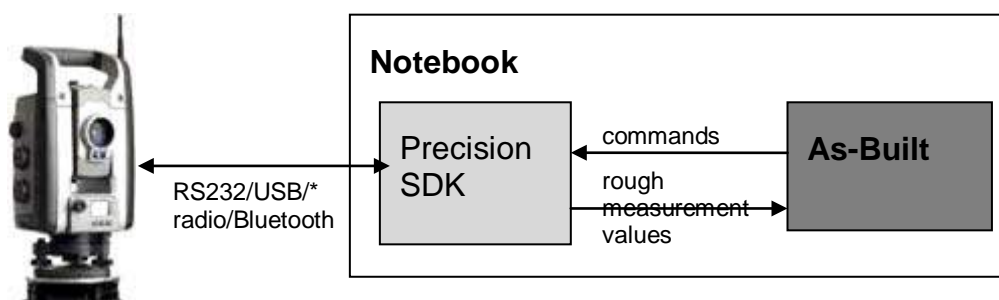
Motorized devices can be controlled from the notebook. Functions like approaching a certain point or the work with prism tracking are possible.

The configuration of the instrument can only be carried out on computer. When there is a connection over the TPSDK, the total station cannot be controlled by a Control Unit or from the keyboard. That means that no settings can be entered on the instrument. The most important observation settings (bubble, battery indicator, EDM Mode, addition constant etc.) can be directly called from As-Built and adjusted.

It has been possible to trigger observations on the total station since As-Built Version 16.5.

Installation

To be able to use the TPSDK with As-Built, the TPSDK Redistributable Pack (Version 2.6) must be installed.

To do so, at the end of the installation process, enable the "Install Trimble device drivers (TPSDK) option:

Under current operating systems, such as Windows 10, a message regarding the non-existence of version 3.5 of the .Net Framework may appear during the installation of the TPSDK. If you get a message stating .NET Framework version 3.5 is missing, install the feature. An internet connection is required for installation otherwise the Microsoft installer will abort.

The "Tell me more about this feature" link in the dialog box above, provides additional information from the following Microsoft help article: Install the .NET Framework 3.5 on Windows 10, Windows 8.1, and Windows 8.

Should you have missed to install the TPSDK durching the As-Built setup, you may install it later. You will find it in the installation folder of As-Built under \Trimble\TPSDKRedist26.exe

The TPSDK Redistributable Pack (after Version 2.6) can also be downloaded from http://service.faro3dsoftware.com/supportfiles/.

Online activation

The TPSDK is free of charge. Before the first use a one-time activation at Trimble is necessary for every computer. The activation process is controlled over the instrument selection dialogue box:
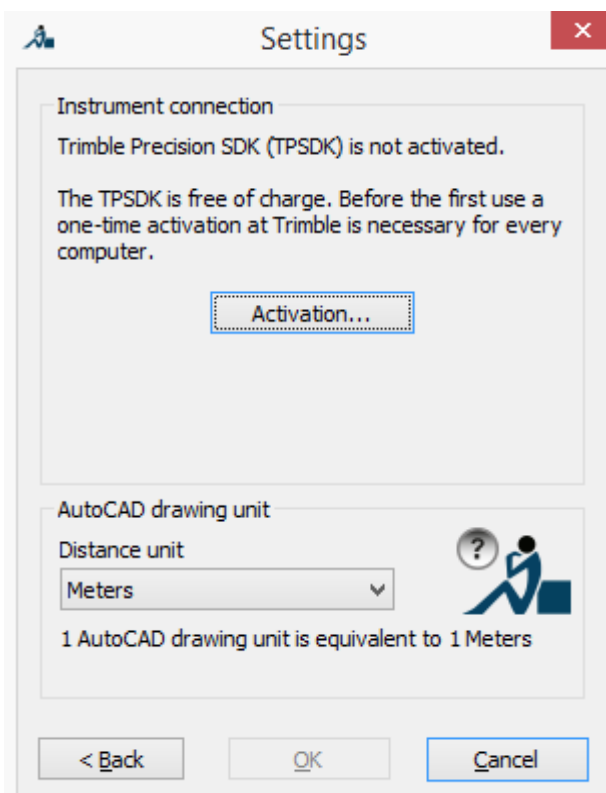
- Over "**Activation**" you are taken to a further dialogue box.

- There select as the first function **"Send activation request by email"** → an email is automatically sent to Trimble with your request and the computer ID

- Wait for an activation confirmation from Trimble.

- As soon as an activation is available, go back to this dialogue box and select the function **"Install license"**. Your licence file will be automatically downloaded from Trimble. You need an active internet connection.

- Done. As soon as the TPSDK has been activated you can make the other settings for communicating with the instrument, in the instrument selection dialogue box.

Connection establishment

On the total station

Should you be using a Trimble Total Station of the type **Typ S3**, you should next close the Trimble Access application on the instrument.
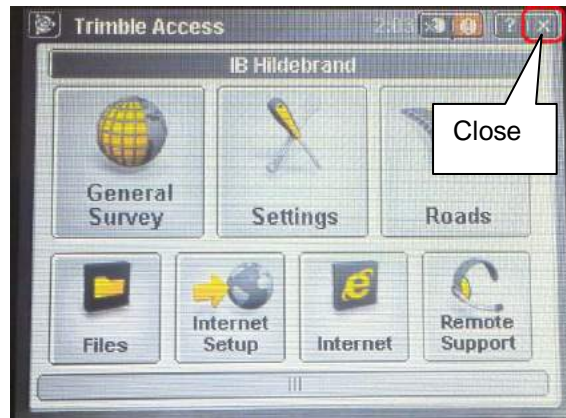
Thereafter start the application "External connection". If you want to make a connection via **Bluetooth**, please check under Settings/Comm. that the configuration value for "Port" is set to "Bluetooth".

On the computer

Set the As-Built Instrument selection to the "Trimble S Series" profile. After choosing the instrument and setting the units you will be asked for the connection .

- **USB cable:** Set the "Connection Types" to USB. Please leave the remaining input boxes empty.

- **Serial cable**: Set the "Connection Types" to "Serial". Select and under "Connection" the corresponding connection port number to on the computer. It is explained in section →"Finding the correct COM port".

- **Bluetooth:** In the first step setup your Trimble instrument with the help of the Bluetooth environment of your operation system. When this is finished select "Bluetooth" as "Connection Type". As port you will have to state the COM port of the serial interface which will be simulated via Bluetooth. You can find it within the settings (heading "Service" of the paired Bluetooth device. Please enter the main key of the device as pin (**last 4 digits of the device serial number**)
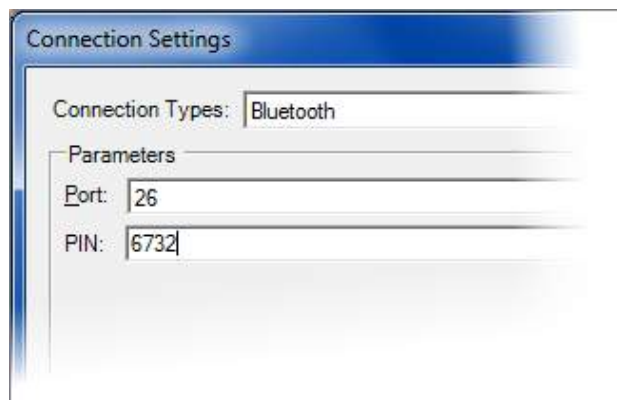
- **Radio**: Select "Trimble Radio (Rover)" for the "Connection Types". Set Port, Channel and Network ID.
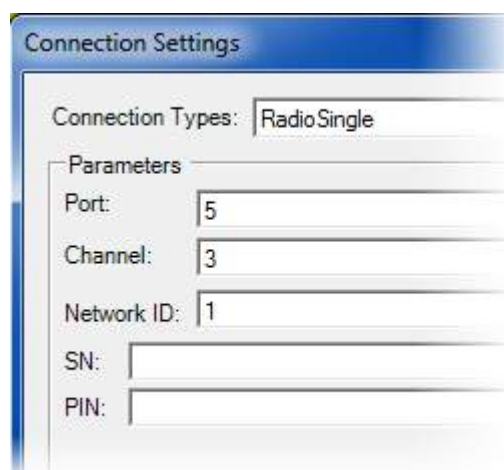
As soon as you confirm the settings with OK the connection to the instrument will be established. You can see by the status report within the Windows info area if the connection was successful.

Total station settings

As soon as the connection between instrument and TPSDK has been established, the configuration of the tachymeter is only possible via the As-Built command →**Total station settings**. As-Built saves the total stationr settings and will load them automatically for every new connection.

Measuring and controlling

As soon as the connection between instrument and TPSDK has been established, measurements can be triggered within As-Built or for motorized devices controlling commands can be used.

The triggering of an observation on the instrument can be done using the ENTER key on the instrument's own internal display.
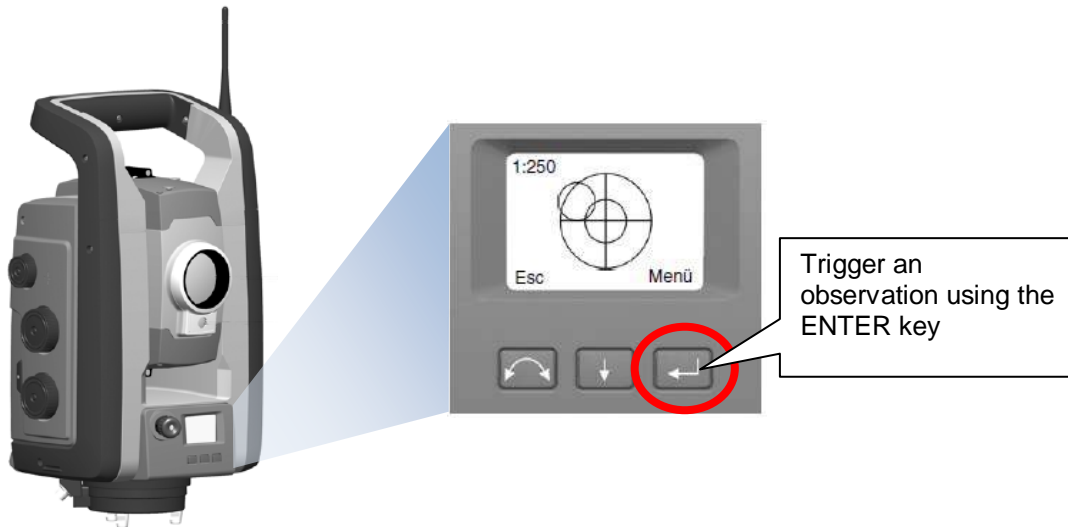
As the removable keyboard/control unit of the instrument does not accept any input in this mode, it cannot be used to trigger an observation.

Please check the total station settings before the first measurement.

7.4.19 Instruments of type Trimble M3

To control a Trimble M3 you need the "Trimble Precision SDK". Information for installation and activation can be found in the section → "Instruments of types Trimble S3, S6, S7, S8, VX".

Establishing a connection

On the total station

First please close the Trimble Access application on the instrument.

Thereafter start the application "External connection". If you want to make a connection via Bluetooth, please check under Settings/Comm. that the configuration value for "Port" is set to "Bluetooth".

On the computer

Set the As-Built Instrument selection to the "Trimble ME" profile. After choosing the instrument and setting the units you will be asked for the connection.

- **USB** cable: Set the "Connection Types" to USB.

- **Serial cable:** Set the "Connection Types" to "Serial". Select and under "Connection" the corresponding connection port number to on the computer. It is explained in section → **"Finding the correct COM port".**

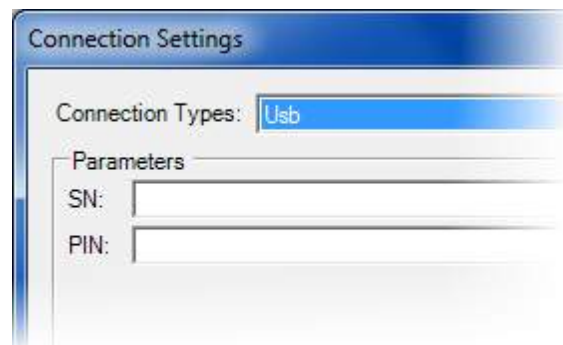- **Bluetooth:** In the first step setup your Trimble instrument with the help of the Bluetooth environment of your operation system. When this is done, select "Bluetooth" as "Connection Types". As "Port" you will have to state the COM port of the serial interface which is to be simulated via Bluetooth. You will find these in Settings (heading "Services") of the paired Bluetooth devices.

As soon as you confirm the settings with OK, the connection to the instrument will be established. You can see if the connection was successfully made on the status bar in the Windows information area.

Total station settings

As soon as the connection between the instrument and the TPSDK has been established, the total station can only be configured from the As-Built command → **Total station settings**. As-Built saves the total station settings and loads them again automatically when a new connection is made.

Measuring and control

As soon as the connection between the instrument and the TPSDK has been established, observations can be triggered in As-Built and on motorised instruments control commands can be issued. It is not any longer possible to trigger any observations on the instrument, as the keyboard/control unit does not accept any input in this mode.

Please check the total station settings before making the first observation.

7.4.20 Instruments of the type Trimble M1

On the computer

Set the As-Built Instrument selection **(→ Select instrument**) to the profile "**Nikon / Trimble M1**".

On the Total station

The settings on the instrument are identical to some NIKON instruments. Please proceed as decribed in Chapter **7.4.36 Instruments of the Nikon DTM 302 type .**

The Trimble M1 can be connected via serial cable or wireless via Bluetooth.

7.4.21 Instruments of the type Spectra Precision Focus 30

To control a Spectra Precision Focus 30 you need the "Trimble Precision SDK". Information for installation and activation can be found in the section → **"Instruments of types Trimble S3, S6, S7, S8, VX".**

On the total station

Open the main menu. Should you have any applications open (e.g. "Survey Pro"), then they should be closed. Should the closing of the applications bring no success, then please start the total station operating system again. Usually after a new start of the operating system no application will start up that can block the port.

If you want to establish a **Bluetooth** connection please start the app "BT Link" in the main menu. For all other types of connection please stay in the main menu.
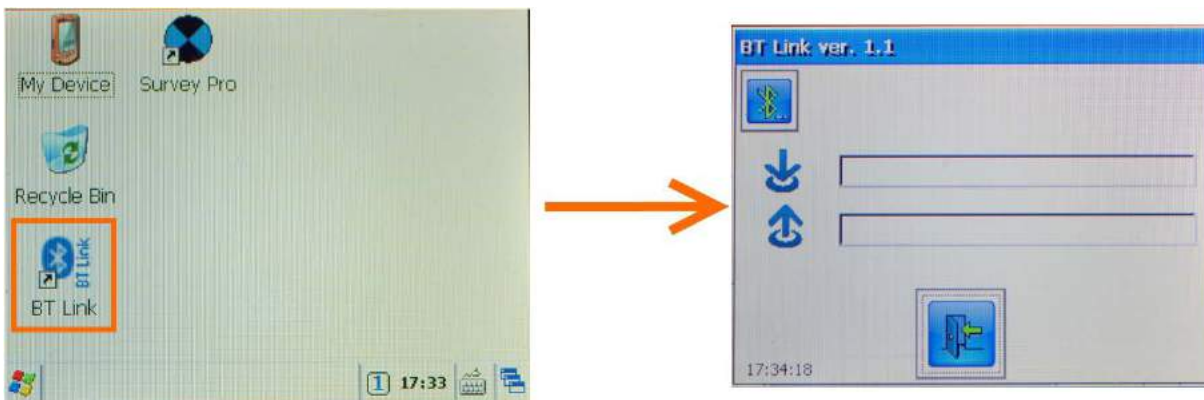
On the computer

Set the As-Built Instrument selection to the "Trimble Precision Focus 30" profile. After choosing the instrument and setting the units you will be asked for the connection.
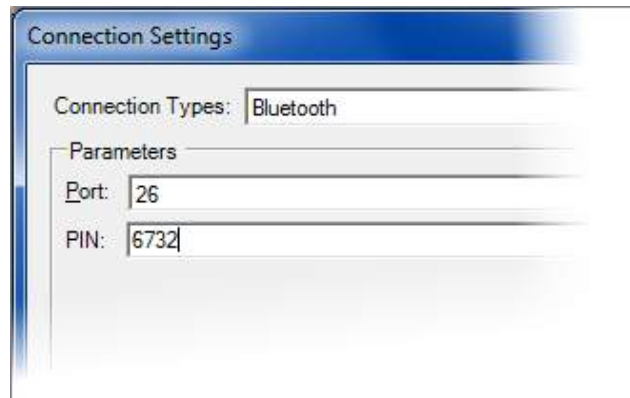
- **USB cable:** Set the "Connection Types" to USB.

- **Serial cable:** Set the "Connection Types" to "Serial". Select and under "Connection" the corresponding connection port number to on the computer. It is explained in section →**"Finding the correct COM port"**.

- **Bluetooth:** In the first step setup your Trimble instrument with the help of the Bluetooth environment of your operation system. In the first step setup your Trimble instrument with the help of the Bluetooth environment of your operation system. As "Port" you will have to state the COM port of the serial interface which is to be simulated via Bluetooth. You will find these in Settings (heading "Services") of the paired Bluetooth devices.

- **Radio:** Select "Trimble Radio (Rover)" for the "Connection Types". Set Port, Channel and Network ID.

As soon as you confirm the settings with OK, the connection to the instrument will be established. You can see if the connection was successfully made on the status bar in the Windows information area.

Total station settings

As soon as the connection between the instrument and the TPSDK has been established, the total station can only be configured from the As-Built command → **Total station settings**. As-Built saves the total station settings and loads them again automatically when a new connection is made.

Measuring and control

As soon as the connection between the instrument and the TPSDK has been established, observations can be triggered in As-Built and on motorised instruments control commands can be issued. It is not any longer possible to trigger any observations on the instrument, as the keyboard/control unit does not accept any input in this mode.

Please check the total station settings before making the first observation.

7.4.22 Instruments of the type Spectra Precision Focus 8

These instruments are, in terms of their functionality and control from As-Built, identical to the **Trimble M3**. Please read the section → **"Instruments of type Trimble M3".**

7.4.23 Instruments of type Trimble M3 (without WIN CE, before 2010)

Important hint:

If you are using and older M3 (without WIN CE, until 2010), you can use the settings described below.

If you are using a newer M3 (with WIN CE running, from 2010), these settings will not work. On these instruments new software is running, which gives no option to send the measurement data on an external interface. To run these instruments with As-Built, please proceed as described in the chapter „**Instruments of type Trimble S3; S6, S7, S8, VX; M3**".

Settings

In the As-Built "Select instrument" dialogue you choose the entry „Zeiss (R-Series), Trimble 3300, Trimble M3".

At the instrument itself you have to make the following settings:

Reflectorless Measurement:

- press the Trimble symbol key → the HOT MENU opens
- "2. DR/PR mode" → by pressing the key **2** one can switch between DR and PR mode (DR = Direct Reflex = reflectorless)

Configure instrument communication:

- switch to the main menu using **MENU**
- Select the „6. Interface" by navigating with the cursor keys and confirm with **MEAS/ENT**
- Make the following settings:

    - Stack: V24/1
    - Reg mode: All
    - Reg setting: yes
    - Format: M5
    - Parity: none
    - Baud rate: 9600

Measure:

Press the key MEAS/ENT at the instrument to run the measurement. Starting the measurement from the notebook is not possible.

The instrument cannot be set to pure direction measurement. It always does a complete measurement. When measuring pipe lines with As-Built or when using the As-Built functionality for arc intersection one has to do complete measurements. The not used distance will simply be ignored by As-Built.

7.4.24   Instruments of type Trimble S6, S7, S8, VX (obsolete connection)

With this obsolete communication method use of the instrument motorization is not possible. We suggest using the new instrument connection (see → **Instruments of type Trimble S3; S6, S7, S8, VX; M3**)

Trimble S6 (S8, VX) instruments are equipped with a detachable control unit (Trimble CU) which runs with a Windows CE operating system and a Trimble software. For configuration do as follows:

Trimble software "Survey Controller"

Switch on the instrument and start the **Survey Controller** application.

Open the menu item **Instrument** – **GDM data output** and make the following settings:

- Output: after measurement
- Format: HZ V SD
- Interface details: Baud rate=9600
- Data bits=8
- Stop bits=1
- Protocol: none

The "GDM data output" window has be open during the measurement, **do not click STOP**. (Use the "Change" option in order to switch back to the Survey Controller.)

Turn on the **Standard measuring mode**: press the Trimble key   and select the „STD mode".

For initiating the measurement at the instrument press „Measure" or the key on the side of the instrument.

Trimble software „Trimble Access"

- Turn on the device and start the "Trimble Access" application.
- Select "**General Survey**".
- Select "Instrument" and then "**GDM data output**" from the list
    - Output: After the measurement
    - Format: HZ V SD
    - Controller interface: Instrument COM
    - Baud rate: 9600
    - Data bits: 8
    - Stop bit: 1
    - Data flow check: none

- The "GDM data output" window has be open during the measurement, **do not click STOP**. Minimize the window with the "_" key down right.

- Select "**General Survey**" – "Instrument" and then "**Survey Basic**" from the list.

- In order to initiate a measurement on the tachymeter push the "**Measure**" button or the button on the side of the instrument.

- In order to set/switch the measurement mode you may push the Trimble button

  at any time. You may switch between:

  - Reflectorless (DR) and measurement to prism

  - Laser point on / Laser point off

  - Standard measurement and tracking (continuous measurement) →Standard (STD) is suggested.

Please note:

Data communication with a Trimble S or VX devices is limited from manufacturer side:

- Normally one could initiate the measurement at the instrument as well as by mouse or keyboard command in As-Built. With a Trimble S6 the initiation of the measurement possibly only works in one direction (initiation at the instrument). Initiation by As-Built is only possible if the software installed on the instrument is older than:

  - Firmware 4.1.3.

  - Survey Controller: 11.32.

- The communication between As-Built and Trimble S or VX series only works when using a combined cable Bluetooth connection. Instead of a direct connection via cable a Bluetooth to serial adapter is used. The advantage of that wireless connection it provides better working conditions with tablet PCs and other mobile solutions.

Notebook

Instrument

Unfortunately a communication directly via the cable or the internal Bluetooth of the tachymeter is not possible.

Please contact your As-Built salesman for more detailed information on Bluetooth to serial adapters.

Remarks regarding the TSC2

Trimble instruments of the types S6/S8/VX can alternatively be controlled by an external control unit called TSC2.

The user interface of the Trimble TSC2 is identical to the detachable control unit (TCU). On both runs a Windows CE operating system. That means all settings for communication are the same as described above.

To connect your TSC2 with your notebook you need a zero modem cable, which you attach to the serial port of the TSC2. The other end of this cable you may plug to the serial com port (COM1) of your computer. If the notebook does not have a serial port you have to use a USB to Serial adapter or if you want to move around more freely you may use a Bluetooth to Serial adapter (see above).

Caution: If TSC2 and notebook are connected via a cable (without Bluetooth) you have to set the handshake to 'Hardlock' in the As-Built dialogue for selecting the instrument.

7.4.25 Instruments of type Topcon GTS 200, GPT 1000/2000/6000/7000

In order to be able to use this instrument with As-Built, the following settings need to be made just once.

<u>In As-Built:</u>

Run the command "Select instrument" and select the instrument profile "Topcon Standard Modus".

<u>On the instrument</u>

In order to use Topcon instruments, the following settings have to be checked and if necessary, to be adjusted. Depending on the precise instrument type the terms in the menu may differ.

- Turn the instrument off.

- Then, turn the instrument on while pressing the '**F2**' button. The menu for setting the instrument parameters will appear.

- Switch to the sub menu "**MEASURING MODES**" and make the following settings:

    - **When starting:** distance measurement (when turning on the instrument it starts directly with distance measurement mode)

    - **Precise/Coarse/Trck:** Precise

    - **HD and SD/VD:** SD,H,V

    - **V Display:** 0 Zenith

    - **N-times/Permanent:** N-times measurement

    - **Number of measurements:N:** 1

    - **ESC switch:** REG (the 'ESC' key gets the functionality 'Measure and register data)

    - **EDM off after:** 0 minutes (if one sets a value different to 0, instrument measures permanently. This makes work faster but uses more power resources)

    - **PRISM:** none

    - **MIN. Display:** 1mm

- Switch to the sub menu "**MISC**" and make the following settings:

    - **ECHOBACK: ON**

    - **CR/LF:** ON

- Leave the menu for setting the instrument parameters by turning the instrument off when all settings are correct. All settings are permanent. Thus, it is not necessary to check and adjust these parameters every time you use the instrument.

- Now turn the instrument on again.

- The parameters for the data transfer have to be set now. Press the button '**MENU**' and choose the setting for Registration/communication. Set the communication parameters for the data transfer as follows:
  - **PROTOCOL**: ACK/NAK
  - **BAUD RATE**: 2400
  - **CHAR./PARITY:** 7/EVEN
  - **STOP BITS:** 1

The communication parameters are allowed to differ from those proposed here. But you have to make sure that in the instrument settings in As-Built you set the same parameters you have chosen in the Topcon menu. Your Topcon is now ready to use with As-Built.

All necessary settings have been done. There is no possibility of obtaining incorrect measurement values because of a wrong setting. All settings are checked by As-Built during the measurement process.

The measurement itself is very simple. Following actions are necessary at the instrument when As-Built asks you to release a measurement: Switch to the measurement menu. This is the menu which is active when the instrument is turned on. Make sure that the registration mask displays:
- slope distance SD
- horizontal direction HD
- vertical direction V

After aiming with the telescope, initiate a measurement at the instrument with '**ESC**' and confirm the recording with 'YES'. This process is repeated for every point. Alternatively, you can also initiate a measurement from your notebook.

7.4.26   Instruments of type Topcon GTS 3000

In order to be able to use this instrument with As-Built, the following settings need to be made just once.

<u>In As-Built:</u>

Run the command "Select instrument" and select the instrument profile "Topcon Standard Modus".

<u>On the instrument</u>

In order to use Topcon instruments, the following settings have to be checked and if necessary, to be adjusted. Depending on the precise instrument type the terms in the menu may differ.

- Switch to Main Menu. Open sub menu "Parameter1 – RS232":
    - **PROTOKOLL**: ACK/NAK
    - **BAUDRATE**: egal
    - **CHAR./PARITÄT**: 7/GERADE
    - **STOPBITS**: 1
    - ACK Modus: Ohne
    - Rec Type: Rec A

- Switch to sub menu "Parameter 1 – Minimum Reading and set to 1mm/1mgon.

- Turn the instrument off. Then turn the instrument on while pressing the '**F2**' button. The menu for setting the instrument parameters will appear (Parameter 2 Mode).
    - **Power on mode: Distance meas. (Select to set the measurement mode for rdistance when the power is turned on.)**
    - **Fine/CRSb/Trk: Fine**
    - **HD & SD/VD: SD,H,V**
    - **V angle: Zenith 0**
    - **N-times/repeat: N-times**
    - **Times of meas.:   N:1 (number of times of distance.**
      1=single measurement.)
    - **ESC Key mode: REC ("ESC" Key gets functionality of „trigger measurement and save data")**
    - **EDM off time: 0 (If set to another value, the instrument measures permanently. That accelerates work but spends more energy.)**
    - **Non prism/Prism: non prism (aiming points directly with laser dot)**
    - **Fine reading: 1mm**
    - **Echo back: off**

7.4.27 Instruments of type Trimble 3600, 700

In order to be able to use this instrument with As-Built, the following settings need to be made just once.

<u>In As-Built:</u>

Run the command "Select instrument" and select the instrument profile "Topcon Standard Modus".

<u>At the Topcon:</u>

1) Parameter settings:
- Min. Dist: **OFF**
- REC –A/B: **REC-A**
- CR/LF: **ON**

2) Communication settings:
- F1:   Protocol
- Speed: **1200**

In As-Built you select Topcon from the list of instruments and set the following communication parameters:
- Baud rate: **1200**
- Parity: **Even (E)**
- Data Bits**:**   7
- Stop Bits: 1
- Handshake: **Xon/Xoff**

Please note that for As-Built you always have to measure in SD (Slope Distance) mode.

In principle the communication parameters may vary from the suggested. All you have to do is to make sure, that the set parameters in the As-Built instrument settings are the same. Now your Topcon is ready to measure.

After aiming with the telescope, activate the measurement at the instrument and confirm the saving. This has to be repeated for every point to measure. Alternatively you may start the measurement from the notebook.

7.4.28   Instruments of type Topcon GPT 8000, 9000

In order to be able to use this instrument with As-Built, the following settings need to be made just once.

In As-Built:

Run the command "Select instrument" and select the instrument profile "Topcon External Link Modus".

On the instrument

First of all switch to the main menu. In case the instrument is set to '**External Link**' mode you have to leave that first by pressing '**ESC**' until you get back to the main menu.

Set units:

In the main menu select the item 'Parameters' (F6) and press F1 to go to 'Measurement". Set the angle unit to **gon** and the distance to meters. Close the dialogue with SET (F1) soft-key. Answer the question 'Set parameters?' with yes (F5).

Communication parameters:

Still in the 'Parameters' menu you press F2 to go to 'Communication' where you set the following parameters for RS232C (F2):

- Baud rate: 4800   (with radio link RC 2400)
- Dat.L: 8
- Parity: none
- Stop bits: 1
- Flow control: none
- End mark: CR/LF
- Type RecA/B: REC B

Set reflector/reflectorless measurement:

- Press the (*****) button and then F5. If 'NP' blinks in the upper right of the display it means no prism needed for measurement. If 'P' blinks the measurement has to be done to a prism.

- In standard measuring mode data cannot be transferred via serial interface. Thus your Topcon has to be set to **External Link** mode.

- In the main menu press F1 to switch to 'Programs' and navigate with F6 to the next page.

- With F4 select 'External Link'.

- Press F2 to select 'Set'. With F1 you can choose between cable, modem and RC radio. Save your setting with F1 (Set) and you get back to the previous menu.

- Here press F2 to select 'Parameters' (cable) to check or set the following communication parameters (analogue you could set the parameters for RC or modem with F3 or F4). With SET (F1) you set the respective parameters and get to the next query:

  - BAUD RATE: 4800 (for RC 2400)

  - BIT FORMAT: D8 S1 KEINE

  - END MARK: EXT+CR+LF

  - RecTYPE: REC-B

- After having set the last parameter you get back to the menu 'SETTINGS' which you exit with ESC.

- Now press F1 to execute and the instrument will hand the control to an external instrument e.g. your computer (As-Built).

Measure

To initiate a measurement use As-Built measure button. With a Topcon GPT 8000 it is not possible to initiate the measurement from the instrument side. Because it runs in 'External Link' mode the key commands widely deactivated so the instrument can only be remote controlled now.

7.4.29 Instruments of the Topcon IS series type

The configuration described below is suitable for the conventional work with the tachymeter. That means you will operate the tachymeter directly at the display and you will approach the measurement points manually. For the use as one-man station other settings have to be executed, those will be described within the following chapter.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The following settings have to be made:

After turning the device on you will first get to the Windows CE desktop view.

Start the application "**Standard Meas**".

Move to the main menu. Open the following sub menus, you can put the individual values through by clicking "Continue".

→Settings

- Measurement – Parameter measurement 1:

    fine reading 1mm

    Distance mode: fine

    Distance display: SD

    V angle Z0/H0: zenith

    distancemeas number: n-time

- Measurement – Parameter measurement 2:

    (Nothing has to be set here.)

- Communication – common parameter:

    REC key data out: RS232C / Bluetooth / SSWireless[5]

    REC TYPE: REC-A

    TRK state: Off

    REC Key: ON

- Communication – RS232C:

    Baud rate: 2400

    Data bits: 7

    Parity: Even

    Stop bits: 1

    CRFL: ON

---

[5] Set your desired connection type here (RS232C=cable).

ACK-Mode: Off

- Value input (EDM):

distmeascont: 1

EDM off time: 1 (measurement)

- Units:

Distance: m

Angle: gon

→Meas

This is the measurement menu which has to be active when measuring.

The following settings do have to be done every time you turn in on.

- Turn on slope distance: Push the display key ◢

- Switch to reflectorless measurement: Push the ★ key, then set the NP/P function to RL (NORMAL) → within the measurement display you will see "NP" for "No Prism".

- Switch on visible laser: Push the ★ key, then set the ⊟ function to ⊟

Measuring:

The first measurement has to be initiated with the display key REC. All other measurement may also be started via the ENT key. The prompt "Data out?" will appear after every measurement, answer with "YES" (This is done fastest by pushing the ENT key.)

The measurement may alternatively be initiated via the measurement key from the As-Built measurement box. Then the "Data out" query will not appear.

7.4.30   Instruments of type Topcon DS

In order to be able to use this instrument with As-Built, the following settings need to be made just once.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

<u>In As-Built:</u>

Run the command "Select instrument" and select the instrument profile "**Sokkia SET**"[6].

<u>On the instrument</u>

First of all switch to the main menu. In case the instrument is set to **'External Link'** mode you have to leave that first, by selecting END and with the PRG key exit the program menu.

In the main menu select the option "Config". Set the following values. Values that are not listed here have no meaning when working with As-Built and you can freely decide yourself what they should be.

- 1.Obscondition:
    - Dist.mode: SDist (measure slope distance)
    - V.obs: Zenith
    - V.manual: No

- 4. EDM:
    - Dist.mode: Fine AVR N=1 (Single observation)
    - Reflector: N-Prism (for reflektorlesss) or "Prism" or "36° prism" (for working with a reflector)

- 5.Comm:
    - Comms.mode: "Bluetooth" oder "RS232" (cable)
    - Checksum: No
    - Rec Type: REC-A
    - Terminate: EXT+CR+LF
    - Ack mode: off
    - Tab "RS232": Here the communications parameters must be set (Baud rate, data bits, parity, stop bit). Here use the same values as those that set in the As-Built instrument selection (command "Select instrument").
    - Tab "Bluetooth" Authentication=No

Measure

The triggering of an observation is done in As-Built by clicking the red Meas key in the measbar.

Unfortunately it is not possible with this instrument type to trigger the observation from the instrument.

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[6] That appears to be illogical, but it is because the current Topcon instrument uses Sokkia technology.

7.4.31 Instruments of the Topcon IS/GPT 8000/GPT 9000 type in one-man operations

The configuration described here is suitable for the use in one-man operations.

For that the person is not standing at the tachymeter but is walking with the reflector pole and the notebook from measurement point to measurement point. The instrument follows the prism automatically with the telescope and points at it. The measurement will be triggered by the push of a button within As-Built.
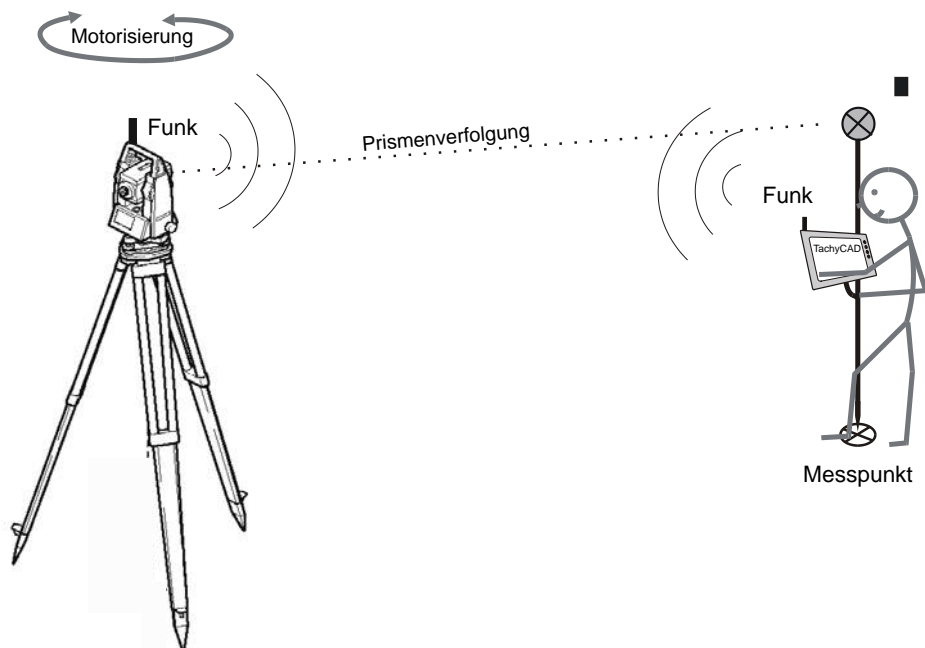
This mode of operations makes sense for measurement projects where topographic objects have to be measured outdoors. It is not suitable for measurements without reflector.

Requirements are an appropriate notebook (suitable for the outdoors, small and lightweight) as well as a stable wireless connection between the tachymeter and the notebook.

Next to the usual As-Built measurement functions a number of special commands for the usage as one-man station are available. You will find further information about this within the As-Built command reference:

- ➜ Start prism search
- ➜ Activate/deactivate prism tracking
- ➜ Poll battery status of the instrument
- ➜ Change face position

Reasonable additional equipment for the one-man operation:
The prism pole may be upgraded with the remote control unit RC-4. It establishes the wireless connection to the tachymeter and enables a fast localization of the prism via the built-in infrared sensors.

Settings

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

The following settings have to be executed once:

After switching the tachymeter on, you will get to the Windows CE desktop view at first.

Start the "**Standard Meas**" application.

You are now within the main menu. There call up the following sub menus. You may put the individual values through with the "Next" key.

PROG – Ext.Link

With this the **External Link mode** has been switched on. This screen will stay open during the complete measurement. The triggering of a measurement is only possible from the outside, not from the instrument itself.

Via the "**Setting**" button you will get to the further settings for the connection type of the tachymeter. After clicking the according button you may set the respective communication parameters for the type of connection:

- RS-232C – communication via cable

  - Baud rate, data bits, stop bits, parity: These have to comply with the parameters set within As-Built.

- RC - communication via the tachymeter wireless

  - "Enable" activate
  - Channel: wireless channel

- SS Wireless – communication via WLAN

  - "Enable" activate
  - Channel: wireless channel

- Bluetooth – communication via Bluetooth

  - "Enable" activate
  - In order for the communication to work, the following communication parameters have to be set within As-Built: Baud=19200, data bits=7, stop bits=1, parity=none.

These settings have to be set for every type of connection:

  - Rec Type: REC- A or REC-B (equal)
  - TERMINATE: EXT+CR+LF

★key (measurement settings)

Here you may access all important measurement settings at any time.

Prism tracking

Fine pointing

Change face position

Set parameters for prism tracking.

NP

Check signal strength

Electronic level

Point guide light or tracking indicator

Autofocus / manual focus

Measurement mode prism / without reflector

Laser point

Adapt addition constant and other settings

Cross hairs illumination
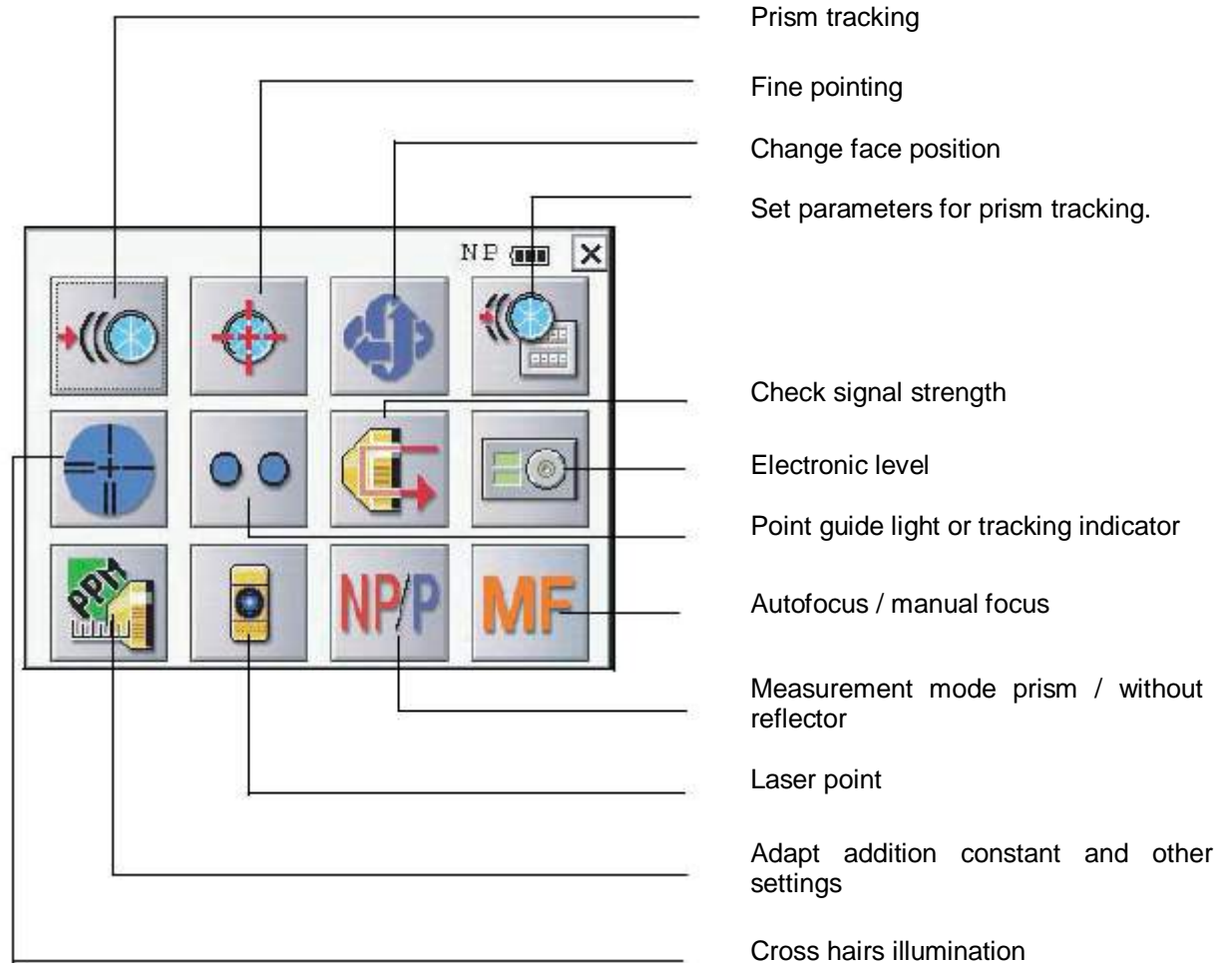
7.4.32 Instruments of type Sokkia SET series 30R

To use instruments of this series with As-Built the following settings have to be checked and if necessary changed.

Settings in As-Built

In As-Built under ➔**Instrument measurement - Select instrument… select the SokkiaSET/POWERSET.**

Settings in the instrument's menu

Open the main menu. You get there by pressing the soft-key **CNFG** on the start side of the display. Please check the following settings.

- *Comm.settings*

  - Baud rate: 2400
  - Data bits: 8bit
  - Parity :   none
  - Stop bit :   1bit
  - Check sum: No
  - XonXoff: No

  In principle you could work with other parameters too. But you have to make sure that identical values are used in As-Built.

  ➔**As-Built command reference ➔ Instrument measurement - Select instrument …**

- *Units*

  - Angles: Gon
  - Dist: Meters

  In principle you could work with other units too. But you have to make sure that identical values are used in As-Built.

  ➔**As-Built command reference ➔ Instrument measurement - Select instrument …**

- *Allocation of keys*

  Go to the sub menu 'Definition'. Link one of the four function keys (e.g. F1) with the symbol for 'Data transfer to PC or external storage':

  Exit the menu by pressing ESC several times.

- **Settings for reflectorless measurement**

  In the measurement menu press the **PPM / PC** key.

  Choose a mode depending on the requirements and select:

  'Reflector: none'

- **Initiation of a measurement**

  If you want to initiate the measurements directly by As-Built you do not have to observe anything.

  For initiating the measurement from the instrument side you have to switch to the measurement menu and press the function key that is linked with the

  symbol. In the following query you select either 'Dist data' or 'Angle data' depending on what you need a full measurement or just a direction measurement. Then press the ENTER key of the instrument. Now the measurement will be done and the data will be transferred to As-Built.

7.4.33 Instruments of type Sokkia CX / Topcon ES

The following instrument settings are to be made to establish a connection over Bluetooth with As-Built.

On the total station

Change in configuration mode.

To do this select the option **CNFG** from the status page.

- **Comms setup**
  - Wireless → **Yes**
  - Bluetooth setup → Mode → **Slave**
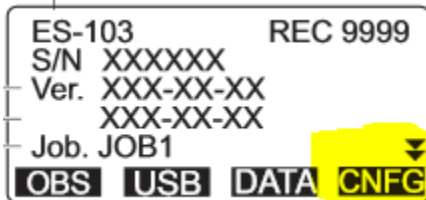 - Bluetooth setup → Authentication → **Yes**
  - Bluetooth setup → Passkey → **0000**
  - Bluetooth setup → Checksum → **NO**
  - ACK/NAK → **NO**
  - CR,LF → **NO**
  - ACK mode → **Standard**

- **Unit**
  - Angle → Angular units for measuring bearings
  - Dist → Linear units for measuring distances

Change into data mode. To do this select the option **DATA** from the status page.
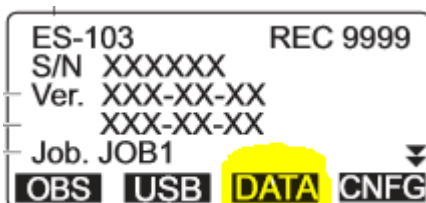
- **Jobs**
  - comms output → **S Type** (Data format)

On the computer

In the Windows Bluetooth environment start a search for devices. As soon as the Topcon/Sokkia device has been found pair it with the Windows computer.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

Should you be prompted to enter a pairings code, please use the pass key 0000, that was previously entered on the instrument. During the establishment of the connection Windows automatically allocates a Bluetooth COM port to the device. Note down the COM port number.

In As-Built start the command ➔ **Instrument settings - Select instrument**.

Select the instrument profile "Sokkia Set" and in the next dialog select the connection type Bluetooth.

Units:

Under "Total station measurement units" you must ensure that units correspond with the units settings in the instrument.

7.4.34   Other instruments of type Sokkia SET

In order to use Sokkia SET instruments, following settings need to be checked and to be adjusted if necessary.

- Distance unit: meter [m]

- Angle unit: gon [gon]

- Vertical angle measured from zenith

- Horizontal angle measured to the right

- Recording – Set code – Skip

- Recording – Set target height – Skip

- Parameters for data transfer:

- Baud rate: 2400

- Parity bit: No

After you have finished the setting process, restart the instrument and As-Built (AutoCAD).

If you want to release a measurement at the instrument, go to the recording mode and select the recording mask 'S V H. The point number will be ignored.

7.4.35 Instruments of type Nikon NPL 800 / DTM 800 series / NPL 302

In order to use Nikon total stations of the series NPL 800, DTM 800 and NPL 302 with As-Built, following settings need to be checked and to be adjusted if necessary.

- Communication parameters: Button FNC – 5 Settings – 6 System – 3 Data transfer:

Baud Rate:	2400
Parity bit:	no
Data length:	7 bits
Stop Bits:	1

- Switch to the measurement menu FNC – 3 Total Stn for a measurement. The measurement can only be released from As-Built on your notebook.

- At a NPL 302 additionally the File format = SET has to be selected.

Please note that a prism constant (length eccentricity) set at the instrument is already considered within the measured distance. Therefore you only have to set the prism constant either at the instrument **or** in As-Built never in both places.

7.4.36   Instruments of the Nikon DTM 302 type

The DTM 332, DTM 352 and DTM 362 belong to this series.

In order to be able to use the instruments of this series you will have to check and change, where applicable, the described settings at the instrument.

Settings in As-Built

In As-Built under ➜ "**Select instrument"** choose the instrument **Nikon**.

Settings at the instrument:

Open the main menu. In order to do so push the MENU button. The following settings are relevant for the work with As-Built:

- Settings – Angle:
    - VA zero = zenith
    - HA = azimuth
- Settings – Communications:
    - ExtComm=SET
    - Baud=2400
    - Length=7
    - Parity=None
    - Stop=1
- Settings – Unit:
    - Angle = GON
    - Distance=meter
- Settings – Recording
    - Data Rec = COM
    - Store DB = RAW
- 1sec-Key – [MSR] – MSR1or MSR2 (keyboard allocation for MSR1 und MSR2[7]):
    - Target: Prism / Sheet (prism or reflecting tapes)
    - Const: addition constant of the used prism
    - Mode=precise/normal ("Precise" has a more exact distance measurement but takes longer)
    - AVE=1
    - Rec mode: ALL (measure and register) / Confirm (measure and register, only something else will be shown on the screen) / MSR only (only measure, register with extra push on the ENT button).

---

[7] You may assign functions to many keys yourself. We only describe the allocations for the MSR1 and MSR2 keys (measurement function). The allocation of the other keys can be done according to your personal needs.

<u>Recommendation:</u>

Allocate MSR1 with Rec mode= ALL → Measure and send values to As-Built with the push of one button.

Allocate MSR2 with RecMode=MSRonly→ separated distance measurement and data transfer The angle can be turned in afterwards.

7.4.37 Instruments of type Nikon Nivo M

In order to use Nikon total stations of the series Nivo M with As-Built, following settings need to be checked and to be adjusted if necessary.

Note: If you want to work wireless, please select the connection type "Bluetooth" in As-Built and read **chapter 11.2** for detailed instructions on how to setup a **Bluetooth** connection for you specific operating system.

To open the settings, please press the MENU key ion the instrument key pad. Go through the following submenus. The settings has to be adjusted only once, they are saved in the instrument.

Settings that are not mentioned in the following list are irrelevant for the work with As-Built.

- **3 Settings – 1 Angle:**   VA zero: Zenith

  HA: Azimuth

- **3 Settings – 5 Comm.:**   Ext.Comm: SET

  Port: Bluetooth (for wireless communication)

  Serial (for cable communication)

To check the communication settings, please switch this setting to „Serial". After this the parameters are shown below. After checking them, switch back to "Bluetooth", if you want to work wireless.

  – Baud: 2400

  – Lengths: 7

  – Parity: none

  – Stop bit: 1

- **3 Settings – 7 Unit:**   Angle: Gon

  Distance: Metre

- **3 Settings – 8 Rec:**   Store DB: RAW

  Data Rec: COM

Leave the menu with ESC (multiple) until the basic measurement display appears. (identifiable at the „DSP" sign in the upper left corner). During all of the measurement the basis measurement display stays open. If you switch on the instrument anew, this display appears automatically after the electronic level display

Further important settings:

- **Switch on the visible laser dot:** press key ⚙✳ choose point 2 ✚•••• with the cursor, press REC/ENT to switch to ✳━

- **Configure the measurement keys MSR 1 and MSR 2:** Press the particular key for 1 sec., The measurement mode settings appear:
  - Aiming type: „N-Prism" or „Prism"
  - constant: additive constant for prism measurements, for „N-prism" set it to 0 (zero)

- Mode: „Normal" oder „Precise"
- AVE: number of measurements, that are carried out at pressing the MSR key. With more than one measurements the mean value will be calculated. If set to 0 (zero) the instrument measures permanent until the MSR key is pressed agaim.
- Rec-Mode: register mode for the measurement values.

> ALL (With the MSR key a measurement is carried out and the values are saved.)

> MSRONLY und CONFIRM (With MSR only a measurement is carried out. To save the value press REC/ENT separately.

Measurement

Start the measurement with MSR1 or MSR2 or directly from the As-Built measurement box.

7.4.38   Instruments of type Pentax R-100 and R-300 series

As-Built supports following series: R-115 / R125 / R-135 /R –115N / R-125N / R 135 as well as R-322NX / R-323NX and R-325NX. Following settings have to be set to guarantee proper function:

Menu item 007(Menu) - 801 (SetupCOM) → Communication parameters at the instrument:

- − Baud Rate: 1200
- − Parity :   None
- − Data length: 8 bits
- − Stop Bits: 1
- − XON/XOFF: off
- − Signal Control: on
- − Through Command: A (Output Angle/ Distance on interface)

The communication parameters in As-Built have to be checked with those at the instrument. In As-Built make the following setting for the flow control:

- − Flow control: Hardware (HW)

7.4.39 Instruments of the Pentax series R-400 type

So this connection can be used with As-Built the following settings have to be executed:

1. Switch on instrument
2. Key combination "Lamp" + "ESC" opens "Help" menu
3. There the necessary settings will be executed:
   a. V-angle type: **Z.0** (Vertical angle measurements will be carried out from the zenith.)
   b. Distance unit: **m** (Important since in As-Built only m is available.)
   c. Angle unit **GRD** (Important since in As-Built only GRD is available.)
   d. COM settings
      - Baud rate: **1200, 2400, 4800, 9600**(All are possible, but setting has to be identical to the baud rate within As-Built though. Important: When being used with the RS232 Bluetooth adapter: select 9600).
      - Parity: **None (or Off)** (Both is possible, but setting has to be identical with the parity set in As-Built.)
      - Data bits: 8
      - Stop bits: **1**
      - Xon/Xoff: **off**
      - Signal Control: **on**
      - Throughout Command: **a**

The particular settings can be set with the arrow keys and they may be changed with F5 SELECT. To confirm the individual settings and the respective sub menu press the ENTER key (or if available F5 ACCEPT)

7.4.40 Instruments of the Flexijet 3D type

Flexijet 3D is a mobile measurement sensor for the determination of 3D coordinates. It consists of a laser distance meter (Leica Disto), which is mounted to a turning and swivel unit. The laser may be moved to the target point to be measured manually as well as with the help of the built-in servomotors. The control of the motor activity happens with a sensitive remote control, with which the target point can be approached exactly and measured even over large distances.

The initiation of the measurement happens contact-free at the device, per keystroke at the laptop or via the remote control. The measurement value will be transferred to the measurement software wireless, via Bluetooth from the measurement device to the laptop

Setting up the system

The following settings and actions have only
to be done when using the system for the very first time.

1) Installation of the Flexijet controller software:

- The setup can be downloaded from
  http://service.faro3dsoftware.com/supportfiles/:

  Data transfer using TachyCAD

  BlueTool.exe ... ... 0.3 MB,

  Flexijet_1_0_160_2_for_TachyCAD_setup.exe ... ... 4.19 MB,

  KensingtonBT_driver.zip ... ... 60.12 MB,

  Reserverierte_COM_Ports_freigeben.zip ... ... 0.07 MB

  Execute the *Flexijet_1_0_160_2_for_As-Built_setup.exe* file and follow the instructions of the setup wizard.

- The license of the Flexijet control software has t*o be activated now. In order to do so cop*y the license file (XX-XX-XX-XX-XX-XX-XX.*sw2) into the Installdirectory of the Fl*exijet control software (e.g. C:\\program files\FlexijetControllerSoftware). If you do not have a license file yet, please contact your As-Built/Flexijet partner in order to receive one.

2) Make sure your computer has a working Bluetooth connection. If necessary you can upgrade your computer with the help of an USB flash drive.

3) Attach the USB wireless part of the Flexijet remote control to your computer. No extra driver has to be installed, Windows will recognize the device automatically. Turn on the remote control (any key, until the red light turns on).

4) Turn on the Flexijet (large button at the side).

5) Turn on the Leica Disto (red ON button). A grey Bluetooth symbol has to be visible within the display. In order to turn this on push the Bluetooth button at the Disto if necessary.

6) Start the Bluetooth software on your computer and initiate the device search. According to the Bluetooth driver this function is called "New Connection" or "Add Device" or something similar. Both devices (Flexijet and Disto) should be found from the computer. Set up both devices, one after the other, by following the instructions of the Bluetooth software. You will be asked to enter a key word for authentication reasons each time. Enter the following key:

   - Leica Disto: 0000

   - Flexijet: 1234

The Bluetooth software finds a "serial service" for both devices and will state a COM port number. You will have to remember those two numbers.
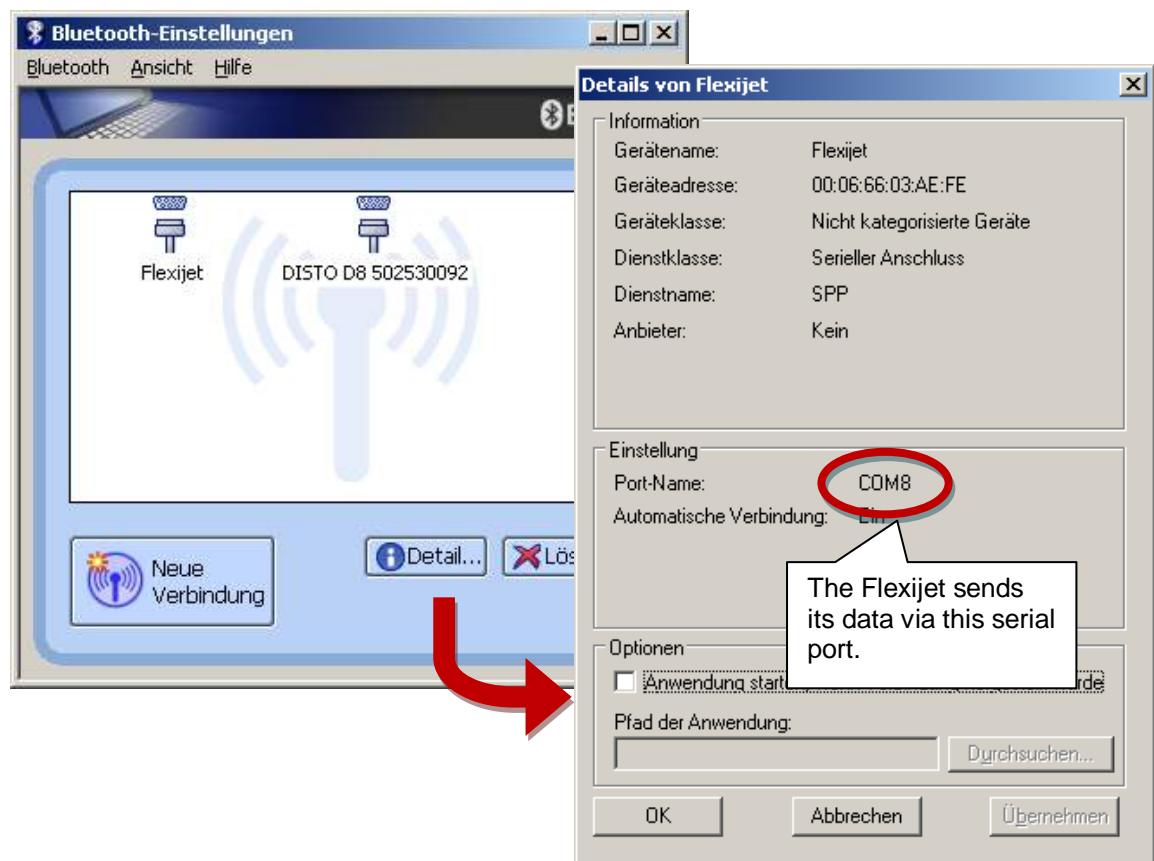
7) Start As-Built and call up the "➔ **Select instrument**" command. Select the Flexijet and move on within the assistant. Following the assistant a window "General Settings" of the Flexijet controller software will open. Here you will enter the following and confirm with OK:

   - Flexijet – Flexijet available   *(activate checkmark)*

   - Flexijet – Comport   *(Enter the COM port number from the Bluetooth software).*

- Flexijet – Slope sensor active *(Checkmark should be active by default.)*

- Laser – Laser present   *(activate checkmark)*

- Laser – Comport   *(Enter the COM port number from the Bluetooth software).*

Now the controller software will open the Bluetooth connections to Flexijet and Disto. You can recognize this because:

- The Bluetooth software shows the status "Connected".

- The Bluetooth symbol at the Disto now shines blue (D8) or it is not blinking anymore but it has a steady light (D3).

Start As-Built. Under "As-Built – Instrument settings – Select instrument" select "Flexijet" from the list and confirm with OK.

Handling

Setup

Set up the Flexijet straight according to visual judgment. A leveling as with other tachymeters is not necessary.

Start

If you want to measure with the Flexijet device the Flexijet controller software has always to be started. This will happen automatically when you start As-Built. But you may start the controller software manually under Start – Programs – "Flexijet Controller Software for As-Built".

The Flexijet symbol will appear within the Windows status bar. When you move the mouse over it, the charging status of the Flexijet battery will be displayed.
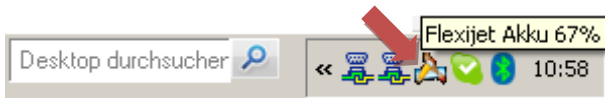
If it has been installed previously, the software establishes the connection to the device automatically when being started.

Stationing

Proceed as described within the As-Built command reference under "→**New/Edit orientation**" The only difference is, that the device will automatically run a calibration upon start of this command. With this it will determine the inclination of its vertical axis, which will be included into every measurement as correction. In order to do so, the device will turn several times. **Please do not interrupt this process!**

For the local orientation ("local orientation") there will be an additional prompt, whether the measurement will take place on solid ground or on a moving ship. In case of "ship" there will be no calibration since the ground is not steady. Instead you will have to perform the "Calibration via three points" in order to determine a vertical reference plane (see below: "Other functions").
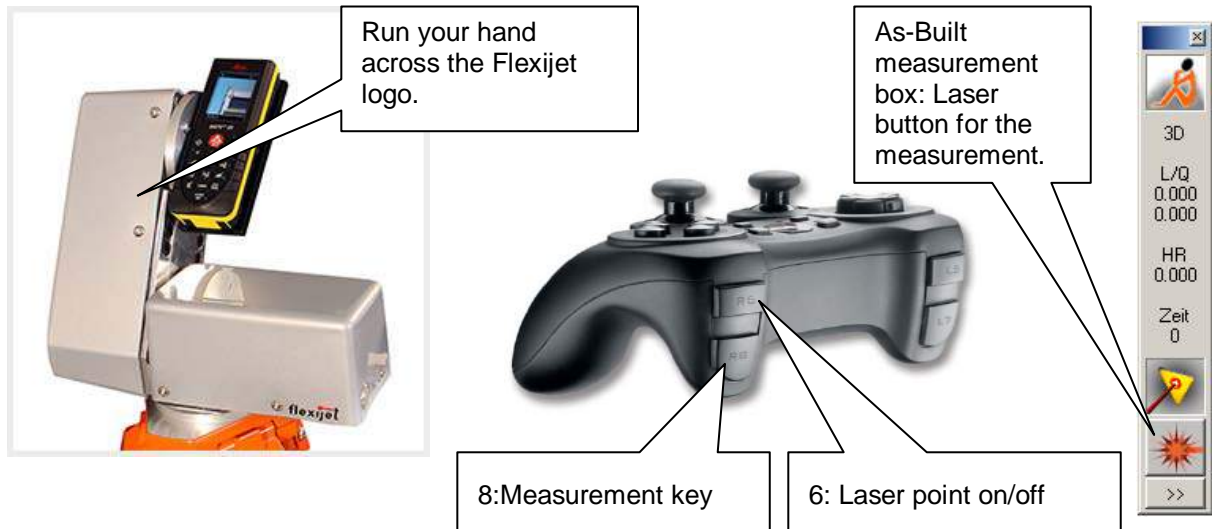
There won't be an inclination calibration for the "Orientation without horizon" as well, since this is also used for measurements on ships.

Measure

A measurement can be initiated in different ways:

- At the Flexijet device via the sensor at the front of the device.

- Within As-Built via the measurement box.

- With the remote control. By default key 8 (back front bottom right) is predefined as measurement key.

Run your hand across the Flexijet logo.

As-Built measurement box: Laser button for the measurement.

8:Measurement key

6: Laser point on/off

Control

Use the remote control in order to approach a measurement point exactly. But you can set the rough direction by turning the instrument manually. The remote control has three speed levels:

Slow (targeted approach)

Very slow (sensitive, exact adjustment, important e.g. for control points)

Fast (rough approach)

The most needed AutoCAD and As-Built commands have been assigned to the other keys. That way it is possible – within a certain scope – not only to telecontrol the measurement device but also AutoCAD and As-Built.

Key configuration standard

- **1 Draw line (AutoCAD: LINE)**

- **2 Draw 3D polyline (AutoCAD: 3DPOLY)**

- **3 Draw point separately (AutoCAD: POINT)**

- **4 Measure outline of wall (As-Built: TCADOUTLINEPLAN)**

- **5 Measure new control point (As-Built: NEWCOPO)**

- **6 Disto laser point On/Off**

- **7 Enter**

- **8 Initiate single measurement (Measurement key)**
- **9 Escape**
- **10 [not assigned]**

You may reconfigure the keys according to your needs. The command named above are available. For configuration see "General settings" further down in this chapter.

Shock sensor

When the device is jolted accidentally, it will be registered by the built-in shock sensor and an according message will be put out:
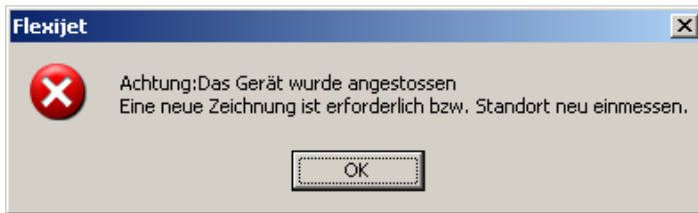
In such a case you will have to define a new orientation. Proceed as if you would really change the station position. Use the As-Built command "New/Edit station".

Other functions

With one click on the Flexijet symbol you will have access to the most important functions and settings.

- **Calibrate inclination sensor: With this function you start the automatic definition of the inclination of the vertical axis. The device will turn several times and the built-in inclination sensor will determine the inclination of the axis. Important note: You should not execute this function during a measurement, since it changes the station definition. It is only to be used when changing the station.**

- **Calibrating via three horizontal points: You will need this function when you are measuring on a moving ship. It is needed for the determination of a horizontal reference plane. The first station will be defined within As-Built with the stationing method " Local orientation". Before you can define this first stationing you will have to execute the calibration via three points. In order to do so measure three points which represent the wanted horizontal plane (e.g. the laser line of a ship).**

- **Cancel**: Stop process (e.g. the calibration of the inclination sensor).

- **Speech output**: Turning speech output on and off. In order to setup the speech output please refer to the separate manual.

- **Status…: A window will open with which you may test the functional efficiency of the device.**

Flexijet Status

Punkt anfahren
X: 0   Y: 0   Z: 0   Los

Close dialogue.

Approach point with the coordinates shown on the right (instrument coordinate system).

Winkel:
AZM:359.945° ALT:359.956°

Info
Standort: Welt
Akku:

Battery status
Is also a possibility to test if the connection to the Flexijet device is connected. If no connection has been established the bar will be empty.

Befehl

Time	Message
13:08.23	Referenzierung abgeschlossen.
13:25.53	*Abbruch*
13:26.04	Referenzierung über 3 horizontale Punkte
13:26.04	Erster Punkt:
13:26.06	*Abbruch*
13:27.04	Neigungssensor kalibrieren...
13:27.07	*Abbruch*

Protocol of the last interactions with the Flexijet instrument.

Ausführen   Abbruch

Info
Info über Flexijet

Information about the software

- **General settings:**

Entry	Standard value	Meaning
System settings		
Language	German	set language
Interface	As-Built	Controller software for As-Built
Flexijet		
Flexijet available:	☑	If checked, the software will try to establish a connection to the Flexijet device when started. Do only uncheck when As-Built is temporarily to be used without measurement instrument.
Comport:		Serial port via which the Flexijet device communicates. The change requires a reboot of the controller software.
Slope sensor active	☑	The device determines the inclination of the vertical axis. The determined correction value will be attached calculational to every measurement. It should always be checked.
Standby time of the engines	20	In [sec]. The motors will automatically turn off after this time, when they were not used. This saves energy.
shock sensor	25	The sensor registers and reports jolts. The

sensitivity		smaller the value, the more sensitive the reaction.
Sensor melody	sensor.wav	This sound will be played when a measurement will be initiated at the device.
Goto construction points very precisely	☑	Checked: Measurement points will be approached very precise during the stake out → exact but slower. Recommended setting. Unchecked: Measurement points will be approached less precise → less precise but fast.
Logitech Cordless RumblePad		
Observation axis data		A dialogue will open where you can see the axis data of the remote control.
Blind area	15	Axis values below this range will be interpreted as zero.
Maximum value	120	Maximum axis value, which the Logitech RumblePad reaches. You only have to change this if you use other joysticks.
Swap up and down	☑	Switch up and down movement of the joystick.
Keyboard layout		
Joystick button 1..10	[see above under "Control"]	Setting, which key of the remote control is to execute which function. Next to "Measure" and "Laser ON/OFF" there are a number of often used AutoCAD and As-Built commands.
Laser		
Laser present		If checked, the software will try to establish a connection to the Disto when started. Do only uncheck when As-Built is temporarily to be used without measurement instrument.
Comport		Serial port via which the Disto communicates. The change requires a reboot of the controller software.
Off time in seconds		In [sec]. The Disto will turn off after this time if no action has taken place. This saves energy.

8 Command Reference (As-Built Total Station Interface)

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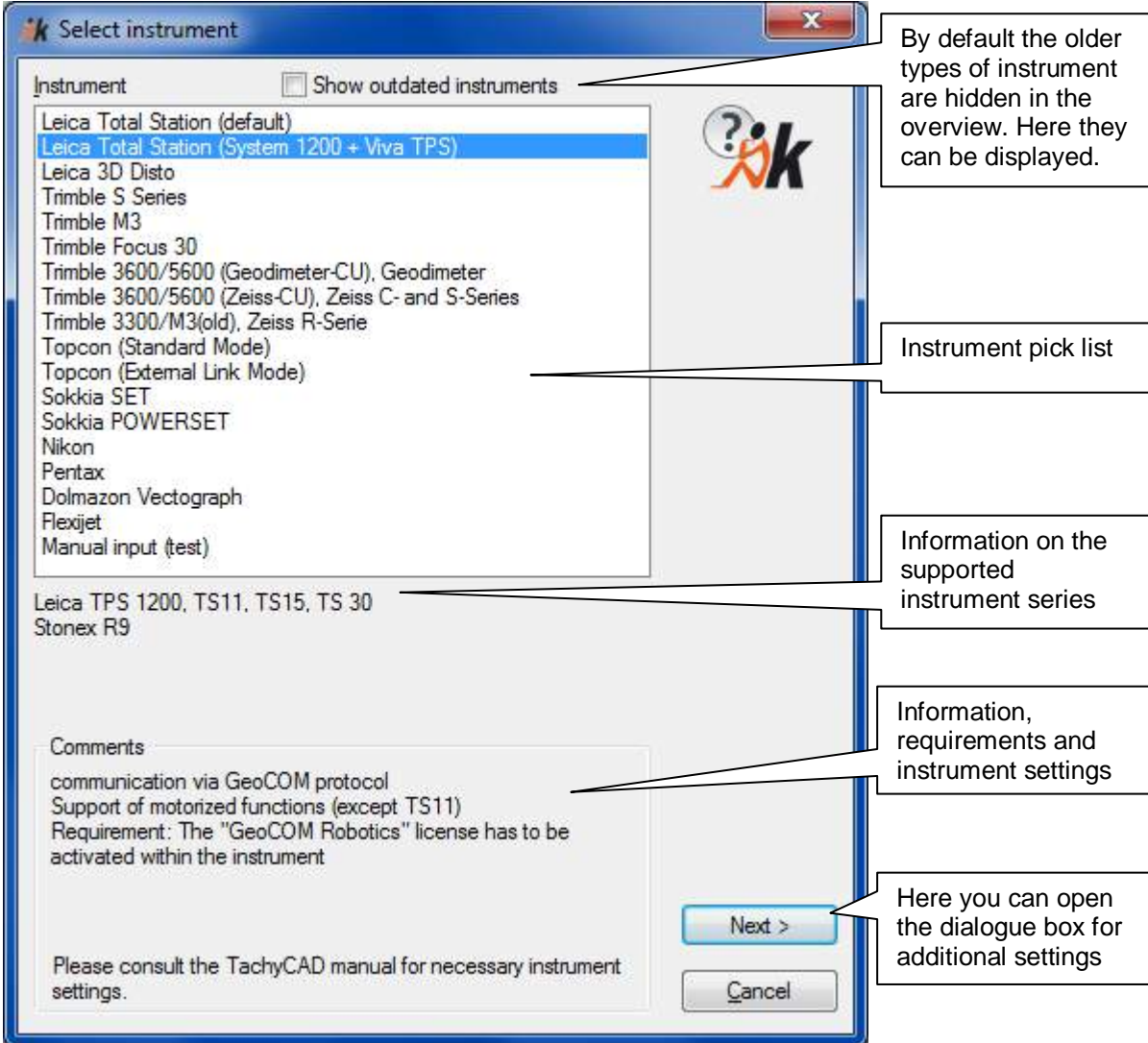
Specifies on which of the various ribbon tabs of As-Built for AutoCAD the command can be found.

8.2 Commands

Total station – Select instrument

	Command:	TCADINSTRUM
	Ribbon tab	As-Built TS Control

With the command 'tcadinstrum' you can select the connected instrument. Most of the commonly available instruments are supported.

By default the older types of instrument are hidden in the overview. Here they can be displayed.

Instrument pick list

Information on the supported instrument series

Information, requirements and instrument settings

Here you can open the dialogue box for additional settings

By clicking "Next" you are taken to the additional settings. Depending on the selected connection type the dialog looks as follows:
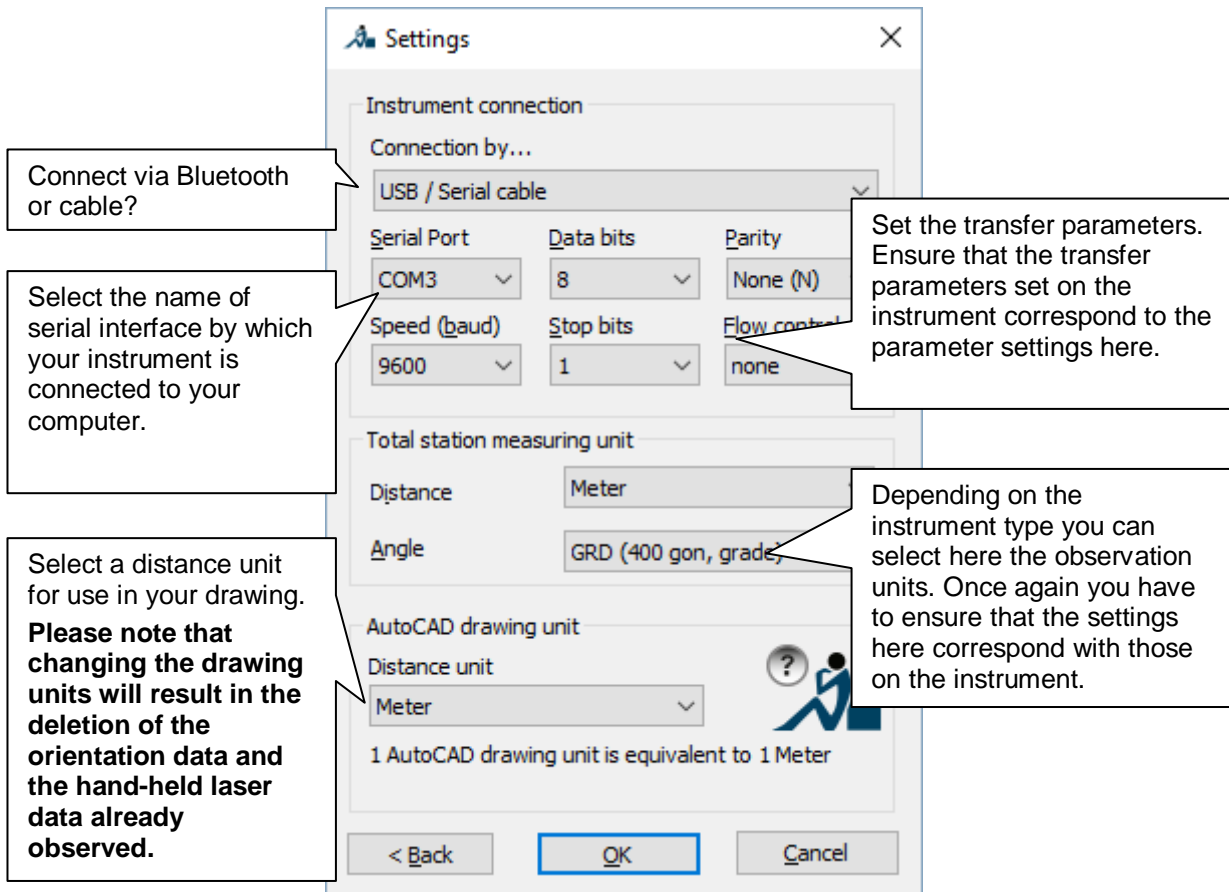
Bluetooth

Connect via Bluetooth or cable?

These are the instruments found in the Bluetooth environment.

Settings

Instrument connection

Connection by...

Bluetooth

Discovered devices:

DISTO 5080220167
RS232 Adapter for TachyCAD
TS3011179
DISTO D3 900820041
TS06p_1389595

Total station measuring unit

Distance   Meter

Angle   GRD (400 gon, grade)

AutoCAD drawing unit

Distance unit

Meter

1 AutoCAD drawing unit is equivalent to 1 Meter

< Back   OK   Cancel

Select a distance unit for use in your drawing. **Please note that changing the drawing units will result in the deletion of the orientation data and the hand-held laser data already observed.**

Depending on the instrument type you can select here the observation units. Once again you have to ensure that the settings here correspond with those on the instrument.

Close the settings dialogue box by clicking the **OK** button.

You will find instructions for setting up the Bluetooth environment of your operating system in chapter **11.2 Bluetooth Connection**.

Cable

For the majority of instrument types the dialog will look the screen capture below. Should the dialogue box differ, then you will find additional information to the settings in the section "Advice relating to supported instruments".

Connect via Bluetooth or cable?

Select the name of serial interface by which your instrument is connected to your computer.

Set the transfer parameters. Ensure that the transfer parameters set on the instrument correspond to the parameter settings here.

Select a distance unit for use in your drawing. **Please note that changing the drawing units will result in the deletion of the orientation data and the hand-held laser data already observed.**

Depending on the instrument type you can select here the observation units. Once again you have to ensure that the settings here correspond with those on the instrument.

Close the settings dialogue box by clicking the **OK** button.

You have to ensure that the transfer parameters set on the instrument correspond to those parameters set here. The settings can be entered here or directly on the instrument.

Detailed information on the total stations of the different manufacturers is available in section **Advice on supported instruments** in this manual.

Total station – Total station settings
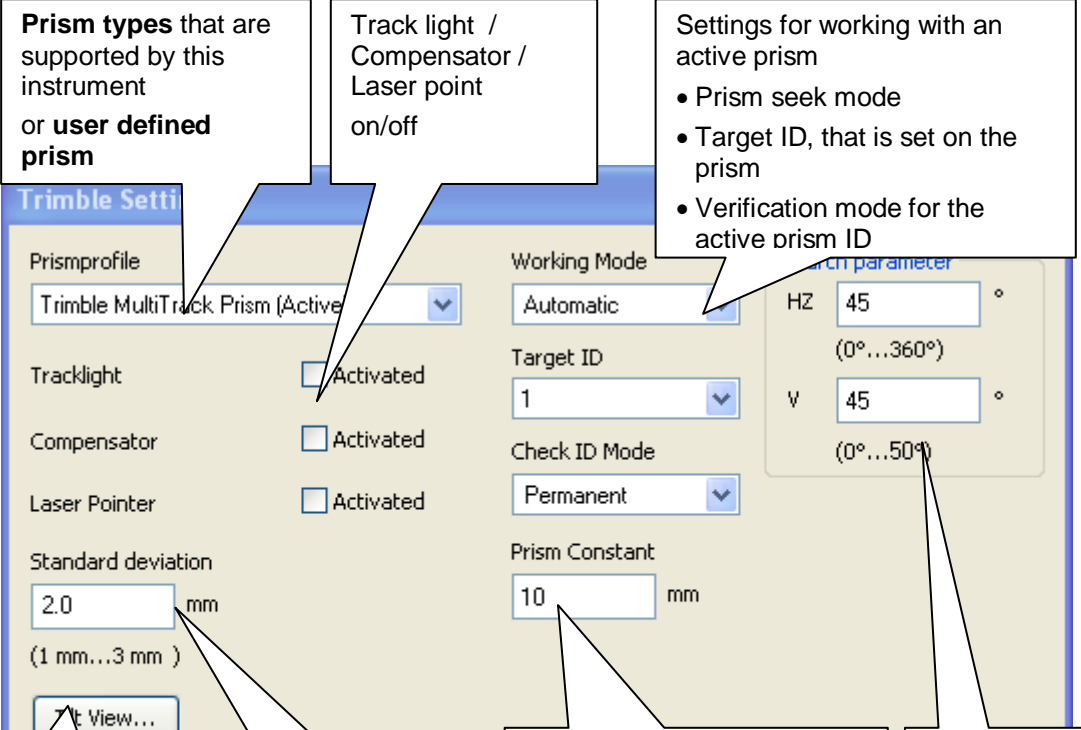
	Command:	TCTSSETTINGS
	Ribbon tab	As-Built TS Control

This command enables the total station to be remotely configured directly from As-Built for AutoCAD.

So far only Trimble total stations are supported (through Trimble Precision SDK). These devices can only be configured in this way, since they are in a kind of remote mode whilst working with As-Built for AutoCAD and therefore do not accept keyboard entries.

The command only works when a connection to the instrument has been established.

The command covers the following important observation settings. The settings that will be offered will depend upon the type of instrument connected.

Prism types that are supported by this instrument

or **user defined prism**

Track light / Compensator / Laser point on/off

Settings for working with an active prism
- Prism seek mode
- Target ID, that is set on the prism
- Verification mode for the active prism ID

Starts the digital bubble view

Standard deviation for distance measurement settings

Addition constant for the selected prism
→for user defined prisms you can set this yourself.

Search scope for locating the prism

Total station – Show total station bubble

	Command:	TCTSTILTVIEW
	Ribbon tab	As-Built TS Control

This command presents the user with a digital display of the bubble of the total station. It is required that As-Built for AutoCAD is connected with the instrument. So far only Trimble total stations are supported (via Trimble Precision SDK).
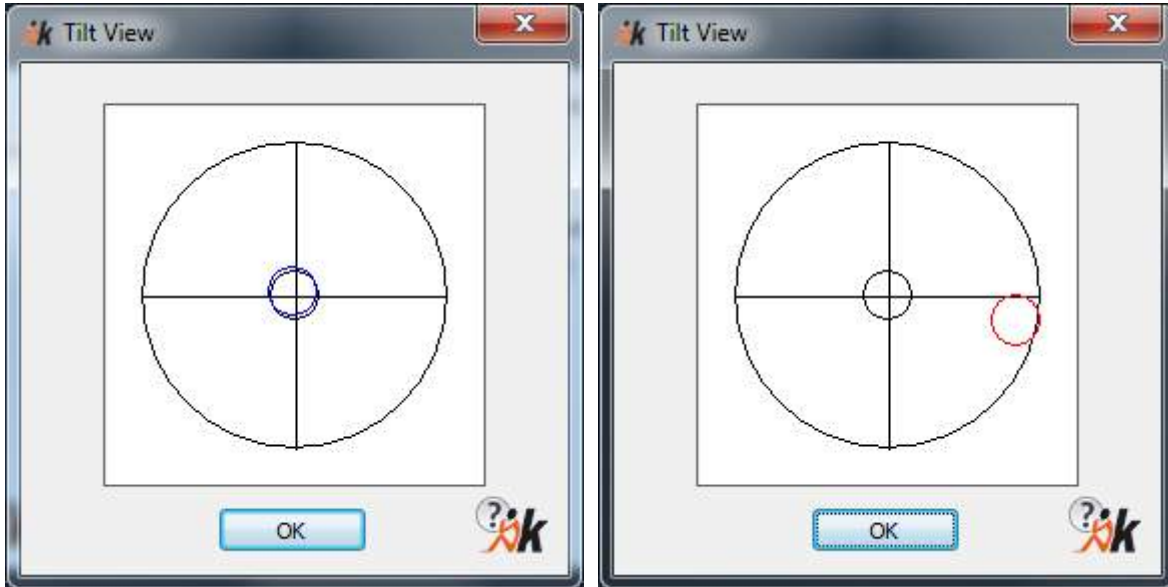
Total station – Measure bar configuration

	Command:	MEASBAR
	Ribbon tab	As-Built TS Control

With the command *'measbar'* you can determine whether As-Built for AutoCAD can receive data from the instrument.
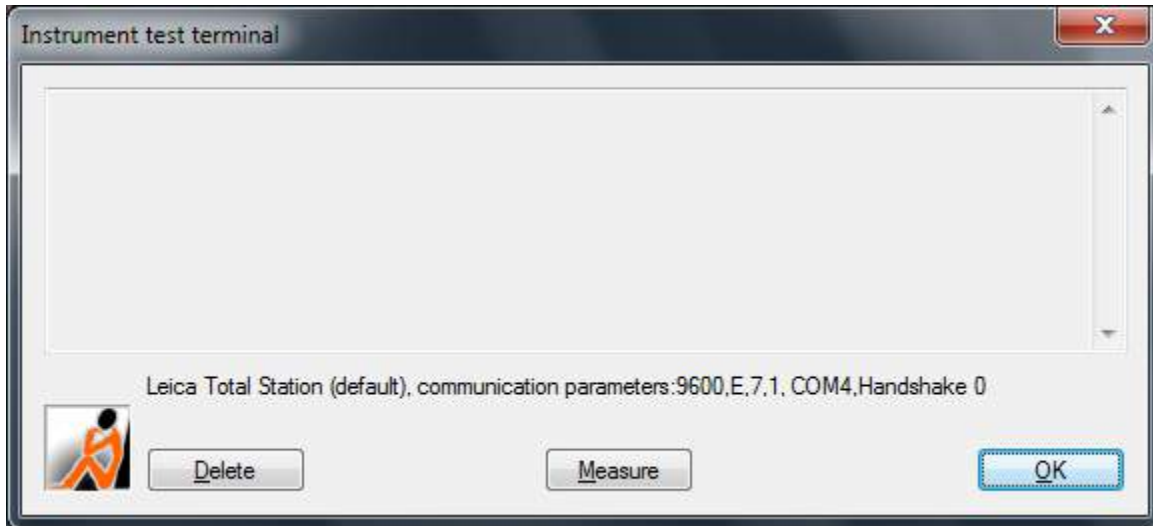
If this check box is ticked TachyCAD will receive data from the instrument.

If ticked this check box hides the measurement box. Data can still be received from the instrument.

Total station – Interface monitoring

	Command:	'TCADTERM
	Ribbon tab	As-Built TS Control

You should use this command to check the communication between As-Built for AutoCAD and your total station. It is possible to initiate an observation from within As-Built for AutoCAD or on the instrument. If this is not the case and no data is received in the interface monitoring dialogue box, then check the communication parameters in the As-Built for AutoCAD settings and those on the total station.

Instrument test terminal

Leica Total Station (default), communication parameters:9600,E,7,1, COM4,Handshake 0

Delete   Measure   OK

Total station – Establish a connection

	Command:	TCCOMON
	Ribbon tab	As-Built TS Control

This command opens the interface and thereby allows the communication between the total station and As-Built for AutoCAD. Also see →Total station – Break the connection**.**

Total station – Break the connection

	Command:	TCCOMOFF
	Ribbon tab	As-Built TS Control

This command closes the interface and disconnects the total station from As-Built for AutoCAD. If during an AutoCAD session there are changes to the interface connection (e.g. the connection cable is removed and then replaced) it is possible that no further observations can be received in AutoCAD because the interface buffer is full. In such a case it could help to close the interface (break the connection) and then open it again (→establish a connection).

In certain situations it is recommended to follow this procedure, such as when changing the instrument station where the data cable is unplugged from the notebook when the equipment is carried to the next station:

- Command 'break the connection'
- Change the instrument station
- Command 'establish a connection'

Total Station – One-man station – Check battery status of the instrument

	Command:	TCTOTALSTATIONBATTERYSTATUS
	Ribbon tab	As-Built TS Control

This command sends a query for the battery status to the total station. The current battery status will then be displayed within the AutoCAD command line.

This function is useful when you use the prism tracking in a one-man-operation. That way you can check the status of the total station battery from afar.

The following instruments are supported:

- Leica TPS 1200 series and Viva TS 15 series
- Topcon External Link mode (GPT 8000/9000, IS,…)
- Trimble S series, VX, M3
- Spectra Precision Focus 30

Total Station – One-man station – Change face

	Command:	TCTOTALSTATIONCHANGEFACE
	Ribbon tab	As-Built TS Control

This command causes the total station to execute an automatic change of the telescope position.

The following instrument types are supported:

- Leica TPS 1200 series and Viva TS 15 series

- Topcon External Link mode (GPT 8000/9000, IS,…)

- Trimble S series, VX, M3

- Spectra Precision Focus 30

Total Station – One-man station – Start prism search (left)

	Command:	TCTOTALSTATIONSEARCHPRISMLEFT
	Ribbon tab	As-Built TS Control

This command causes the total station to start a prism search.

Leica:

The instrument will start the search counterclockwise.

If the total station has the PowerSearch option, this option will be used for the search. If the instrument does not have PowerSearch, the "conventional" prism search will be started, but that search will take a lot more time.

Topcon:

There is no difference between right and left. The prism search will just be started.

If a prism has been found, the prism tracking will be started automatically. With this the telescope will track the prism motorised when it is moved.

The following instrument types are supported:

- Leica TPS 1200 series and Viva TS 15 series
- Topcon External Link mode (GPT 8000/9000, IS,…)
- Trimble S series, VX, M3
- Spectra Precision Focus 30

Total Station – One-man station – Start prism search (right)

	Command:	TCTOTALSTATIONSEARCHPRISMRIGHT
	Ribbon tab	As-Built TS Control

This command causes the total station to start a prism search.

Leica:

The instrument will start the search clockwise.

If the tachymeter has the PowerSearch option, it will be used for the search. If the instrument does not have PowerSearch, the "conventional" prism search will be started, but that takes a lot more time.

Topcon:

There is no difference between right and left. The prism search will just be started.

If a prism is found, the prism tracking will automatically be activated. With this the telescope will be moved motor-driven upon movement of the prism.

The following instrument types are supported:

- Leica TPS 1200 series and Viva TS 15 series
- Topcon External Link mode (GPT 8000/9000, IS,…)
- Trimble S series, VX, M3
- Spectra Precision Focus 30

Total Station – One-man station – Activate prism tracking

	Command:	TCTOTALSTATIONPRISMTRACKON
	Ribbon tab	As-Built TS Control

This command causes the total station to turn on the tracking mode.

If a prism is within the field of view of the telescope, it will be tracked with the help of the servomotor when it is moved.

If no prism is within the field of view, the prism tracking can not be started. In that case perform a search → **Instrument measurement** → **one man station – Start prism search (right/left).**

If the movement of the prism is too fast, the target can be lost. Make sure you don't move the prism faster as stated within the technical specifications.

You can detect the loss of the prism at the signal light of the tachymeter[8]. By means of this light it is possible to roughly move back into the target beam[9]. If the prism appears back in the field of view of the telescope the prism tracking will be continued.

The following instrument types are supported:

- Leica TPS 1200 series and Viva TS 15 series
- Topcon External Link mode (GPT 8000/9000, IS,…)
- Trimble S series, VX, M3
- Spectra Precision Focus 30

---

[8] **Leica:** The EGL (target guide light) starts blinking and the tachymeter starts beeping. Depending on whether you are too far left or right of the target beam you will see a yellow or a red blinking light. If you see both lights you are right in the middle of the target beam.

Topcon: When the instrument is still searching the tracking indicator light is constantly on upon loss of the prism. It will blink slowly when the prism cannot be found.

[9] Depending on whether you are standing to far to the left or right of the target beam you will see a yellow or red flashing light. If you see both lights you are directly within the target beam.

Total Station – One-man station – Deactivate prism tracking

	Command:	TCTOTALSTATIONPRISMTRACKOFF
	Ribbon tab	As-Built TS Control

This command causes the total station to stop the tracking mode.

Then the prism will not be tracked any more.

See → **Instrument measurement** → **one man station – Enable prism tracking**.

Total Station - Single measurement (Extended Measurement box)

	Command:	'MMT
	Ribbon tab	As-Built TS Control

With this command you call up the "**Extended Measurement box**", where the parameters for the coming measurements will be defined.

By default, the extended measurement box will be opened via the **>>** button of the small measurement box.

Alternatively you may open the extended measurement box via the "Single measurement" command from the command line or the menu.

There are three different measurement modes: point measurement, plane intersection and horizontal eccentricity. For every mode there are separate setting options. The tab of the current measurement mode is always on top.

Displays the currently **selected instrument**.

Permanent: all following measurements are run with this value.

3D: The measurement command returns 3D (X,Y,Z) coordinates

2D: The measurement command returns 2D (X,Y) coordinates.

Delay time: The measurement will not be triggered right away but after the set seconds have passed (like a" self-timer").

Eccentricity: statement of a longitudinal or lateral deflection in case the point to be measured cannot be aimed at directly.

Initiation of a measurement. Depending on the selected mode, direction and distance or only the direction from the instrument to the target will be checked.

Closes the box without measurement and saves all settings

With every measurement a numbered **protocol point** is inserted, which records the measurement data (Values, coordinates, orientation, optional additional information).

Enter a **tolerance in z-direction** for the exact measurement of intersections along a defined intersection plane.

The command is transparent i.e. it can be used during the processing of another command. The apostrophe at the beginning of the command leads to a transparent implementation.

Settings

According to the set measurement mode different settings will be displayed. They will be explained further down in the "Measurement modes" paragraph.

Point

If you set "3D", As-Built for AutoCAD returns 3D coordinates (X,Y,Z) to the UCS during the measurement. If "2D" is selected, 2D coordinates will be returned. The latter option is useful if you want to employ certain AutoCAD commands in connection with As-Built for AutoCAD e.g. 2D polylines.

Z-variation

This is where you state the tolerance range for the Z direction (UCS) within which a measurement is valid. This option is for the exact measurement of sections along a defined section plane.

An example: You want to measure the wall line of a room in the exact height of 1m. The floor's height is level zero. Enter "Min Z = 0.95 Max Z = 1.05" and check the "Only accept points within Z-corridor of current UCS" checkbox. For the following measurements As-Built for AutoCAD will check if the measured point is, regarding the height, within this 10-cm-corridor. If not, the point will be ignored and the computer will return an acoustic warning.

When you activate the Z-variation the measurement mode symbol within the measurement box will change:

Measurement point

If the button "Create observation point" is activated, a protocol point is inserted into the drawing with every measurement. It is a block that is inserted on the template layer "Observation points". The protocol point has following attributes:

- Number
- Measurement values (HZ, V, S, mm, hr)
- Instrument station (Number)
- Calculated coordinates (X, Y, Z)
- Additional information (optional)

Except for the number and the additional information, these attributes are, by standard, invisible in order to avoid too much text in the drawing. However, they can also be displayed. You can as well export the attributes to a text file.

The number of the observation point and the additional information can be determined in the entry window. The value will automatically be counted up during

the following measurements. In the As-Built for AutoCAD settings you may define which way the observation points are numbered.

Measurement mode: Full measurement

You will mostly work with the measurement mode full measurement. You aim at a point with your instrument and release a **full measurement**. The horizontal and vertical direction as well as the distance to the target, with the help of the laser, are measured. Out of these values a 3D coordinate of the target is calculated. You activate this measurement mode by clicking the respective register. The current measurement mode is displayed by a symbol in the small measurement box.

Symbol for activated measurement mode 'Full measurement'

Height of reflector

The height of the reflector is subtracted from the height value of the targeted point. Thus, obstructed points can be measured indirectly with a support tool (e.g. meter pole).

The entered reflector height will be kept until the next alteration through the activation of the checkbox "permanent". If this option is deactivated, the reflector height will be set to the previous value after the measurement.

Note: As-Built for AutoCAD ignores the insertion of a reflector height directly at the instrument since As-Built for AutoCAD only considers pure measurement values.

Eccentricity

If a point cannot be measured directly, a cross or longitudinal deviation can be entered. Above all, this is important for measurements with reflector.
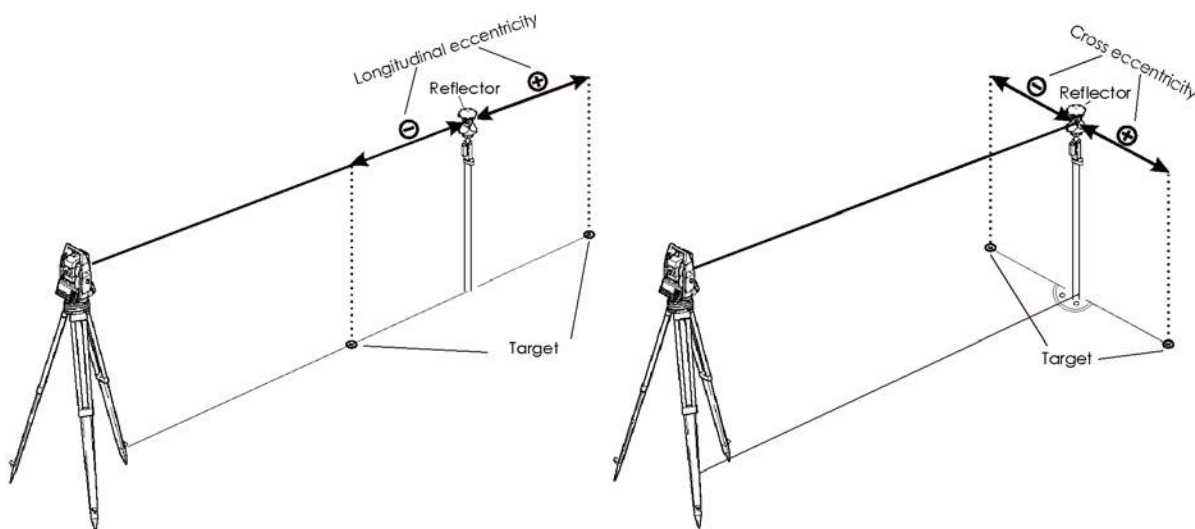
Longitudinal (length) eccentricity   **Cross eccentricity**

The point to be aimed at is located either

- in direct extension before or after the assumed point (longitudinal eccentricity) or
- right angled, left or right to the assumed point (cross eccentricity)

Depending on the position of the target point, left or right (cross eccentricity) or before or after (longitudinal eccentricity) the assumed point, a positive or negative value must be entered (see sketch).

Delay

Via this field you can determine that the measurement is released with a certain delay. The measurement is then released with the **fire** button.

Measurement mode: Plane intersection

You activate this measurement mode by clicking on the respective tab within the large measurement box. Within As-Built for AutoCAD's small measurement box the current measurement mode will be displayed by a symbol.

The measurement mode plane intersection proves useful for several applications e.g. the survey of a facade. You aim at a point with the instrument and release an **angle observation**. Thereby, only the horizontal and vertical direction to the target are measured. Before the measurement you have determined an intersection plane in your drawing. A measurement beam is defined by the measurement. This beam then intersects with the plane. A 3D coordinate is generated.

Display of the coordinates of the current **intersection plane**.

3D: The measurement command returns 3D (X,Y,Z) coordinates
2D: The measurement command returns 2D (X,Y) coordinates

Symbol for activated measurement mode "Plane intersection"

Selection of intersection plane by entering either a kubit plane or using the XY plane of the current UCS.

Visualisation of the intersection plane in the drawing.

With every measurement a numbered **protocol point** is inserted, which records the measurement data (Values, coordinates, orientation, optional additional information).

This method has the advantage that no distance measurement to the target needs to be done. If you have to measure many points that are all in one plane (e.g. facade, floor, ceiling), this method is much faster than the full measurement mode. You can also determine points on surfaces that are unsuitable for a laser meter (frosted or matt black, glass, stainless steel or mirrored materials).

Intersection plane

- You can use the XY plane of the current UCS as intersection plane. For this you must have defined a new UCS which corresponds to your measurement plane. You can use the As-Built for AutoCAD command "mucs".

- A construction plane (kubit plane) can also serve as intersection plane. Use the command "Draw plane" or "Fit plane" to create such a plane.

Click "Select new plane" in the dialogue to select the intersection plane. Following query appears in the command line:

Use current UCS or select construction plane [UCS/construction Plane] <UCS>:

Return accepts the XY plane of the UCS as intersection plane and returns to the start dialogue. If you select the construction plane with "p", you will be asked to select the plane.

In the frame "Intersection plane" you can now see the new coordinates of the intersection plane.

With the help of "Show intersection plane", you can visualize the intersection plane in AutoCAD. This is only possible if you have not called up the command Individual measurement as transparent command.

Horizontal eccentricity measurement mode

You will activate this measurement mode by clicking on the respective tab within the large measurement box. The current measurement mode will be displayed within As-Built for AutoCAD's small measurement box by a symbol.

The measurement mode **Horizontal eccentricity** is interesting for measuring points where a direct distance measurement is not possible. An example is the measurement of the center of a pole or tree trunk.

The measurement is split for this measurement mode. At first a measuring point next to the actual target point is being aimed at (target mark, reflector pole) and the distance as well as the vertical direction will be measured. Then the instrument is swiveled in the actual target direction and the horizontal direction will be measured. Following to that As-Built for AutoCAD will combine both measurements and will generate a coordinate.
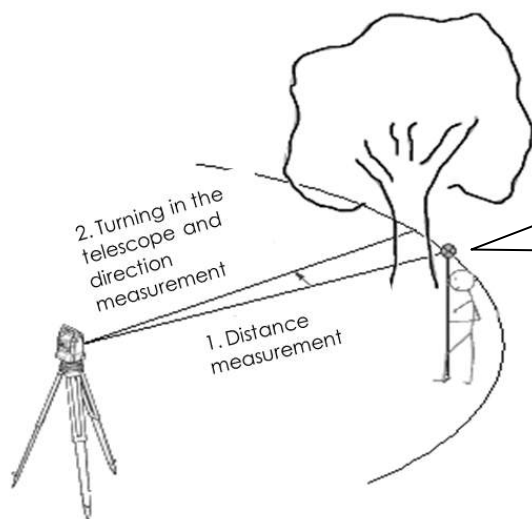
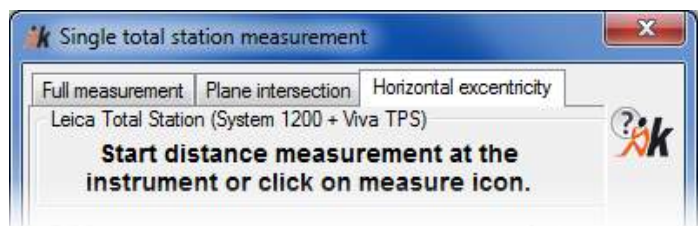Symbol that the "Horizontal eccenter" measurement mode is active.

A point next to the tree is being aimed at with the reflector pole.
Important: The distance has to be the same as to the actual target point!

2. Turning in the telescope and direction measurement

1. Distance measurement
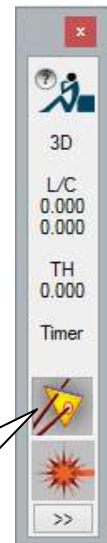
Process:
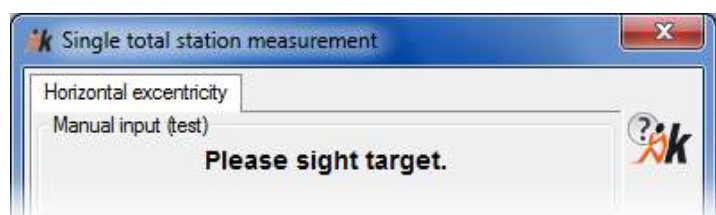
1.) Measurement of distance and vertical angle:

The point, off which the distance and the vertical angle will be taken, will be pointed at. The horizontal direction will not be saved. The measurement will be triggered via ☀ or the measurement key at the instrument.

Leica Total Station (System 1200 + Viva TPS)
Start distance measurement at the instrument or click on measure icon.

2.) Measurement of the correct horizontal direction towards the actual target point.

With the same vertical angle the instrument has to be swiveled to the target point. By pushing the ☀ measurement triggering key the missing

Manual input (test)
Please sight target.

horizontal direction will be measured.

The options in detail:

Information about the currently set instrument

Permanent: All following measurements will be executed with this value.

3D: The measurement command returns 3D (X,Y,Z) coordinates.

2D: The measurement command returns 2D (X,Y) coordinates.

Input of the **tolerance range in Z-direction**.

Triggering of a measurement

According to the set measurement mode the direction and distance to the target point, or only the direction will be polled from the instrument.

A **protocol point with a number** that records the survey data (measurement values, coordinates, optional additional information) will be inserted with every measurement

The Permanent check applies to the Information field: all following measurements provide the measuring point with information for the display.

Closes the box without measurement and saves the settings.

Reflector height

Each time the reflector height will be subtracted from the height value of the aimed at point. By doing so offset points can be measured indirectly with an aid (folding rule) for example

Checking the "permanent" option causes that the entered reflector height will be kept until the next change. If the option is unchecked, the reflector height will be set to the previous value after the measurement.

Hint: The input of a reflector height directly at the instrument will not be regarded by As-Built, since As-Built for AutoCAD only transfers straight measurement values from the instrument delay time.

Via this option you can control that the measurement will not start before a certain time has passed, after it has been triggered from a Notebook/PC. The measurement can be triggered with the respective delay via the As-Built for AutoCAD measure key.

Total Station - Execute continuous measuring

	Command:	'RMMT
	Ribbon tab	As-Built TS Control

Some instruments (e.g. the Breithaupt WMS) do not have keys for entering data. With these instruments, the measurement can only be released in As-Built for AutoCAD. This means that the fireball button in the measurement box must be clicked for every measurement. This procedure proves tiresome, especially for longer measurements. Therefore, we have developed the command 'Rmmt, which allows to process repeated measurements. When this command is called up, the extended measurement box appears (see manual **4.3 The measurement box - interface between AutoCAD and instrument**). Thus, the measurement can be released with Enter. The measurement box is shown after every measurement and you can release any number of following measurements by pressing Enter. If you enter the command in the AutoCAD command line, it is important that you call up the command as transparent command. Just put an apostrophe before the command ('rmmt).
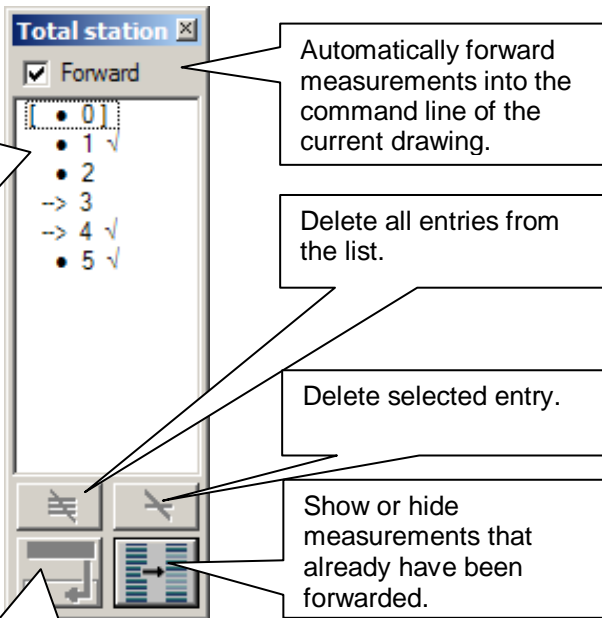
Total Station - Show instrument values

	Command:	TOTALSTATIONMEASUREDVALUES
	Ribbon tab	As-Built TS Control

Using this command all measured values that have been received by As-Built for AutoCAD can be looked at.

If you for instance forgot to execute a command like 'line' or 'Measure outline plan' before initiating the measurement at the total station, you may get apparently lost observation points ex post into the drawing. Thus, the points do not have to be measured again.

Another application is twosome work. While one colleague is doing constructions using the notebook the other one can already proceed with measuring. Those measurements can then be sent into the command line as whole lot of points.

When running the command the following window opens.

List of received measurements

Format:

[..] Forwarded measurement no command was active

.. √ Successfully forwarded measurement

.. Measurement that has not been forwarded.

Measurements:

● Full measurement

→ Plane intersection

Automatically forward measurements into the command line of the current drawing.

Delete all entries from the list.

Delete selected entry.

Show or hide measurements that already have been forwarded.

Forward selected measurements manually into the current drawing.

Remarks

Single measurements can also be forwarded by double clicking the referring list entry.

This command is a transparent command. I can be run during the use of another command (e.g. 'Measure outline plan' or 'Measure segments')

Measurement methods - Points by intersection

	Command:	PTINTERSECT
	Ribbon tab	As-Built TS Control

This command allows the definition of a point using the method of Points by intersection.

The point to be determined is aimed at from two or more orientations, and an angle measurement (without distance measurement) is taken from each station. The point of intersection of the beams is calculated.

After the command has been called up, following dialogue appears:

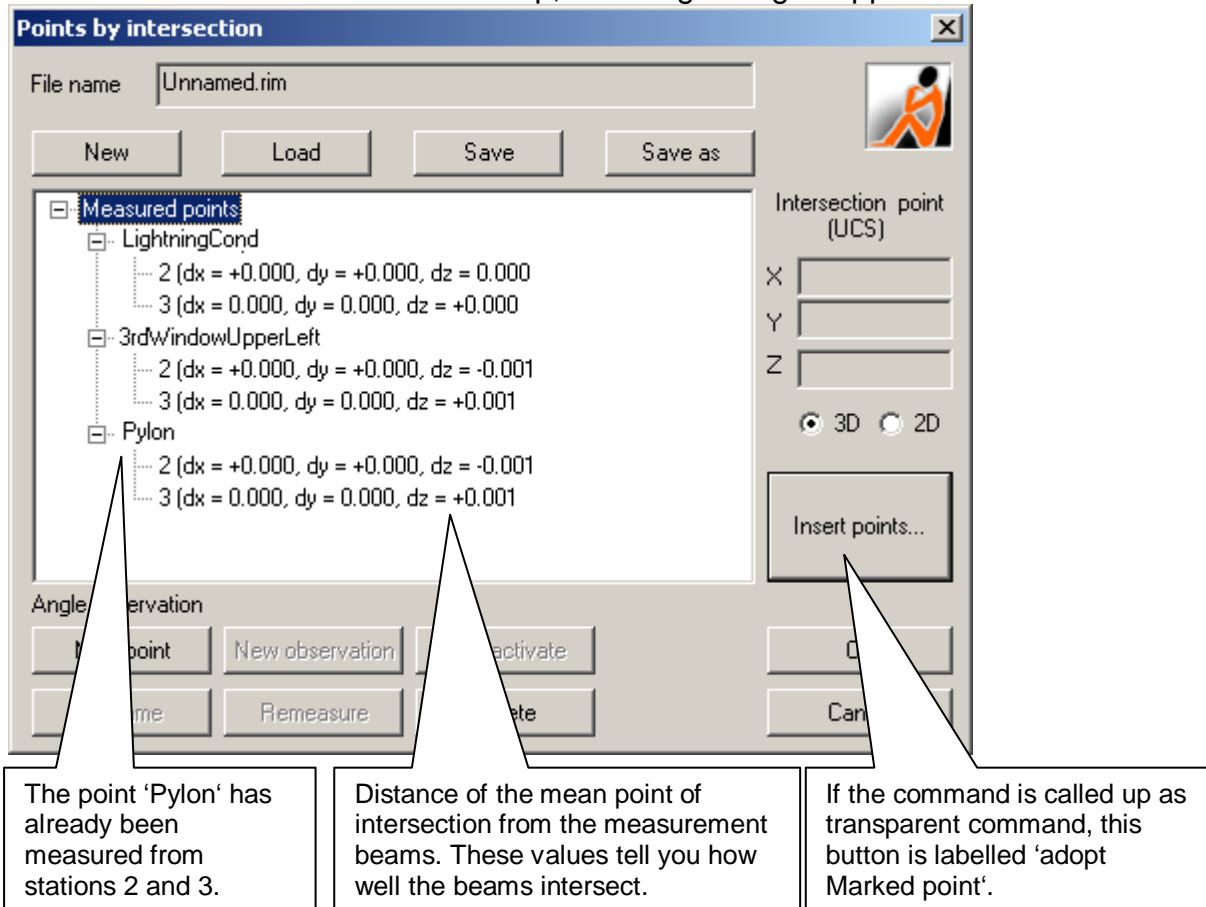| The point 'Pylon' has already been measured from stations 2 and 3. | Distance of the mean point of intersection from the measurement beams. These values tell you how well the beams intersect. | If the command is called up as transparent command, this button is labelled 'adopt Marked point'. |
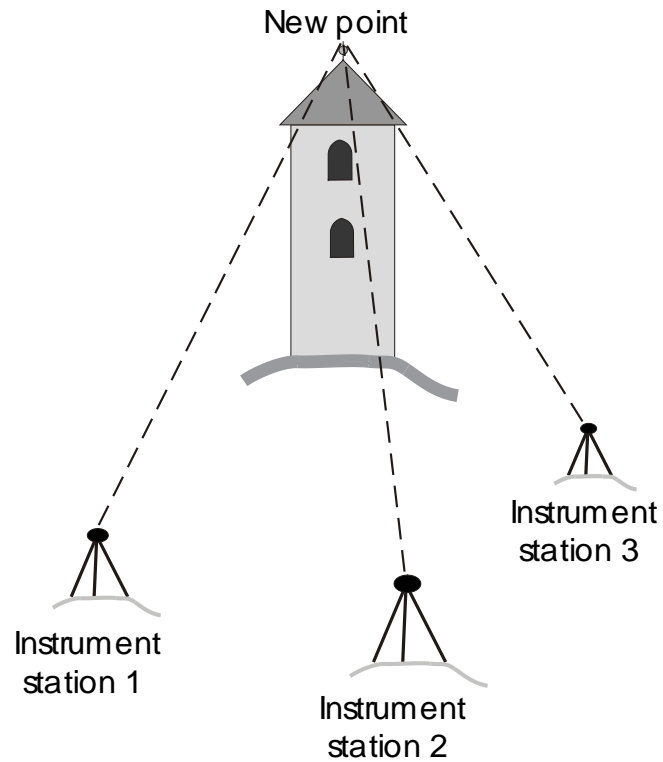
The measurement beams for the individual points are collected here.

The list on the left side contains all points to which measurements have already been taken. For every point there is a note indicating from which stations the measurement has been taken. Below you see a number of buttons with which you can initialize new measurements or edit existing ones. Above the list there are buttons for loading and saving direction measurement data to a file.
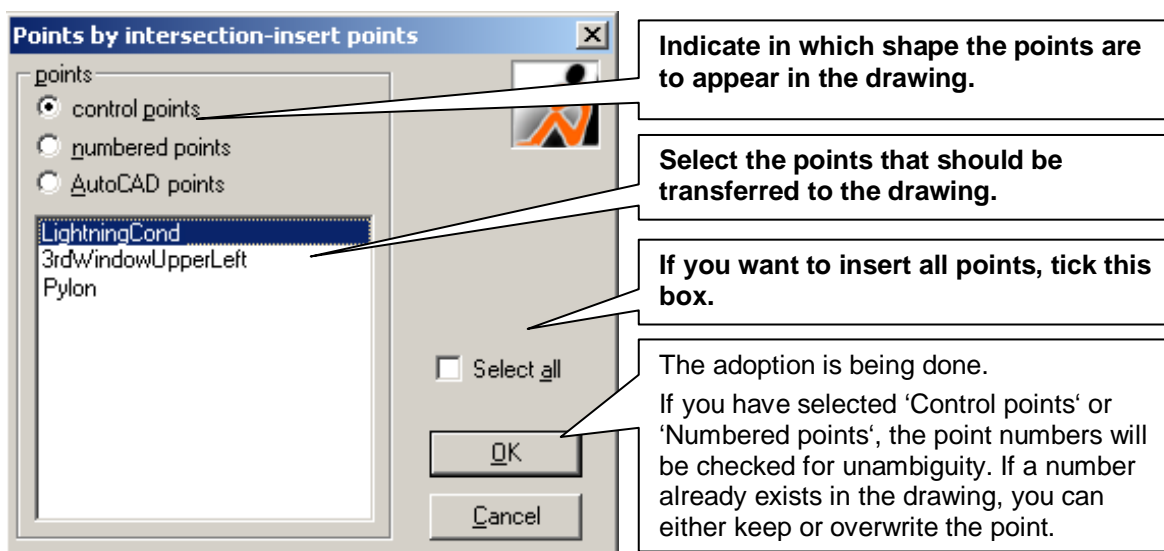
As soon as sufficient (i.e. at least two) measurements have been taken from different orientations, the coordinates of the point of intersection of the beam are displayed under "**Intersection point(s)**". A mean point of intersection is calculated. Normally, the beams will run a bit warped, i.e. will not intersect exactly. As-Built for AutoCAD calculates the shortest distance of the beams to each other, the point of intersection of which is the mean point of intersection.

- **New Point**: You can enter a new point in the list with this button. The extended As-Built for AutoCAD measurement box appears and asks you to take a direction measurement. After the measurement, the point will be added to the list. It will receive a clear name e.g. 'New point1'. By double clicking the point, you can open an entry that tells you the number of the instrument station from which the point was measured.

- **New observation**: With this button you can add a new measurement to a point that already exists in the list. Precondition is that you are standing on a new instrument station. You must have marked the respective point entry in the list

before. The extended As-Built for AutoCAD measurement box appears and asks you to take a measurement.

- **New observation**: You can repeat a measurement that had already been taken. You must have marked the respective entry in the list before. The extended As-Built for AutoCAD measurement box appears and asks you to take a measurement.

- **Rename**: Allows you to give a point a clear name (e.g. Spire Wiltshire). This is particularly important since you must be able to find the point from other orientations later.

- **(De-)activate**: This function allows you to deactivate a measured direction in the list. This direction will not be considered for the calculation of the point of intersection. Select a direction in the list and click Measure again. A respective note will be added to the entry.

- **Delete**: Allows you to remove entries from the list. Depending on which entry you have selected before clicking the button, you delete individual directions, points or all points.

- **Adopt marked point**: You will only find this button if you have called up the command transparently. This means. you call it up within other commands when a coordinate is asked there. The intersection coordinate of the point selected in the list is written into the AutoCAD command line.

- **Insert points**: You will find this button if you have called up this command normally (not transparent). With this command you can insert all intersection points that can be calculated in your drawing. Another dialogue will appear:

Indicate in which shape the points are to appear in the drawing.

Select the points that should be transferred to the drawing.

If you want to insert all points, tick this box.

The adoption is being done.

If you have selected 'Control points' or 'Numbered points', the point numbers will be checked for unambiguity. If a number already exists in the drawing, you can either keep or overwrite the point.

- **OK**: You can close the dialogue with this button. The points and directions are saved.

The use of "Points by intersection" is suitable for the determination of points to which no distance measurement can be taken, be it due to the distance (spires, points in the field), be it due to their shape (thin points) or their material (glass, dark, frosted areas).
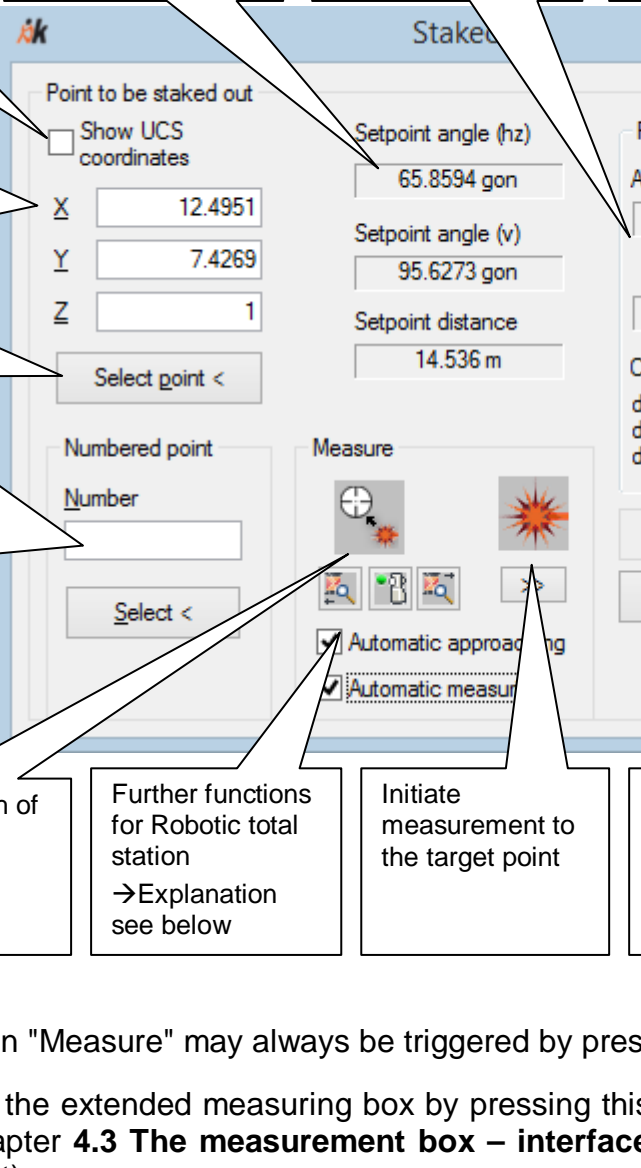
This method is very precise, provided that you have aimed at the point accurately. Always use the telescope with reticule.

Measurement methods – Stake out

|	Command	STAKEOUT
	Ribbon tab	As-Built TS Control

The "Stakeout" command is used to stake out coordinates on site. The points to be staked out can be taken from the drawing or you may enter their coordinates directly into the dialogue. Additionally, you may enter the number of a Numbered Point to get the coordinates.

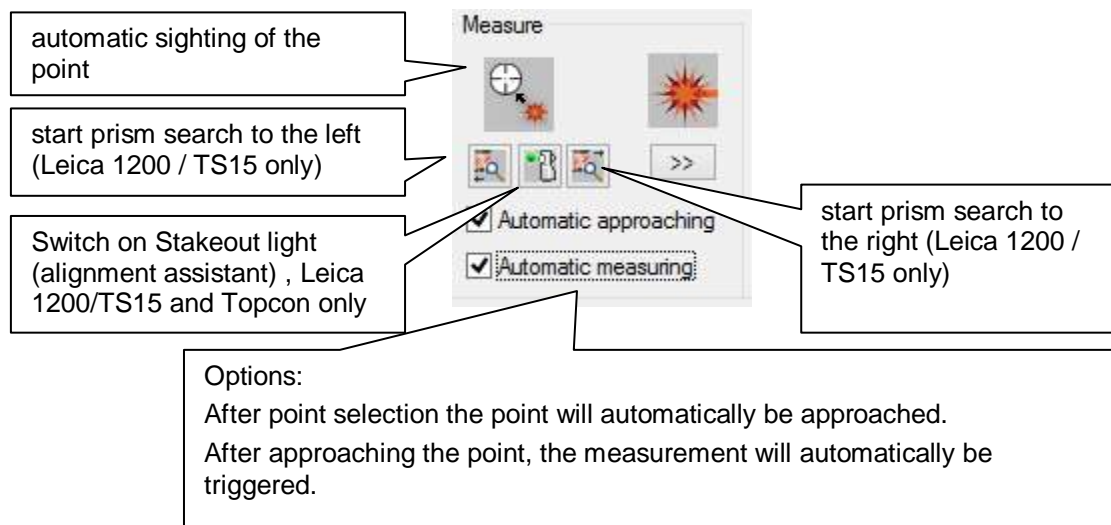After you have called up the command the following dialogue will open.

Display of the coordinates and deviations in WCS or UCS

Angles in horizontal and vertical direction, to be set at the total station

Correction values for the iterative approach to the reference position

Coordinate offset to the set position of the point to be staked out (in WCS coordinates)

Direct coordinate entry of the point to be staked out

Get point to be staked out from drawing

Stake out of a Numbered Point (special TachyCAD point) – entry of the point number or selection from the drawing.

Motorized approach of the point (only for instruments with motorization)

Further functions for Robotic total station →Explanation see below

Initiate measurement to the target point

Documentation of the finally staked out point with a symbol within the drawing

- The button "Measure" may always be triggered by pressing the **Enter** key.

- You open the extended measuring box by pressing this key (Please refer to the manual chapter **4.3 The measurement box – interface between AutoCAD and instrument**)

- By using the **tab** key you will get from the button "Measure" directly to the number entry for the Numbered Point, for those two buttons will be needed very often. By

pressing the **tab** key again you will get to the button "Choose point". That way you may use the dialogue without having to use your mouse.

For motorized total stations there are additional options:

automatic sighting of the point

start prism search to the left (Leica 1200 / TS15 only)

Switch on Stakeout light (alignment assistant) , Leica 1200/TS15 and Topcon only

start prism search to the right (Leica 1200 / TS15 only)

Options:
After point selection the point will automatically be approached.
After approaching the point, the measurement will automatically be triggered.

Stake out – This is the way to do it

In order to transfer a point with known coordinates to the site please do the following. This is an iterative process therefore you will have to repeat it several times:

- State the point to be staked out. You may do this in three different ways:
    - directly via the coordinate entry X, Y, Z (In WCS or UCS coordinates, according to the set option)
    - selecting any point from the drawing by using the button "Select point <"
    - by naming a Numbered Point which has previously been defined within the drawing (a special As-Built point; please refer to the "Insert Numbered Point" command)

- At the instrument set the horizontal and vertical angle which are shown under **Setpoint angle (hz)** and **Setpoint angle (v)**.

- When using a reflector please have the person holding it, move to the correct position. You will have to estimate the distance on the basis of the **Setpoint distance**.

- Click the button "**Measure**".

- Within the "Position offset" field you will now find the longitudinal offset and the angle offset by means of which you will be able to see how much the measured point differs from the point to be staked out.

Measured point

Angle Offset

Point to be staked out

Longitudinal offset

Setpoint distance -->

Station point
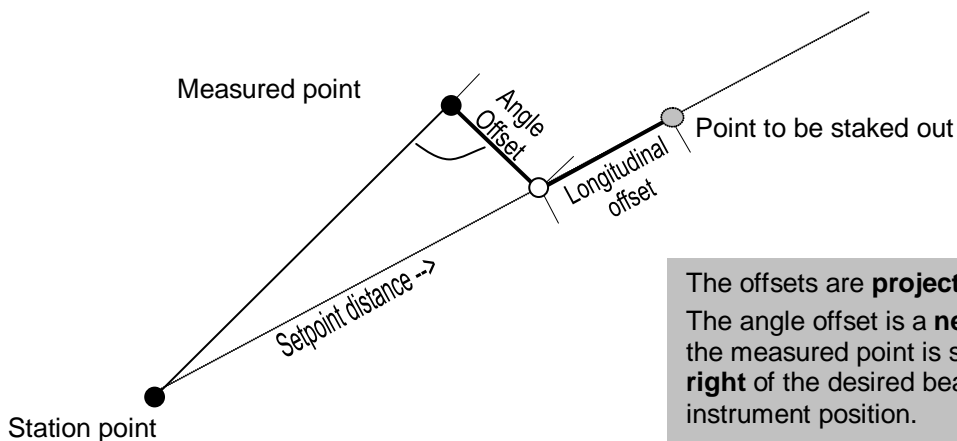
The offsets are **projected** onto XY plane.

The angle offset is a **negative** value, if the measured point is situated **to the right** of the desired bearing seen from the instrument position.

The longitudinal offset is a **negative** value, if the measured point is situated **beyond** the set position.

- Adjust the point's position. When using a reflector have the person holding it move to the new position.

- Measure again and repeat the procedure...

- Once you have reached the needed accuracy, you may use the "Documentation" button to mark the staked out point. A block with the stakeout differences as attribute will be inserted at the position of the staked out point.

If you are using an instrument with robotic function (motorization), you may approach the point to be staked out automatically. In that case you do not have to turn the nominal angles manually at the instrument. The following instrument series are supported:

- Leica TPS1200, TS 15

- Leica 3D Disto

- Trimble Total stations with motorization (S-Series, VX)

- Topcon (via external link mode)

- Flexijet

To initiate the automatic stake out by the instrument just click this button.

If a different instrument is connected to As-Built this button stays disabled or a message appears that no motorized instrument is connected.

Remark: For this command it is important that at the total station the Hz increment is set to RIGHT. That means the horizontal angles are positive counted clockwise. The vertical angles should be counted positively starting at the zenith.

Measurement Methods – Measure hidden point

|	Command:	'TCHIDDENPOINT
	Ribbon tab	As-Built TS Control

With this command it is possible to calibrate a point, which is hidden and cannot be measured directly, via a special measurement pole

Examples for applications:

- Points on the ground of sewage tunnels.

- The corners of houses, curbstones, end points etc. behind hedges and bushes.

- Not visible points behind ledges or within niches.

A measurement pole will be used that has two or three firmly installed target marks. For the work with a reflector there are special posts available at surveying equipment stores ("Hidden Point Pole")

For a reflectorless measurement (especially interior) you may either make such a pole yourself or you equip an existing folding rule or a solid rule (level rule) with sticking markers.
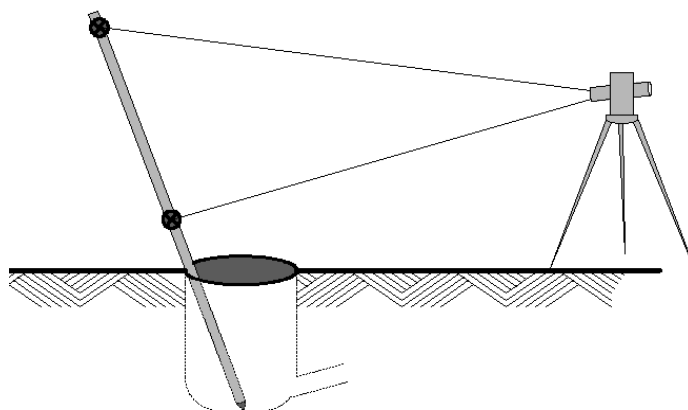

The hidden point will be measured by holding the tip of the measurement pole on the point and aiming at the measuring points on the pole one after the other. The measurement pole may be held freely without any restrictions, it is advisable though to lean it against something for fixation during the measurement.

The hidden point can be calculated via the angles and distances to the measuring marks.

Two measuring marks are necessary, to use a third mark enables you to control the accuracy.

Command cycle:

After starting the command a dialog field will open:
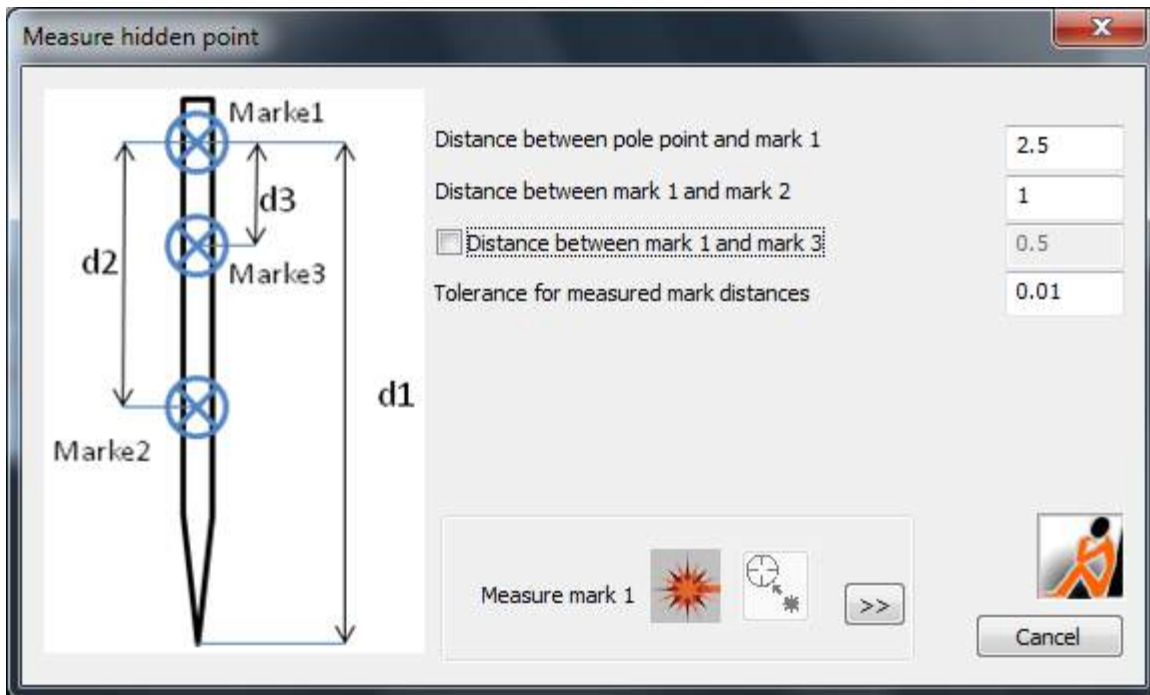
In the upper part you will enter the distances of the measuring marks to one another as well as the length of the pole tip.

Distance d1: Distance between the measuring mark mark1 and the measurement pole tip.

Distance d2: Distance between the measuring marks mark1 and mark2.

Distance d3: Distance between the measuring mark mark1 and mark3 (Activate option only for measurement poles with 3 measuring marks.).

Tolerance measure for the distances of the measuring marks:

These settings are valid for a certain measurement pole. You do not have to enter them every time, but As-Built will remember the values and show them the next time.
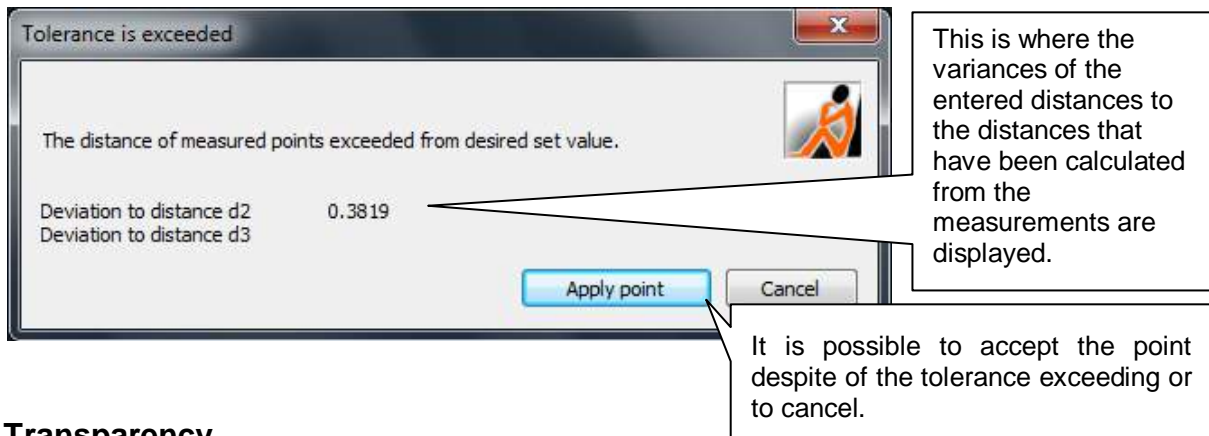
In the lower part is the measurement function. You will be prompted to calibrate the points one after the other.

A third measuring mark can automatically be approached roughly via this button. Youonlyhaveto do thefine-sighting.

Pay attention to the stated measurement order!

If all measuring points have been measured, the dialog will close and a XYZ-coordinate will be calculated. It will be transferred to the command line by As-Built and may then continued to use.

If the measurements should exceed the tolerance measrue, a warning will be put out. It will be displayed within the dialog which distance has been exceeded with which value.

Tolerance is exceeded   ✕

The distance of measured points exceeded from desired set value.

Deviation to distance d2   0.3819
Deviation to distance d3

                                    Apply point   Cancel

This is where the variances of the entered distances to the distances that have been calculated from the measurements are displayed.

It is possible to accept the point despite of the tolerance exceeding or to cancel.

Transparency

The TCHIDDENPOINTcommand has been defined transparently.

"Transparent" means that this command may be called up within any other command. If you want to measure a polyline for example, and you want to determine one or several point with the help of the "Hidden-point-function", you will use this command within the AutoCAD line command. With the help of the command line output the application flow will be explained:

Command: **line**

LINE state first point: *(The first point will be clicked regularly for this example.)*

State next point or [Back]: **'_TCHIDDENPOINT**

                                           (Call up of the command "Measure hidden point".)

Dialog will open. The measuring marks will be requested and calibrated.

Resuming LINE command. (*"Hidden-point-function" completed. Back to the line command.*)

State next point or [Back]: 174.0274,104.5096,-1.4474

                                           (Calculated point will be displayed.)

State next point or [Close/Back]   *(Line command expects next point.)*

…

You may also apply the command within any other command. Every time a coordinate is expected, it can be determined via the *hidden-points-function*.

Measurement method – Net adjustment

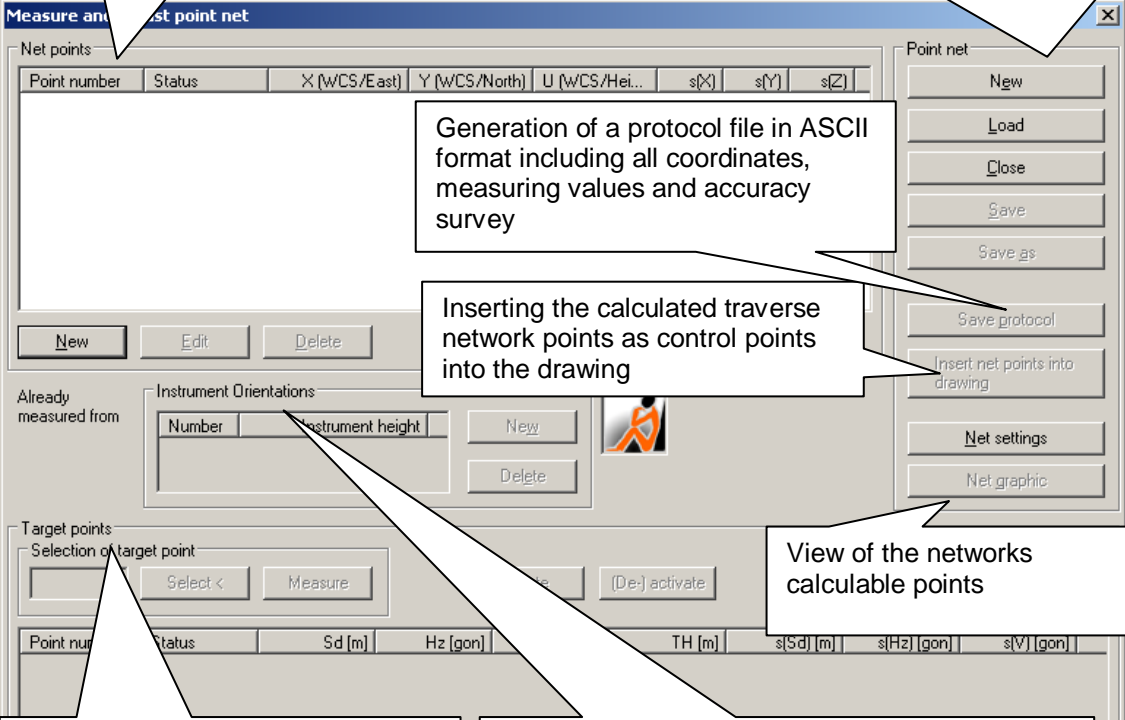|	Command	TCADNETADJUSTMENT
	Ribbon tab	As-Built TS Control

You use this command to measure and calculate a traverse network respectively a traverse. Then As-Built for AutoCAD will calculate a statistical adjustment for the point coordinates. If you made an orientation before, this is ignored during the net measuring process.

Measuring a traverse network resp. traverse is a certain procedure which allows determining the coordinates of points very precisely. This measurement will be performed ahead of the actual measuring up, in order to get a basis of highly precise control points. It is advisable to use a traverse network or traverses for large surveying projects with many base points or if the object to be measured is a very long one.

After you have started the command a dialog will appear. This window will be active during the complete network measurement. Here the measuring results will be collected and the adjustment calculation will be carried out.

This frame shows **all points, involved in the network**. New points may be added, existing ones may be edited or deleted.

General functions:
New: creating a new network measurement
Load: load existing network from file
Save: write network data to file
Save as: save network data with other file name

Generation of a protocol file in ASCII format including all coordinates, measuring values and accuracy survey

Inserting the calculated traverse network points as control points into the drawing

View of the networks calculable points

This frame shows the **measurements towards the target points** made during an orientation. New measurements may be initiated, existing ones may be repeated or deleted.
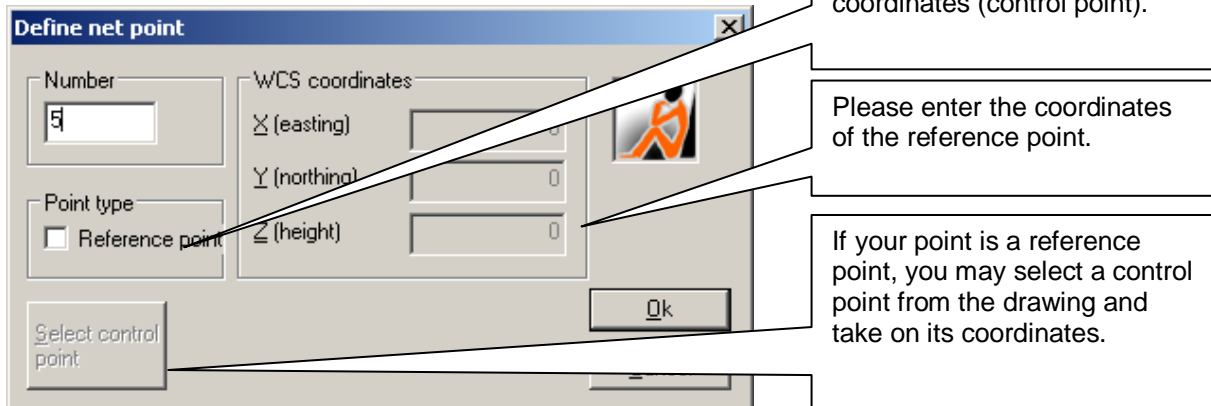
This frame shows all **instrument orientation procedures**, carried out at one station. An orientation is required in order to be able to carry out measurement at a certain position.

The following describes step by step the measurement procedure for a traverse network.

1. Preparation

At first all points of the measuring net will be defined. Therefore you click the "Net points – New" button. The following dialog opens.

Please check this box if your point is a point with known coordinates (control point).

Please enter the coordinates of the reference point.

If your point is a reference point, you may select a control point from the drawing and take on its coordinates.

Please enter a well-defined point number. Please check the "reference point" box, if your point has known coordinates, and enter the coordinates. If the known point is a control point within the drawing you may get the coordinates directly from there. Therefore please click the button "Select control point" and pick it from the drawing. The command line will show the following:

> Select control point or [Number/Finish] <Number>:

If you cannot or do not want to click the point directly, but select it by entering the control point number, please use the option "N" (number).

Create all points, involved in the measurement by doing so. The points will be included into the "Net points" table and will have the status "Underdetermined" or "Reference point".

2. Instrument orientation

Search for your first position now. Please create it as described above, if it is not yet included in the table.

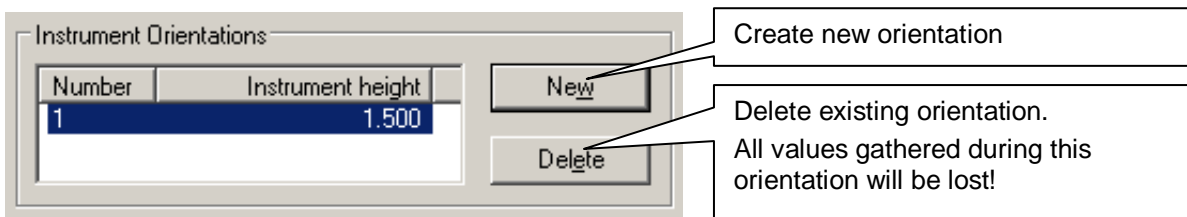Please mark the number of the position within the "Net points" table.

Now move further down in the dialog and click the "New" button within the "Instrument orientation" frame. That way As-Built for AutoCAD knows, that you have positioned the total station above this point.

The following dialog will appear:
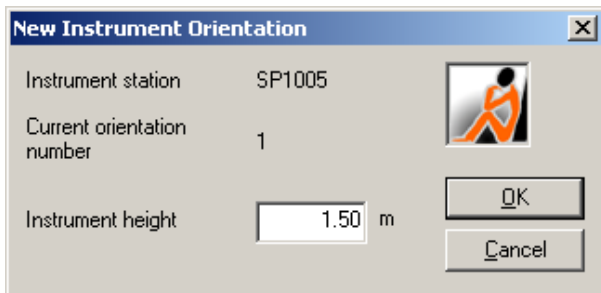
Please measure the height of your instrument (height between the point on the ground and the center of your instrument) and enter it in the respective dialog field if your position is a pegged[10] point. This way the coordinates do not refer to the center of the instrument but to the pegged ground point. Please confirm with OK.

The stationing will be shown within this table:

Create new orientation

Delete existing orientation.
All values gathered during this orientation will be lost!

---

[10] That means a permanent point mark has been created, e.g. driving a pin into the ground. Pegged points may be used again, e.g. as instrument stations or control points. Unmarked points can only be used for the duration of an instrument station.

3. Point measurement

Now you measure the net points that you can see from your instrument position. Pay attention, that you have marked the respective station within the net point table.

Click the "Select <" button within the "Target points" frame and select the point to aim at next. You may enter the point number in the adjusting field instead. Aim at the point and click "Measure". The big measurement box from As-Built for AutoCAD will open. Now you may start the measurement.

Measure all, from your station, visible net points this way. Under "Target points" the aiming will be written into the table below.

Information about the instrument station: first orientation on this point, instrument height above the marked ground point

Deletion of an aiming from the list or deactivation of an aiming.

Point number	Status	Sd [m]	Hz [gon]	V [gon]	TH [m]	s(Sd) [m]	s(Hz) [gon]	s(V) [gon]
1	active	3.4950	147.24400	108.93900	0.000	0.0000	0.0000	0.0000
2	active	5.1090	330.14100	98.09350	0.000	0.0000	0.0000	0.0000
4	active	3.3530	392.00700	106.77050	0.000	0.0000	0.0000	0.0000
9	active	3.9570	234.75300	100.86000	0.000	0.0000	0.0000	0.0000

Measurement values for the target point:

Sd = slope distance

HZ = horizontal direction

V = vertical direction

Reflector height

Standard deviation of the measurement results:

s(Sd) = stand. dev. distance measurement

s(HZ) = stand. dev. horizontal direction

s(V) = stand. dev. vertical direction

At the moment these values are set to zero since there is no match so far

If you want to measure a point again, mark its point number within the table and click "Measure". After a security query the measurement box opens and you may repeat the measurement.

4. Additional positions

Now pick a new position. The procedure for orientation and point measurement works as described in steps 2 and 3. Now repeat this procedure for all positions. The result is an aiming network between the points. Every point should have been measured from two other points. You may check the current net configuration by clicking the "Net graphic" button.
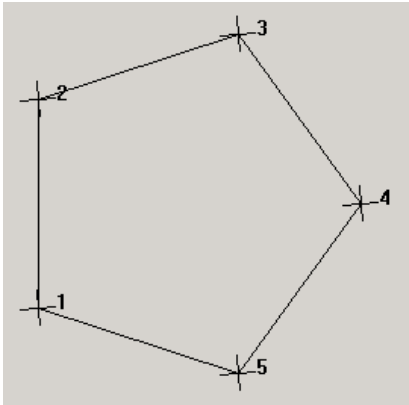
4. Traverse: every point has been measured from the previous and the following point.
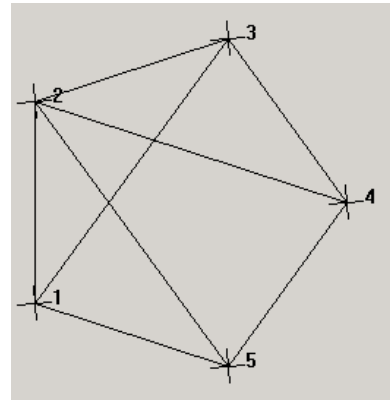
5. Traverse network: several additional cross links have been measured between the points.

If you use one point twice as position, e.g. in order to close a circular traverse, you will have to create a new orientation. Then, in the small table, every orientation will be shown separately under "Instrument orientation". For every orientation the aiming will be saved separately.

So far the table **"already measured from"** has not been mentioned. It shows from which other positions your current position has already been measured from. Usually you will want to measure the opposite direction as well. An Example: If your current position STP2 has already been measured from STP1, you will want to measure STP1 from STP2 vice versa. The table is also a "short cut" for the regular work process. If you double click a point in the list, it will automatically be registered as the next target point.

5. Accuracy inspection

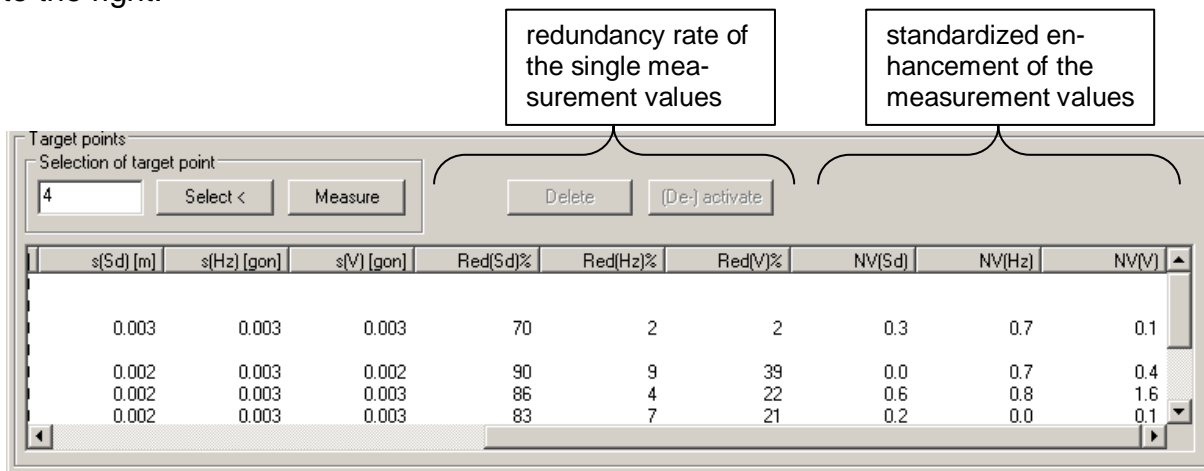Now take a look at the field "Net points" within the large table.

- Points, which are marked as **"reference points"** have known coordinates. They do not have a standard deviation, since they will not be changed by the adjustment calculation. Their coordinates stay the same.

- Points, which are marked **"underdetermined",** cannot be calculated, since they have to less or even no measurement values.

- Points, which are marked **"determined",** have just as many measurements as are needed for the calculation of the coordinates. An error check is not possible though.

- If a point is marked **"overdetermined"**, it has more measurement values than needed for the coordinate calculation. This is the status wanted. As-Built for AutoCAD executes a statistical adjustment, which means, the remainder errors will be spread by geodetic-mathematical criteria and the most likely coordinate will be calculated.

As soon as As-Built for AutoCAD can calculate a net point, the coordinates for the **standard deviations** (s(X), s(Y), s(Z)) will be stated. You can determine the accuracy of the point definition by these values. The standard deviation is a degree for the deviation from the average value. The larger the standard deviation is, the less does the point fit into the overall geometry of the network and the more likely it is that there is a large measurement error.

In order to be able to locate the error, when having large standard deviations, you may **deactivate measurements**. Therefore select the net points one after another and take a look at the correspondent targeting results. By selecting a targeting and clicking "deactivate", you take it out of the calculation. Take a look at the coordinates' standard deviation. If it gets smaller, you should either delete this targeting or measure it again.

The appointed enhancement (NV) and the redundancy rate (Red) are stated for every aiming as an additional help when trying to locate measurement errors. In order to make these visible, move the horizontal scroll bar of the "Target points" table to the right.

redundancy rate of the single measurement values

standardized enhancement of the measurement values

Target points

Selection of target point

| 4 | Select < | Measure | | Delete | (De-) activate | | | |

s(Sd) [m]	s(Hz) [gon]	s(V) [gon]	Red(Sd)%	Red(Hz)%	Red(V)%	NV(Sd)	NV(Hz)	NV(V)
0.003	0.003	0.003	70	2	2	0.3	0.7	0.1
0.002	0.003	0.002	90	9	39	0.0	0.7	0.4
0.002	0.003	0.003	86	4	22	0.6	0.8	1.6
0.002	0.003	0.003	83	7	21	0.2	0.0	0.1

The **appointed enhancement** is the result of a special statistical test procedure for the error detection ("Data snooping"). The larger NV is for a measurement, the more likely it is, that there is an error at this point. As rule of thumb applies:

0 < NV < 2,5 → measurement error unlikely

2,5 < NV < 4 → measurement error possible

NV > 4   → measurement error very likely

If you think the standard deviations of the network are too large, just look for the measurement(s) with the highest NV values and deactivate those.

The **redundancy rate** states, how good a measurement is controlled by other measurements. That means, it states the degree of concordance for this measurement value. The redundancy rate is always between 0 and 100. As rule of thumb applies, a measurement has been controlled enough, if Red lies between 70 and 100. If it is lower, you should execute further measurements for the target point that means creating new positions.

Important hint: The measurement accuracy of the total station is taken into consideration by the statistical adjustment. Therefore, you should in any case pay attention, that the values at **"Net settings"** are the same as your instruments. The manufacturer of your total station provides the applying accuracy values for your instrument.

6. Exporting points to drawing

After additional or repeated measurements have been completed and the standard deviations are satisfactory, you will have to export the points with their adjusted coordinates into your drawing. Please use the button "Points to AutoCAD" to do so. The points will be inserted at the corresponding position in the drawing as control point symbol. If there already is a point with that number in the drawing, a note will appear and the point will not be inserted.

A control point with this number already exists. Please delete this control point from

the drawing or select a different number.

7. Saving net data

In order to be able to call up the measurement and adjustment values again, you have to save them to a file. By clicking the "Save" button a dialog opens from which you can choose the path and name for your new file. The files will have the extension .PUZ

By clicking the "Load" button you are able to open existing net data from a .PUZ file.

Creating protocol file

You may create a protocol file in ASCII format, which includes all coordinates, measurement values and accuracy inspections of the net. Therefore click the "Save protocol" button. A dialog opens where you are prompted to enter the file name. A protocols file extension is always .PRT.

8. Closing dialog

Click the "Close" button. The dialog will close after you have been prompted to confirm to save the net data.

Important: The orientation protocol TACHYCAD.STP will not be overwritten for any orientation within the net adjustment. After starting TachyCAD the program looks for the last orientation. It is to be found in the TACHYCAD.STP file. The following window appears if no orientation has been performed before
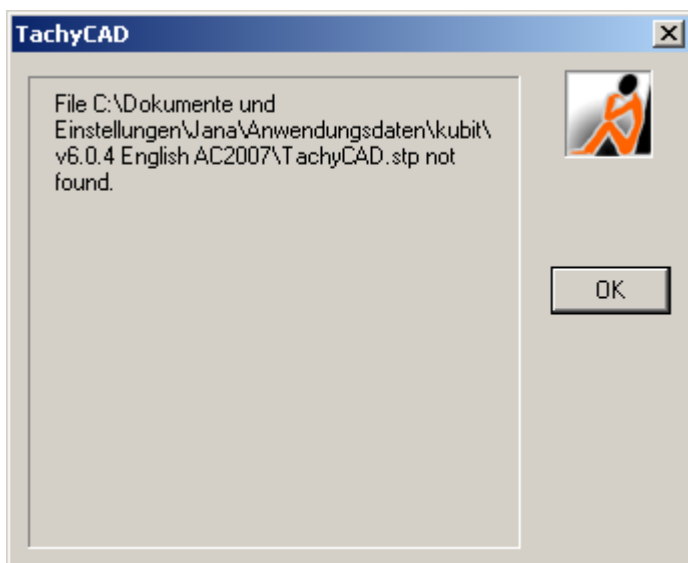
and As-Built for AutoCAD starts a new STP file with the standard values. The STP file will not be overwritten by an orientation within one net adjustment (no matter whether it is a new or existing file). Therefore, points which are measured during a net adjustment have a false position and would appear at a wrong place within the drawing.

That means that you will have to perform an orientation from the last station of the net adjustment if you want to start a measurement from that point! The orientation protocol will not be updated before you do so.

Comments to the mathematical approach of the net adjustment

A "least squares approximation for overdetermined linear systems" (Gauss-Markov-model) will be calculated. It bases on the "method of least squares" according to GAUSS. Please refer to the specialist literature for information regarding the applied standard method.

Refraction and curvature of the earth – how large can my net get?

Generally the total station adjusts the measured distance values regarding the atmospheric conditions by an adjustable factor in ppm. Please refer to the manual of your instrument for the appropriate information.

As-Built for AutoCAD itself adjusts the zenith angle (v in rad) concerning refraction and curvature of the earth and in order to do so uses an (adjustable[11]) refraction coefficient of k=0.13.

$$v' = v + (k - 1) * dH / 2R$$

dH = measured horizontal distance (in meter)
R = Earth's radius = 6380000 meters

K=0,13 is a mean value, in reality the refraction coefficient is subject to considerable fluctuation, especially close to ground level and during heavy solar radiation.

Conclusion: In standard conditions, close to ground level and an accuracy demand in the range of the measurement accuracy of the instrument (about 5mm), the justifiable horizontal measurement distance is at **150-200m**.

Larger horizontal distances cannot be measured reliably with As-Built for AutoCAD close to ground level. For those distances simultaneous foresight and backsight are necessary in order to exactly eliminate the current refraction, which are not intended in As-Built for AutoCAD.

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[11] Can be set via the "As-Built – Settings" command. You will find further information in the respective section within the As-Built for AutoCAD manual.
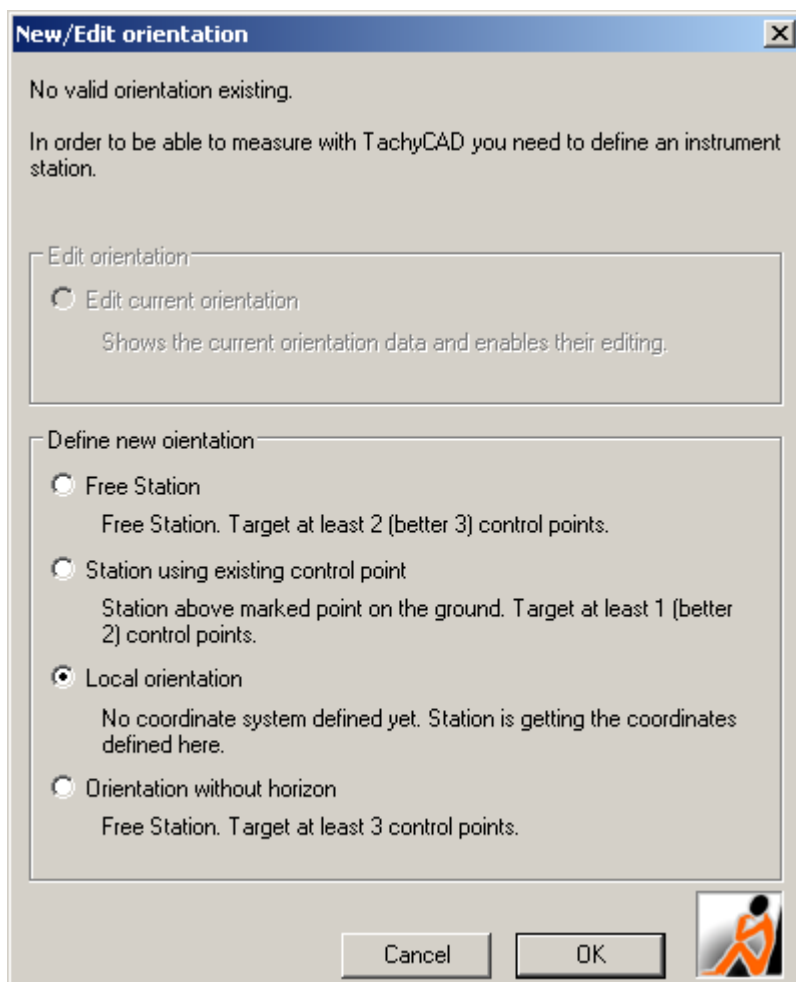
Orientation - New/Edit orientation

	Command:	NEWORIENT
	Ribbon tab	As-Built TS Control

With this command you may define a new instrument station or edit the current one.

If a station does not exist yet, you will be prompted to select the new orientation method.

According to the selected orientation method, a dialogue follows which controls all entries that are necessary for the orientation.

The orientation methods 'Free Station', 'Station using existing control point' and 'Orientation without horizon' require control points. As-Built for AutoCAD control points exist in AutoCAD as a block **CONTROL** with the attribute **NUMBER**.

Please also consider the information in section 4.4 Instrument station, control point, orientation in this manual.

If a station already exists, you may edit it.
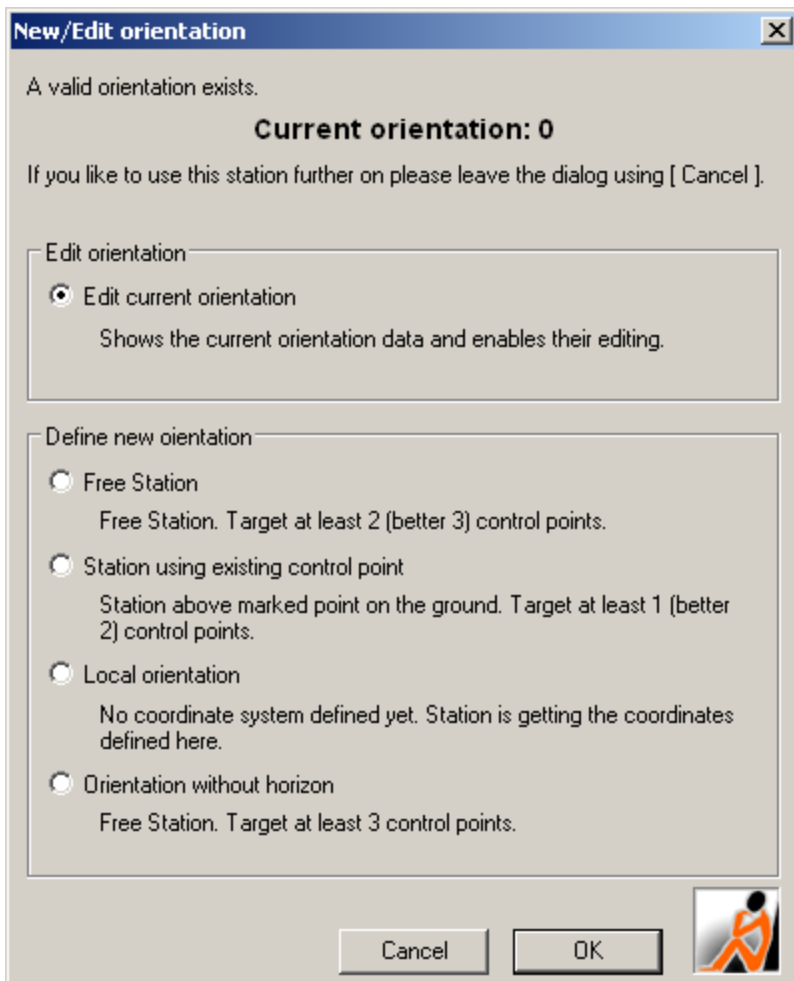
After clicking OK a window, according to the selected orientation method will appear. You may now measure additional control points or remeasure already included control points.

Free Station

Free Station is the orientation method used most often in As-Built for AutoCAD.

For free orientation you can place your instrument at any place from which you can see known points. Then you have to measure at least two known points (control points) for the definite determination of the instrument station. If the number of control points is higher, an adjusting calculation of the instrument station will be run. If you are satisfied with the achieved error values, close the orientation via **OK**. As result of the free orientation, an instrument station symbol will be inserted in AutoCAD at your instrument position.

The following dialogue controls all entries that are necessary for the free orientation.

Enter a number for the **instrument station**; As-Built automatically rises by 1.

Optionally, you can mark the instrument station with a control point in the drawing. This requires the insertion of an instrument height.

The **scale** is very important within free orientation in order to determine the accuracy of your orientation.

Enter the number of the control point to be measured or select the **control point** graphically.

Via these buttons you can control the **status** of the used **control points**.

Here you find the **error values** and the status of all used control points.

The 'traffic light' displays state and accuracy of the orientation.

Instrument station

Enter the desired number for the instrument station in the insertion field. As-Built for AutoCAD checks if this number is already in use. The coordinates of the calculated instrument station are also displayed here.

Control points

Enter and confirm the number of a control point in the insertion field for the measurement. After that you will be asked to measure the control point right away. Alternatively, you can also select the control point directly in the drawing via the button **Select control point**.

Status of control points

You can remove a marked control point from the list with the button **Delete**. By using **(De)activate**, you can exclude a control point from the calculation. However, this control point is not removed from the list.

With the buttons **Full (XYZ)**, **Position (XY)** and **Height (Z)** you can achieve that a control point is used in the calculation of the instrument station with only part of its coordinates. By standard, control points are used as full control points.

Create control point

'Create control point' means that a control point symbol is inserted additionally at the position of the instrument station.

Activating 'Create control point' only makes sense if you want to use the new instrument station as control point for a different instrument station later. 'Create control point' requires that you mark your instrument station at the site (control point sticker, nail), and that you measure and enter the instrument height above this point.

Scale

The scale indicates how well the measured control points match each other in a general perspective. As a result of the free orientation, all measurements that have been taken from the instrument station are multiplied with a scale number. If you do not wish to do so, you can fix the scale at the value 1 with the option 'Scale set to 1'.

Error values

If you use more than two control points for the free orientation, the orientation is over determined. In this case, the remaining deviations are shown for every used control point.

Traffic light

The colour of the traffic light indicates the state of the orientation. The colour red means that the orientation is incomplete – you need more control points. The colour yellow means that the orientation can only be calculated definitely or is relatively imprecise. The colour green means that the orientation is over determined and that the remaining deviation and the scale are within useful tolerances. If you achieve results outside these tolerances, however, the traffic light remains yellow.

If necessary, you can adjust the tolerances in the As-Built for AutoCAD settings.

Station using existing control point

For this orientation method you have to place your instrument above an already existing point (control point).

For the definite determination of the orientation, you have to measure at least one further control point. If there are even more control points, an adjusting calculation of the instrument station is run. If you are satisfied with the achieved error values, close the orientation with the button **OK**.

The following dialogue controls all entries that are necessary for the orientation using an existing point.

Enter the number of the control point that is to be your new **instrument station**. Alternatively, you can pick the point directly from the drawing with the button **Select control point**.

Enter the height of your instrument above the marked instrument station here.

For the evaluation of the general accuracy, the **standard deviation** of the oriented direction is displayed here.

Enter the number of the control point to be measured here or select the **control point** graphically.

You can control the **status** of the used **control points** with this button.

Here you see the **error values** and the status of all used control points.

The 'traffic light' displays state and accuracy of the orientation.

Instrument station

Enter the number of the control point above which you have put up your instrument here. As-Built for AutoCAD checks if this control point is defined in the drawing and displays the coordinates. Alternatively, you can also select your instrument station directly from the drawing with the button **Select control point**.

With the option 'calculate instrument height' you can achieve that the height of the instrument station is not adopted from the control point entered above, but is calculated anew from the measured control points. This option is useful when you are not sure whether the height of the marked floor point is still correct or if you could not measure the instrument height.

Instrument height

Enter the height of your instrument above the marked floor point. This is necessary for As-Built for AutoCAD to calculate the new station coordinates.

Control points

Enter and confirm a number in the insertion field for the measurement of a control point. After this you will be asked to measure the control point right away. Alternatively, you can select the control point directly in the drawing with the button **Select control point**.

Status control points

You can remove a marked control point from the list with the button **Delete**. By using **(De)activate**, you can exclude a control point from the calculation. However, this control point is not removed from the list.

With the buttons **Full (XYZ)**, **Position (XY)** and **Height (Z)** you can achieve that a control point is used in the calculation of the instrument station with only part of its coordinates. By default, control points are used as position control points.

Standard deviation

The standard deviation from the oriented direction is displayed here.

Error values

If you use more than one additional control point for the orientation on a known point, the orientation is over determined. In this case, the remaining deviations are shown for every used control point.

Traffic light

The colour of the traffic light indicates the state of the orientation. The colour red means that the orientation is incomplete – you need more control points. The colour yellow means that the orientation can only be calculated definitely or is relatively imprecise. The colour green means that the orientation is over determined and that the remaining deviation and the scale are within useful tolerances. If you achieve results outside these tolerances, however, the traffic light remains yellow.
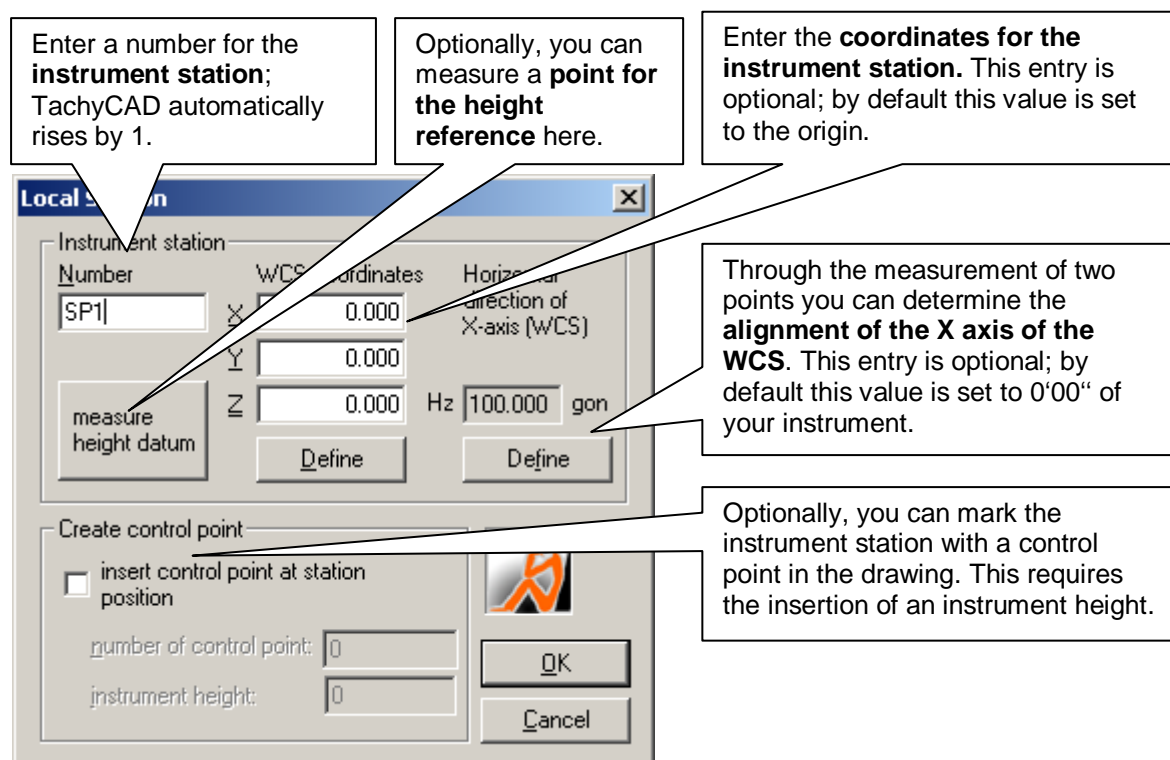
If necessary, you can adjust the tolerances in the As-Built for AutoCAD settings.

Local orientation

In comparison to the other orientation methods, you will use this method rather seldom. With this method you can assign the current position of your instrument to an AutoCAD coordinate. Thus, you **do not need control points** for this orientation method, which is literally speaking no orientation.

You will use this method when control points are not available. Within a project, this will mostly be the case for the first instrument station. Another possible use is when you do not want to connect the measurements from one station with measurements from other stations. In this case, there will be no direct geometrical reference between the measurements from different instrument stations.

The following dialogue controls all necessary entries.

Enter a number for the **instrument station**; TachyCAD automatically rises by 1.

Optionally, you can measure a **point for the height reference** here.

Enter the **coordinates for the instrument station.** This entry is optional; by default this value is set to the origin.

Through the measurement of two points you can determine the **alignment of the X axis of the WCS**. This entry is optional; by default this value is set to 0'00" of your instrument.

Optionally, you can mark the instrument station with a control point in the drawing. This requires the insertion of an instrument height.

Instrument station

Enter the number for the instrument station in the insertion field. As-Built for AutoCAD checks if this number is already in use.

Optionally, you can measure a height reference. This would enable you to define the system in a way that a defined height (e.g. 0.00) is assigned to a certain reference point (e.g. floor of ground floor). If you click the button **measure height reference**, you will be asked to measure a point with the instrument, and to enter a height.

Another option is the definition of a direction for the X axis of the WCS. Thus, you could align the X axis at the wall of a house by measuring two points on this wall. If you do not use this option, the currently set direction 0'00" of your instrument will be used for the alignment of the X axis.

'Create control point' means that a control point symbol is inserted additionally at the position of the instrument station.

Activating 'Create control point' only makes sense if you want to use the new instrument station as control point for a different instrument station later. 'Create control point' requires that you mark your instrument station at the site (control point sticker, nail), and that you measure and enter the instrument height above this point.

Orientation without horizon

Situation

When measuring on solid ground you can assume that the vertical axis of the tachymeter stands absolutely perpendicular. This will be realised through

- a stable tripod and stable underground

- levelling with the help of the foot screws and the level at the tachymeter

- the use of the built-in compensator of the device, which can adjust the remaining inclination and the least unsteadiness of the device electronically.

Things are different when measuring on moving ships. Both, level and compensator are oriented towards the natural plumb line direction of the earth. Since a moving ship is always rocking on the water, the plumb line direction changes constantly and level and compensator do not work anymore. Therefore it is impossible to set up the tachymeter perpendicular.

The solution

The tachymeter will not be levelled with the foot screws and the level but just put up straight according to visual judgement. The internal compensator of the device will be turned off. For the station determination the "Free orientation without horizon" will be used. In comparison to the "usual" free orientation you will have to measure at least 3 (instead of 2) control points in order to calculate the new station. Otherwise the process is very similar.

Only for the very first station of the project a separate mode of operation is necessary. The reference plane has to be determined in order to tell the system, where the wanted horizon is (for there is no natural horizontal plane).

Procedure

1) Turning off the compensator of the tachymeter

▶So the device will not try to adjust the inclination of the vertical axis automatically. Because of the permanent rocking this would result in errors.

2) Station 1 – Local orientation

▶Set up device without levelling, it is enough to set it up straight according to visual judgement.

▶Run As-Built for AutoCAD command "New/Edit orientation" - Local Orientation ". As usual with As-Built for AutoCAD, the measurement of a reference point of height is not necessary.

3) Measure in horizontal reference plane (e.g. on a plane floor or along a laser line)

▶Measurement of 3 points: As-Built for AutoCAD command "Construction Tools"/ "Design" – Draw Plane (KP_DRAW_PLANE)

or

▶Measurement of several points: As-Built for AutoCAD command "Construction Tools" – "Fit Plane" (KP_CREATE_PLANE_FROM_OBJECTS) → As-Built for AutoCAD will calculate the middle plane.

4) Align the world coordinate system of the drawing along the reference plane

▶As-Built for AutoCAD command **TCADWCSTOPLANE** (enter into command line!) → the WCS of the drawing will be changed, so the XY plane will be identical to the reference plane. A height offset may be stated.

5) Ready for the measurement! Geometry will be measured until a new station becomes necessary. Do not forget to measure control points!

6) Station 2 – Free orientation without horizon

▶Set up device without levelling (visual judgement is enough).

▶As-Built for AutoCAD command "New Station – Orientation without horizon" → Measurement of at least 3 control points.

7) Ready for the measurement! Geometry will be measured until a new station becomes necessary. If needed measure additional control points for succeeding stations.

Steps 5 through 7 will be repeated until the end of the measurement project. Only for the very first station of the project a separate mode of operation is necessary

Free orientation without horizon

The following dialogue controls all entries that have to be done.

Enter a number for the **instrument station**; TachyCAD automatically rises by 1.

Enter the number of the control point to be measured or select the **control point** graphically.

Here you find the **error values** and the status of all used control points.

Via this button you can control the **status** of the used **control points**.

The 'traffic light' displays state and accuracy of the orientation.

For the orientation without horizon you can set up your instrument at any position from where you can see known points. Then you will have to measure at least three known points (control points) in order to be able to determine the station. For a larger number of control points there will be an adjusting calculation of the station. If you are satisfied with the obtained error values, close the orientation via the **OK** button. In AutoCAD a station symbol will be inserted at the instrument position within the result of the orientation without horizon.

Instrument station

Enter the desired number for the instrument station in the insertion field. As-Built for AutoCAD checks if this number is already in use. The coordinates of the calculated instrument station are also displayed here.

Control points

Enter and confirm the number of a control point in the insertion field for the measurement. After that you are asked to measure the control point right away. Alternatively, you can also select the control point directly in the drawing via the button **Select control point**.

Status of control points

You can remove a marked control point from the list with the button **Delete**. By using **(De)activate**, you can exclude a control point from the calculation. However, this control point is not removed from the list.

Error values

If you use more than three control points for the orientation without horizon, the orientation is over determined. In this case, the remaining deviations are shown for every used control point.

Traffic light

The colour of the traffic light indicates the state of the orientation. The colour red means that the orientation is incomplete – you need more control points. The colour yellow means that the orientation can only be calculated definitely or is relatively imprecise. The colour green means that the orientation is over determined and that the remaining deviation and the scale are within useful tolerances. If you achieve results outside these tolerances, however, the traffic light remains yellow.

If necessary, you can adjust the tolerances in the As-Built for AutoCAD →**Settings**.

Usage of a motorised tachymeter

If you use a tachymeter with motorisation, it is possible to roughly approach the control points automatically.

Starting from a certain number of control points the large measurement box provides an according function:

- Free orientation: starting with the third control point

- Orientation on known point: starting with the second control point

- Free orientation without horizon: starting with the fourth control point

If you want the automatic approach to start upon opening the measurement box (without having to click a button) you may set the following variable within the → **As-Built for AutoCAD settings**:

System – TachymeterAutoMotorApproach → value = true

Orientation - Verify current orientation

	Command:	VERIFORIENT
	Ribbon tab	As-Built TS Control

The command "veriforient" allows the verification of the current orientation. The command compares the real position of a control point with the measured position of a control point. Thus, the command asks you to enter a control point number.

Command: **veriforient**

Enter control point number or <Pick from drawing>: **CP56** *(Enter number of control point)*

After the measurement the real and measured coordinates as well as the spatial distance are displayed in a dialogue:
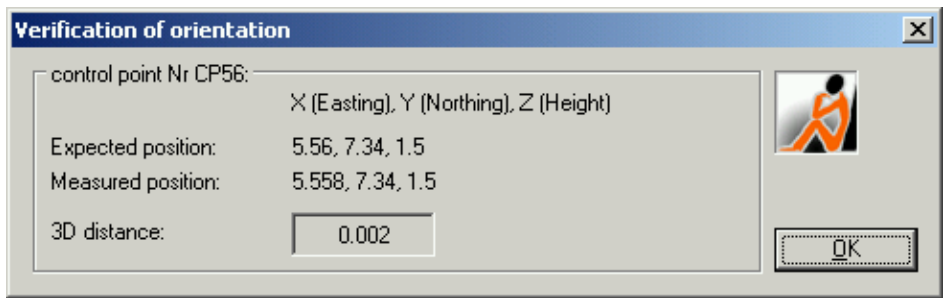
Usage of a motorised tachymeter

If you use a tachymeter with motorisation, it is possible to roughly approach the control points automatically. The large measurement box provides an according function:
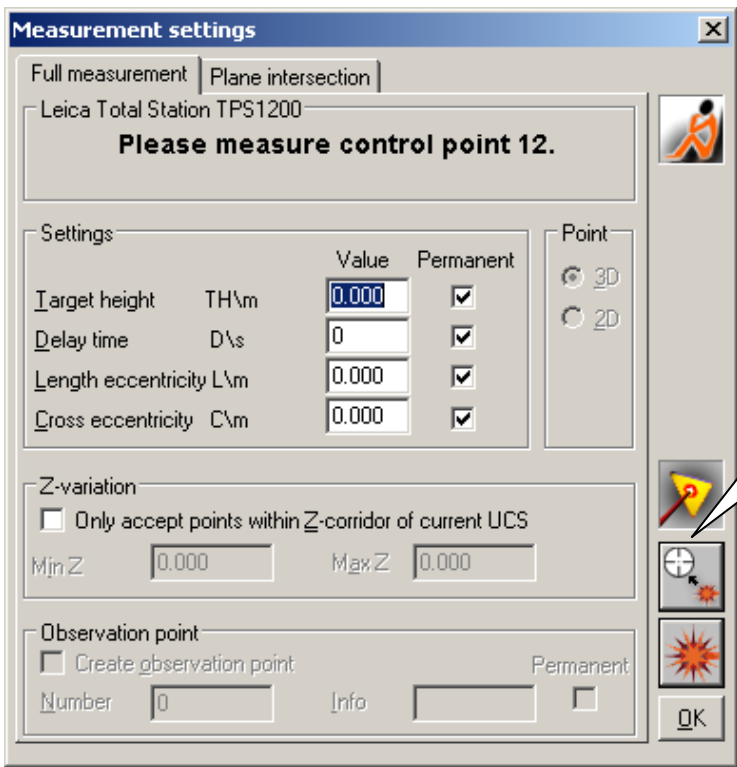
Via this button the control point will be approached motorised.
Caution: It is only a rough approach. The fine aiming has to be done manually.

If you want the automatic approach to start upon opening the measurement box (without having to click a button) you may set the following variable within the → **As-Built for AutoCAD settings**: System –

TachymeterAutoMotorApproach → value = true

Orientation - Find instrument station

	Command:	FINDST
	Ribbon tab	As-Built TS Control

The command "findst" checks if an instrument station already exists in the current drawing.

Command: **findst**

Station number: **57** *(Enter number)*

If the instrument station is found in the drawing, As-Built for AutoCAD displays its coordinates in the command line.

Station 1 found at (-11.895, 10.160, 98.769).

If you answer following question

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

With 'Yes', the station will be zoomed to the middle of the screen.

Orientation - Define new control point

	Command:	DEFCP
	Ribbon tab	As-Built TS Control

The command 'defcp' is used for creating a new control point by entering its coordinates. The coordinates must be in the World Coordinate System (WCS) i.e. in the same system in which all orientations are done.

After the command has been called up, following dialogue appears:
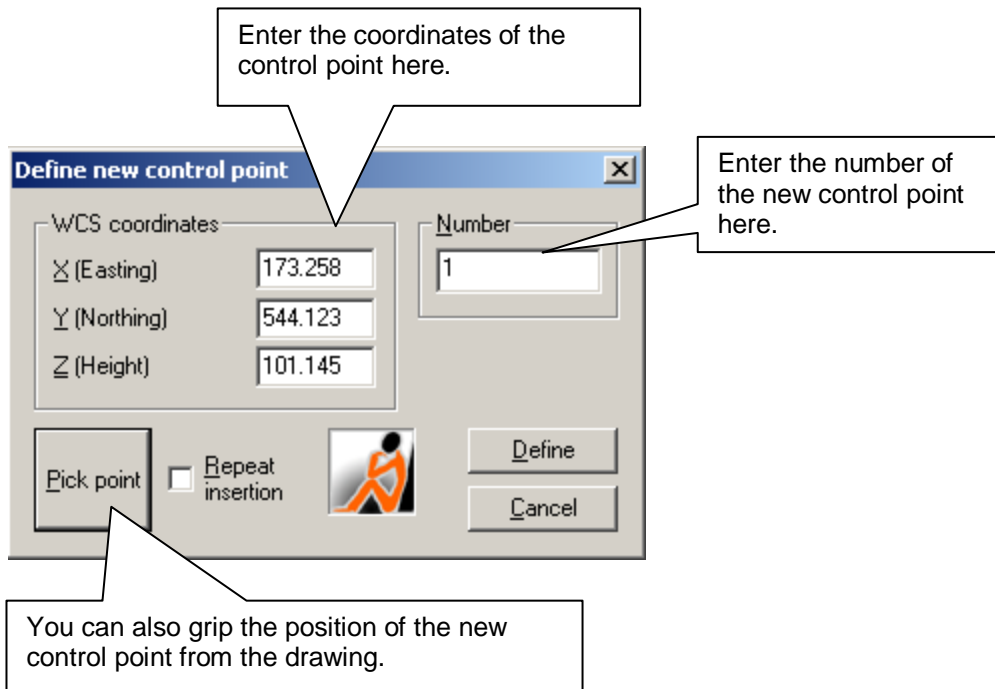
With **Define** a new block of the type CONTROL will be inserted on the layer "Control" at the indicated position. It will also be checked if the entered point number already exists. If this is the case, you will be notified. When you are asked to insert a point number, As-Built for AutoCAD offers a number that is raised by one compared to the previous number.

Via the button 'Repeat insertion' you have the opportunity to define several control points consecutively without having to start the dialogue for every single point.

Advice: Organise your project in a way that all control points are in only one drawing. Switch to this drawing for measuring a new control point and for orientation, and switch back to the survey drawing after the measurement / orientation. This method makes it easier to keep and enlarges security, especially for bigger projects.

Orientation - Measure new control point

	Command:	MEASCP
	Ribbon tab	As-Built TS Control

The command 'meascp' is used for measuring a new control point.

Command: **meascp**

Enter new control point number <2>:

(Enter point number or press ENTER for taking over the default value)

After the measurement, the measured point serves as insertion point for a new control point symbol. The symbol is the block **control** which is laid on the layer 'CONTROL'.

When you are asked to insert a point number, As-Built for AutoCAD offers a number that is raised by one compared to the previous number.

The command also checks if the inserted number already exists in the drawing. In that case, you will be notified and a different number has to be entered.

Advice: Organise your project in a way that all control points are in only one drawing. Switch to this drawing for measuring a new control point and for orientation, and switch back to the survey drawing after the measurement / orientation. This method makes it easier to keep and enlarges security, especially for bigger projects.

Orientation - Find control point

	Command:	FINDCP
	Ribbon tab	As-Built TS Control

The command "findcp" checks if a control point exists in the drawing.

Command: **findcp**

Enter control point number: 3   *(Enter number)*

If the control point is found in the drawing, As-Built for AutoCAD displays the coordinates in the command line.

Control point 3 found at (19.608, 14.137, 0.000).

If you answer following question:

Zoom to control point? [Yes/No] <Yes>:

With 'Yes', the control point is zoomed to the middle of the screen.

Edit drawing - Import points from file

⊠	Command:	TCADIMPORT
	Ribbon tab	As-Built TS Control

The command "tcadimport" is used for reading ASCII coordinate data into AutoCAD drawings.

Select the path of your file. A select file dialogue will appear.
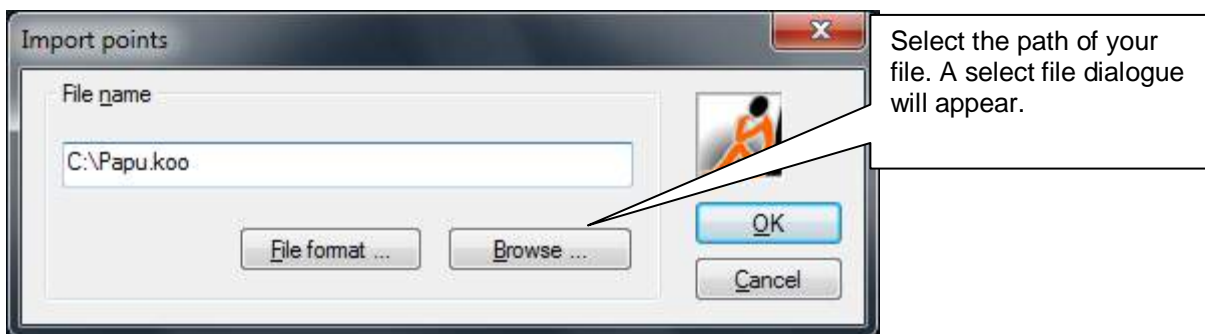
You can determine the format of the ASCII file to be read with the button **File format...**. By creating an info block, you also have the possibility to display additional information. For this a block will be inserted next to the point to be created.

Indicate which value stands in which column in your file.

Indicate how your coordinate file is formatted.

Indicate which values are to be displayed in the info box.

Indicate in which way the points are to be read into the drawing.
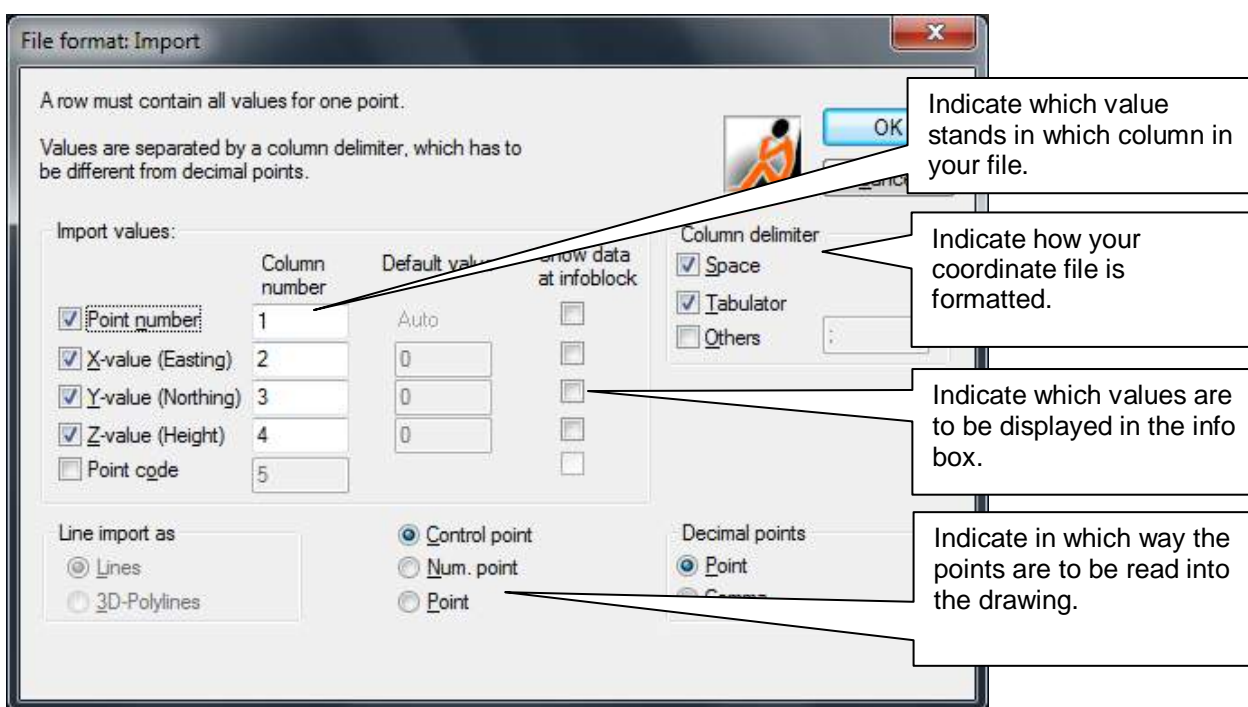
Importing data makes sense in different situations:

- If you use official fixed points, control points may be in a ASCII coordinate file. This command allows you to insert these points into your drawing as control points. For this you have to activate the option "**Control point**".

- For bigger surveying projects it is often necessary to copy control points from one drawing into another. Use the command "Export points" to copy the control points from a source file into a text file. Then you can insert the control points into the target file with "Import points". For this you have to activate the option "**Control point**".

- If you measure with your total station offline (i.e. record data in the instrument without As-Built for AutoCAD), a text file is created, in which all observation points are saved with point number and coordinates. Use this command to read these points into an AutoCAD drawing.

  - If you activate the option "**Num. point**", all measurement points are inserted into the drawing as numbered points.

  - If you activate the option "**Point**", ordinary AutoCAD points will be set onto the coordinates.

  - You can also use the command to read points which you have recorded encoded with a total station layer structured into AutoCAD. Use the option "**Point code**".

  The used three-digit point code has following format:

Control points	Code **000** (Import on layer 'CONTROL')
Single point	1st Code digit **1**; 2. and 3. Code digit Layer name
Line start	1st Code digit **2**; 2. and 3. Code digit Layer name
Line passage	1st Code digit **3**; 2. and 3. Code digit Layer name
Line end + close	1st Code digit **4**; 2. and 3. Code digit Layer name
Line end	1st Code digit **5**; 2. and 3. Code digit Layer name
Arc start	1st Code digit **7**; 2. and 3. Code digit Layer name
Arc passage	1st Code digit **8**; 2. and 3. Code digit Layer name
Arc end	1st Code digit **9**; 2. and 3. Code digit Layer name

Edit drawing - Export points to file

	Command:	TCADEXPORT
	Ribbon tab	As-Built TS Control

The command "tcadexport" is used for exporting data of control and numbered points into an ASCII file. These data can then be imported into other AutoCAD drawings with the command "tcadimport".

Select the path of your file. It then appears in the dialogue.

Specify the format of your selected ASCII file.

Select the type of the point you want to export.

This list shows all points of the type selected above that exist in the drawing.
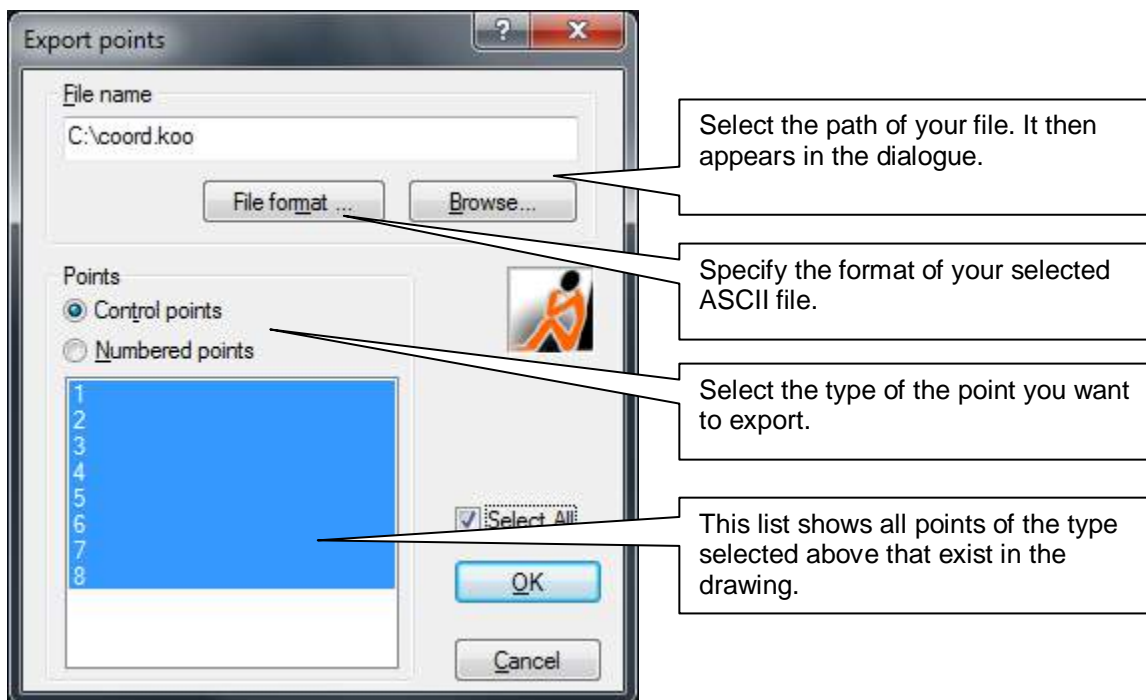
The format of the ASCII files to be read in can be determined with the button **File format...**:

Enter how your coordinate file has been configured.

Enter the accuracy with which the measurement values are to be exported.
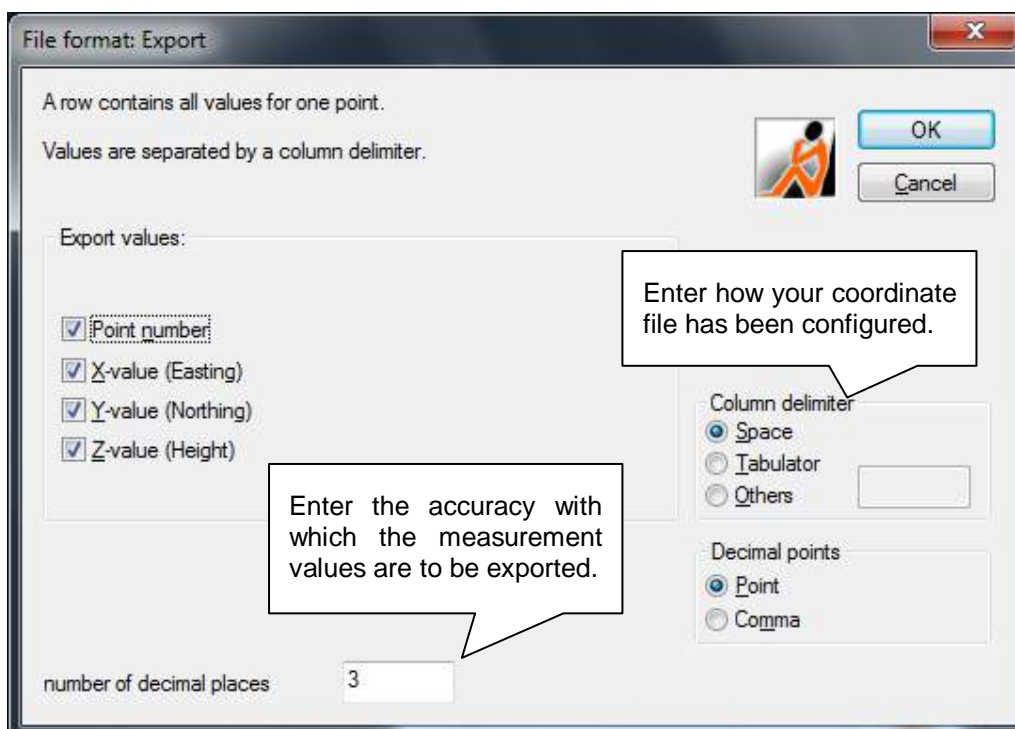
Edit drawing – Flatten
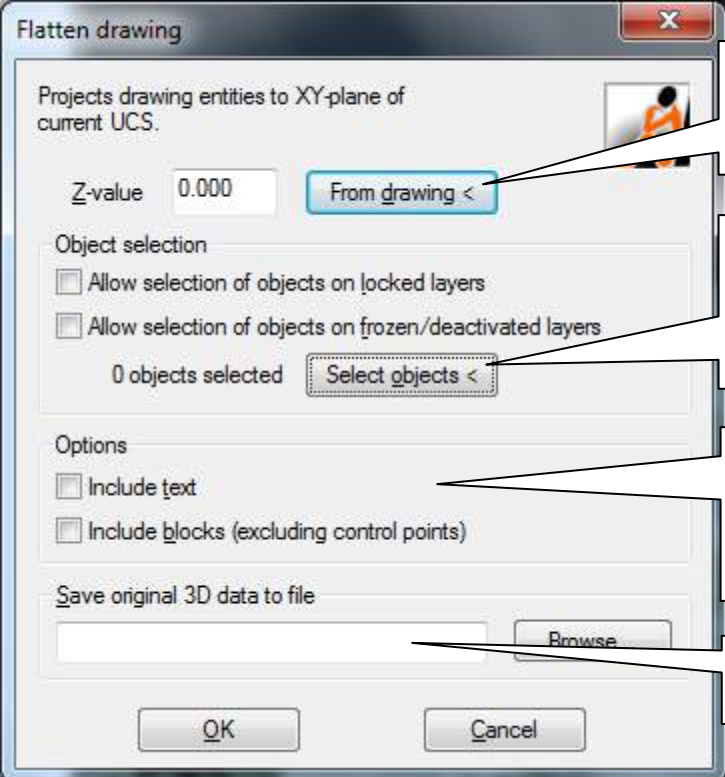
	Command:	TCADFLATTEN
	Ribbon tab	As-Built TS Control

The command "tcadflatten" 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.

Flatten drawing

Projects drawing entities to XY-plane of current UCS.

Z-value 0.000 [From drawing <]

Object selection
☐ Allow selection of objects on locked layers
☐ Allow selection of objects on frozen/deactivated layers
0 objects selected [Select objects <]

Options
☐ Include text
☐ Include blocks (excluding control points)

Save original 3D data to file
[_____] [Browse]

[OK] [Cancel]

All elements are projected to this Z value of the UCS. Via the button you can grip 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, a backup file may be generated before the flattening.

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.

The As-Built for AutoCAD blocks for control points, instrument stations and survey points are not flattened. A flattening does not make sense here, since the 3D positions are absolutely necessary for orientation or when used as measurement protocol.

Important advice!

If at the time when you wish to flatten the drawing the absolute reference datum of the heights should not be determined definitely or if you cannot obviate that you have to change it again later, then please exclude the height blocks from the flattening process. This can be done be simply <u>not</u> selecting them for flattening. You may also freeze or lock the object layer of the height blocks and exclude those layers from the flattening process (default setting). The reason for this advice is, when a height block is flattened its Z coordinate and thus its height changes. While flattening the drawing As-Built for AutoCAD does not touch the attribute values of the heights. But when changing the absolute reference datum As-Built for AutoCAD correctly refers to the actual height (Z coordinate) and not to the text attribute of the block. Thus, the height values now no longer fit to any of the pervious measurements.

Also see →**Height dimension tools - Subsequent editing - Set absolute reference datum**

Edit drawing – Transformation – Simple HELMERT transformation

	Command:	TCADHTRAFO
	Ribbon tab	As-Built TS Control

This command runs a simple Helmert transformation without over dertemination. Only two pairs of points have to be asigned. The options 'shifting', 'rotation' and 'scaling' are always enabled.

Please read ➔**Edit drawing – Transformation – Advanced HELMERT** transformation for further details on this command.

Edit drawing – Transformation – Advanced HELMERT transformation

	Command:	TCADNTRAFO
	Ribbon tab	As-Built TS Control

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 **TCADNTRAFO**). The other one would a calculation with only two pairs of points (Simple Helmert transformation **TCADHTRAFO**).

After starting the command 'TCADNTRAFO' 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 ➔**Edit drawing – Transformation – 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.

Edit drawing – Transformation – Set text height

	Command:	TCADNTTEXTHEIGHT
	Ribbon tab	As-Built TS Control

With this command you may customize the text height for the temporary point numbers used during the Helmert transformation in case they are to big or to small.

Also see ➔**Edit drawing – Transformation – Advanced HELMERT** transformation.

Edit drawing – Plan analysis – Find gaps

	Command:	ANAGAP
	Ribbon tab	As-Built TS Control

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.
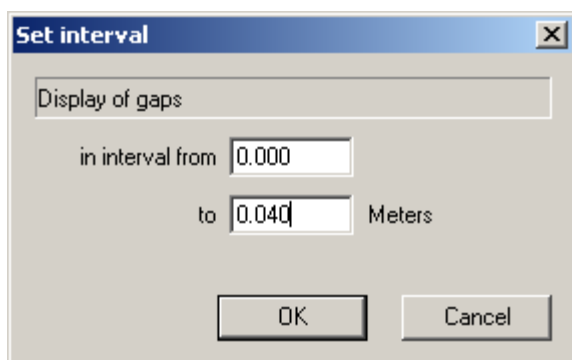
Here you indicate a length interval. All gaps that lie within this interval are marked. After confirmation with **OK** you are prompted to select the objects to be analysed. As-Built for AutoCAD will now mark the gaps by a blue symbol and you may now revise the plan accordingly.
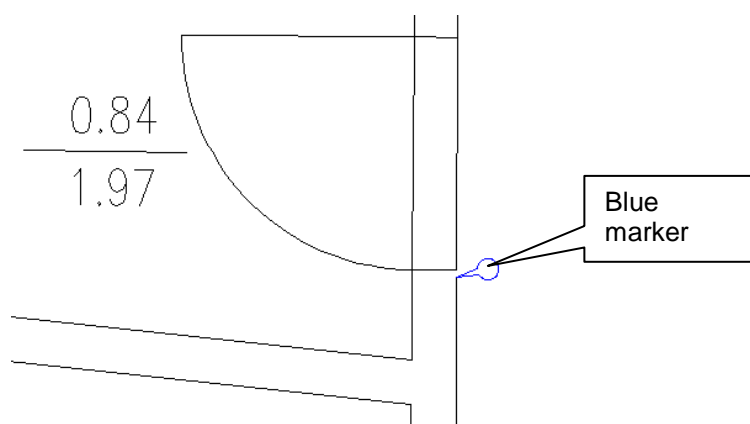
The size of the red marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command ➔ **Edit 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 software 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 command →**As-Built Settings** (Area data and archaeological data). Thus at least for working with As-Built for AutoCAD 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.
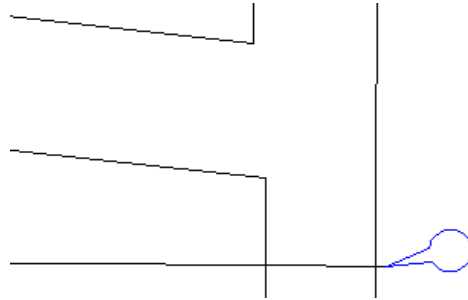
The search for gaps is an elaborate process of analysing the CAD graphics. It therefore may take some seconds (depending on the selected plan section) until the result is displayed.

Edit drawing – Plan analysis – Find short lines

	Command:	ANASHORT
	Ribbon tab	As-Built TS Control

You may find short lines using this command. They are marked by a green flag.

.
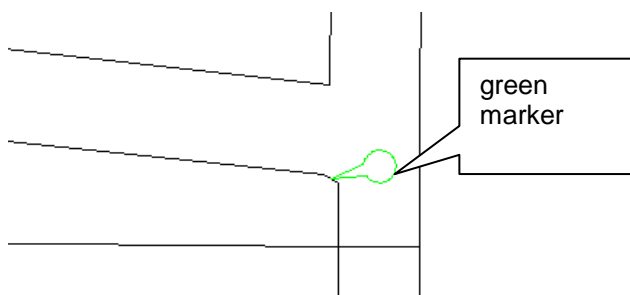
green
marker

The length of the lines to be detected is prompted by this dialogue.
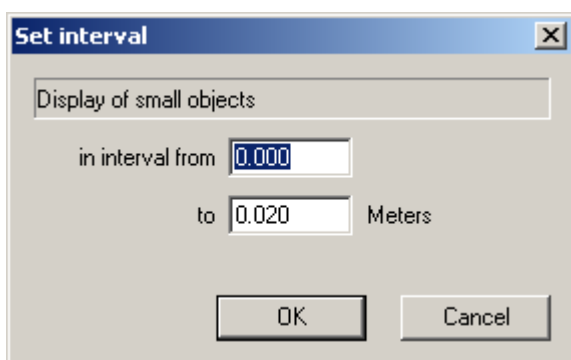
The size of the green marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command ➔**Edit 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.

Edit drawing – Plan analysis – Find double lines

	Command:	ANADOUBLE
	Ribbon tab	As-Built TS Control

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.
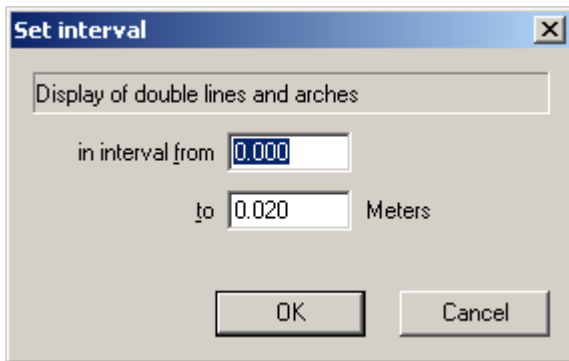
Here you indicate a length interval. All lines whose maximum distance lies within this interval are marked. After confirmation with **OK** you are prompted to select the objects to be checked for double lines and arches. As-Built for AutoCAD will now mark the double draws by a red symbol and you are able to revise the plan accordingly.
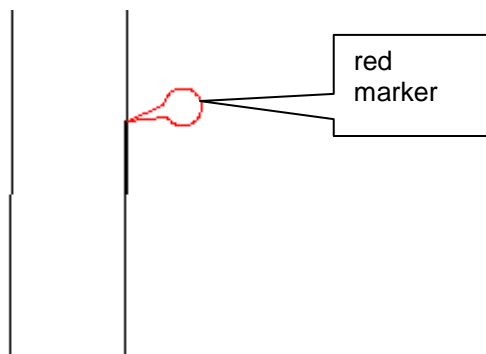
The size of the red marker depends on the level of the zoom.

All markers are deleted when running another plan analysis command. Alternatively you may initiate the deletion yourself by using the command ➔**Edit 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.

Edit drawing – Plan analysis – Trace outline

	Command:	ANAPATH
	Ribbon tab	As-Built TS Control

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 arborisations the side of the line is preferred which you selected when clicking close to the line.
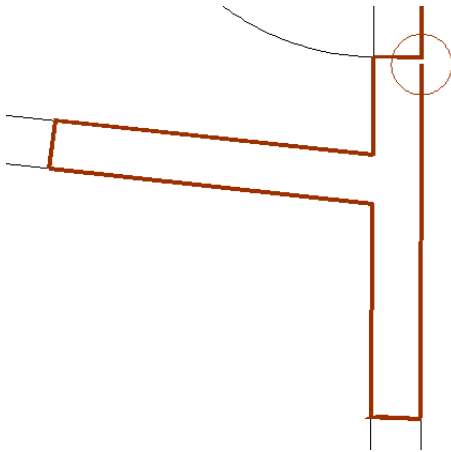
The ends of an outline are highlighted by circles.

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:
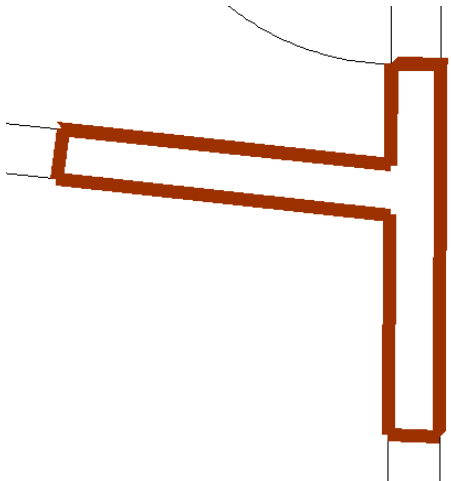
If the outline tracing results in a so called self-overlapping, the intersection points are marked by concentric circles. The outline line is also highlighted in purple.

Such a polyline has to be seen critically since its surface area is not well defined (even though it is closed).
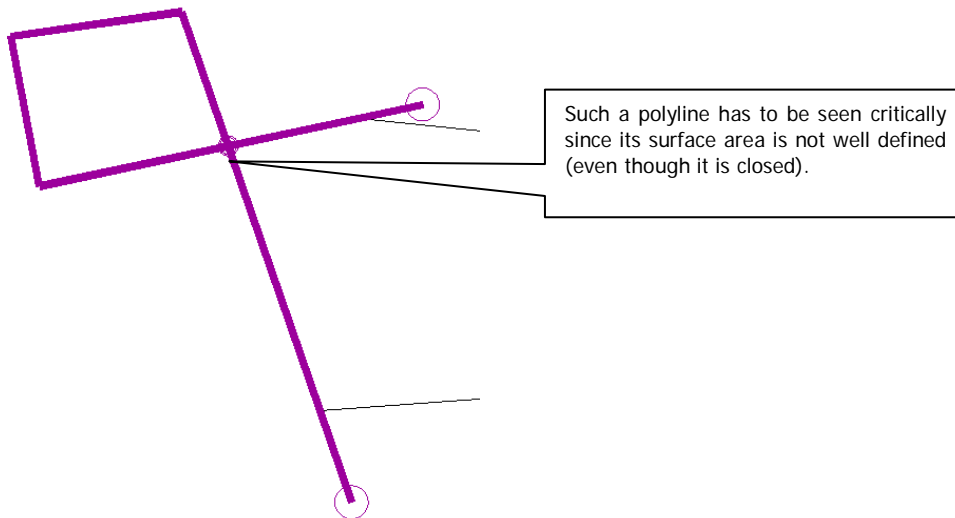
The outline tracing function runs a complex analysis of the CAD graphic. Therefore, the process may take a couple of seconds (depending on the selected plan section).

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.
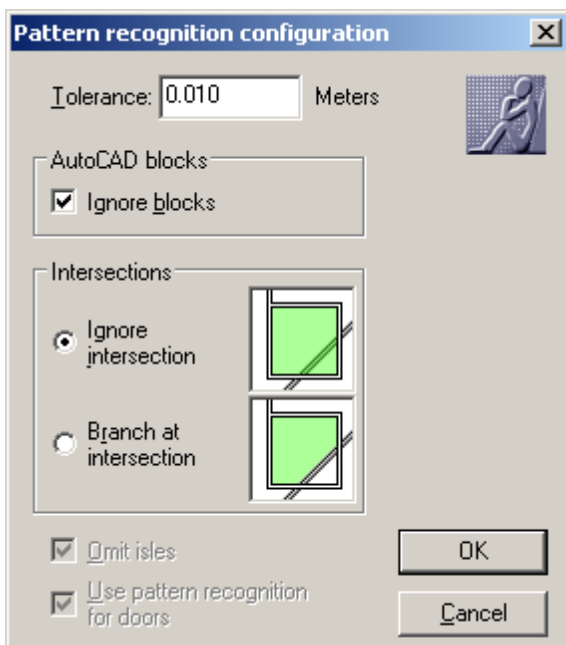
- **Ignore intersection** means that the outline tracing does branch at true X intersections.

- **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.

Position of the click point
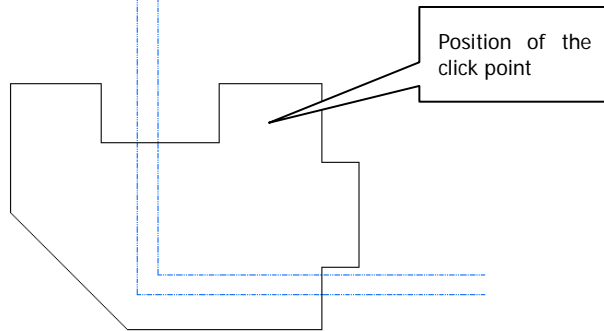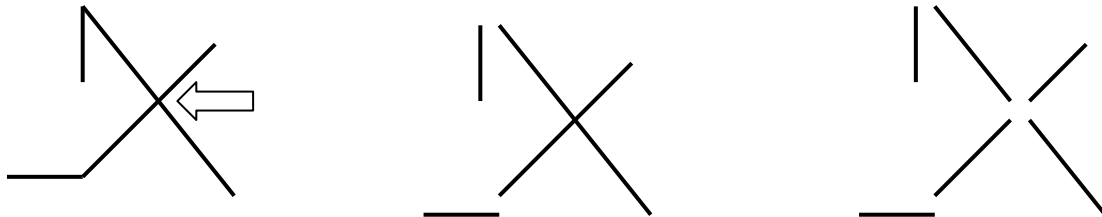
Initial situation
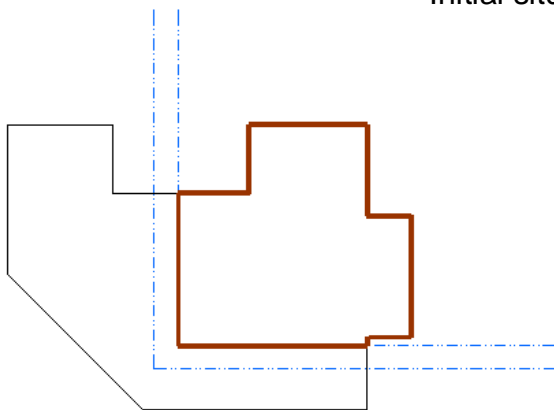
Intersections enabled
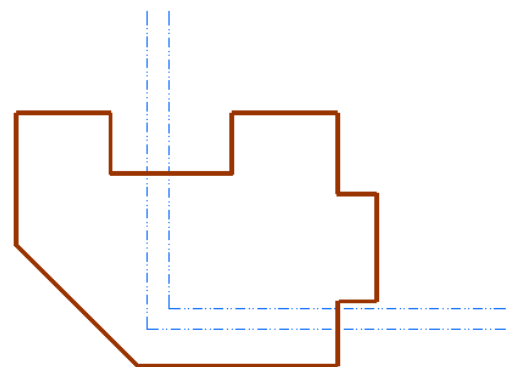
Intersections disabled

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.

Edit drawing – Plan analysis - Purge

	Command:	HYPURGE
	Ribbon tab	As-Built TS Control

The command 'HYPURGE' deletes all AutoCAD objects that As-Built for AutoCAD created temporarily. Such objects originate from the 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.

Edit drawing – Define block

	Command:	TCADBLOCKDEF
	Ribbon tab	As-Built TS Control

As-Built for AutoCAD are 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 FARO software 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:
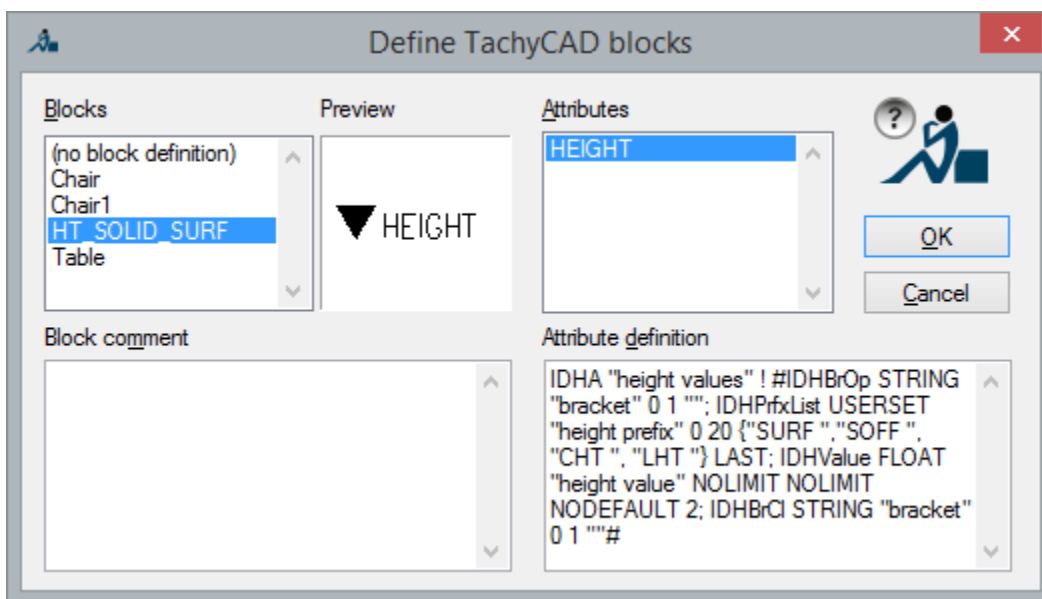
If the drawing already contains block definitions, these are displayed under "Blocks". Select the block that is to edited. Under "**Attributes**" select the attribute that is to be edited.

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, I 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 t 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:
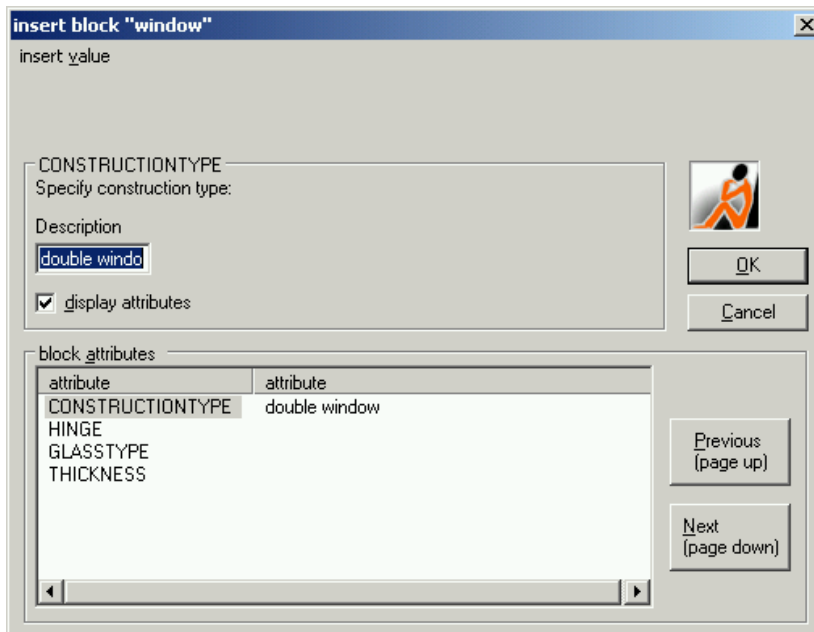
Only one element of the type STRING was defined. The parameter string of this type consists of the elements LIMITSTRING, LIMITSTRING and STRINGDEFAULTSTRING. The first two resemble the minimal or the maximal number of characters respectively which are intended for this attribute. In this example, text 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:
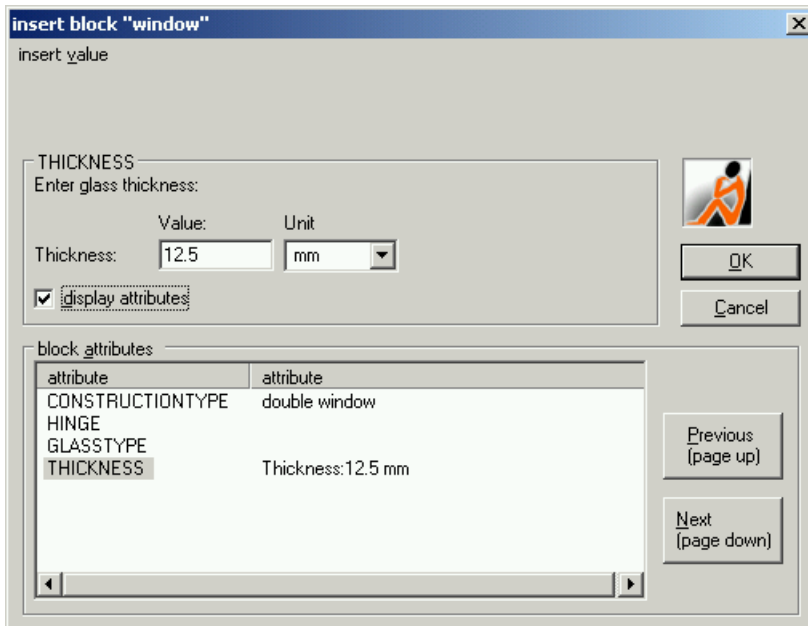
Three elements have been defined. The first element IDConst of the type CONST puts out the constant text "**Thickness**". The element IDValue of the type FLOAT allows the insertion of an decimal larger or equal to zero with one digit after the point. With the last element IDUnit of the type USERSET a list is generated, from which the units "mm" or "cm" can be selected, or an own value can be entered which has to between one and three characters long.

Individually adapt FARO block templates

The software requires, 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 the FARO Box[1].

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:
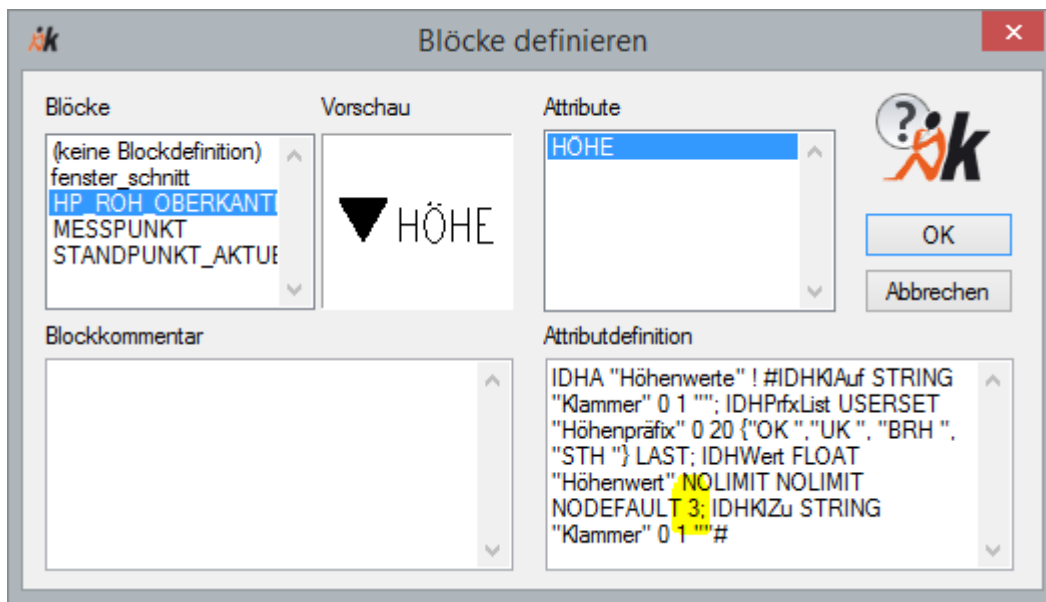
IDHA "Height value" ! #IDHKITo STRING "Bracket" 0 1 "";	→ Definition opening bracket
IDHPrfxList USERSET "Height prefix" 0 20 {"OK ","UK ", "BRH ", "STH "} LAST;	→ Definition prefix choices
IDHValue FLOAT "Height value" NOLIMIT NOLIMIT NODEFAULT 3;	6. →Definition Height value 7. From type FLOAT: NOLIMIT=Min.Value unlimited NOLIMIT= max.Value unlimited, NODEFAULT=no default value , **3=Decimal places**
IDHKITo STRING "Bracket" 0 1 ""#	→ Definition closing bracket

Edit drawing – Insert extended block

	Command:	TCADBLOCKINSERT
	Ribbon tab	As-Built TS Control

If you call up the command "blockinsert", you get a list of block definitions that exist in your drawing with a graphical preview. If the block is not in your drawing, you can select a block template in your directory structure with the button **Browse…**. In this case, no graphical preview will be shown.
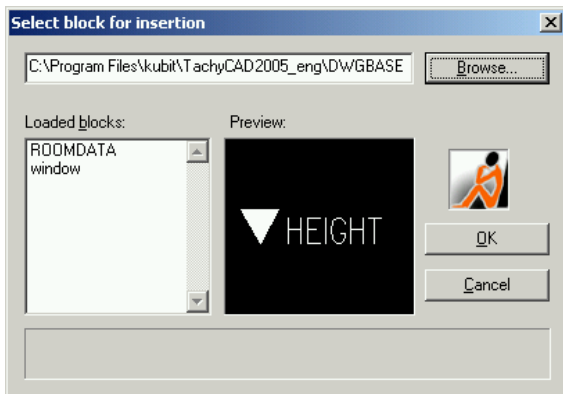
After you have selected a block definition, you reach the input mask for the attributes of the block:
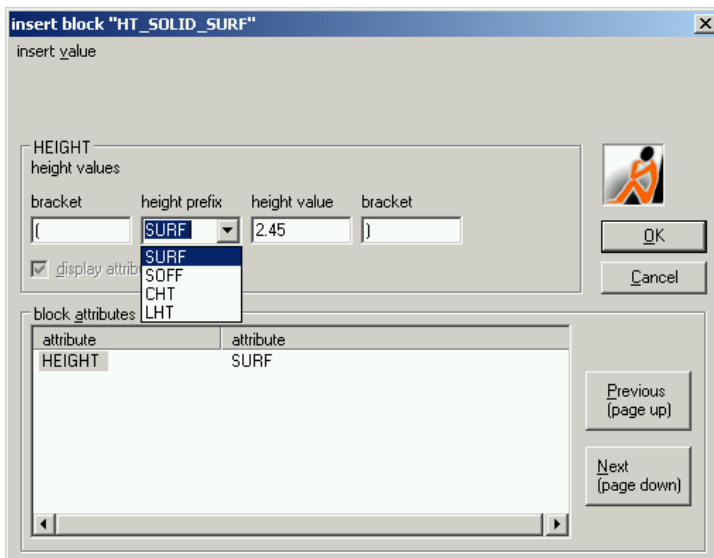
If the attributes were not edited with the command ➔**Edit drawing** – Define blocks (hence it is a simple AutoCAD attribute), the input mask is a bit more simple:

Fill in the blank fields and indicate whether the attribute should be shown or not. When you close the dialogue with **OK**, the validity of your entries is checked. Finally, you are asked to enter the position and rotation of you block in the drawing.

Edit drawing – Edit extended block

	Command:	BLOCKEDIT
	Ribbon tab	As-Built TS Control

You can edit blocks that exist in the drawing with the command "blockedit". After you have selected the respective blocks, you reach the entry mask described under **➔Edit drawing – Insert extended block.**

However, the default values of the selected block will be displayed here. You cannot change the position of the block with this command.

Clipboard – Copy

	Command:	HYCOPY
	Ribbon tab	As-Built TS Control

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 FARO clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the FARO clipboard copies/inserts FARO feature data 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.

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 ➔ **Clipboard – Paste**

        ➔ **Clipboard – Cut**

Clipboard – Cut

	Command:	HYCUT
	Ribbon tab	As-Built TS Control

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 FARO clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the FARO clipboard copies/inserts feature data 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   ➔ **Clipboard – Copy**

             ➔ **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.

Clipboard – Paste

	Command:	HYPASTE
	Ribbon tab	As-Built TS Control

With this command objects from the special As-Built for AutoCAD 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 ➔ **Clipboard – Copy** or ➔ **Clipboard – Cut**.

In contrast to the AutoCAD clipboard commands the FARO clipboard is able to copy and insert the object with its WCS coordinates, even if another UCS is currently activated.

Additionally the FARO clipboard copies/inserts feature data 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 the software 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.:

Insert - Log

Log:

```
Object Area2 renamed in Area3
Object Area2: Value of attribute Description (R2) change
Object Area1 renamed in Area4
Object Area1: Value of attribute Description (R1) change
Object Subarea4 renamed in Subarea7
Object Subarea5 renamed in Subarea8
Object Subarea6 renamed in Subarea9
Object Subarea1 renamed in Subarea10
Object Subarea2 renamed in Subarea11
Object Subarea3 renamed in Subarea12
```

OK

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   ➔ **Clipboard – Copy**

                ➔ **Clipboard – Cut**

Measure – Measure outline plan

	Command:	OUTLINEPLAN
	Ribbon tab	As-Built TS Design

The command 'outlineplan' offers the opportunity to construct an outline plan from a combination of measured points and additional measurement entries. The command is designed in a way that it differentiates between the **measurement of corner and wall points**.

The command 'outlineplan' is the ideal command to measure the outlines for the ground plan for all kinds of rooms. It offers a variety of options that are especially of advantage when surveying rooms with complicated outlines.

Let's start with a simple example. The following illustration shows a room with for sides.
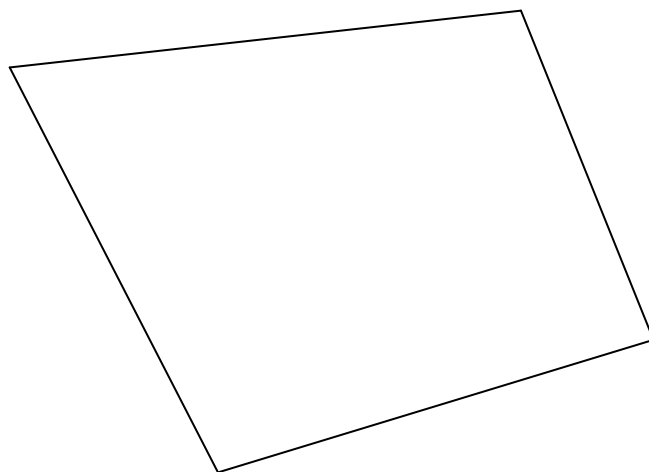
Depending on the demands, this room can be surveyed in different ways.

- Four corner points are measured.

- Two points are measured on each wall. The outline of the room is then constructed by intersection.

- A combination of corner and wall points, complemented by measurements taken by hand, is, of course, also possible.

The command processing for the survey with wall points is as follows:

Command: **Outlineplan**

First point on face or [Corner/Extend/Mode] <Corner>:	*(Measure point 1)*
Last point on face or [Corner/Extend/Undo/Mode] <Corner>:	*(Measure point 2)*
First point on face or [Extend/Perp/Undo/Mode] <Extend>:	*(Measure point 3)*
Last point on face or [Corner/Extend/Undo/Mode] <Corner>:	*(Measure point 4)*
First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>:	*(Measure point 5)*
Last point on face or [Corner/Extend/Undo/Mode] <Corner>:	*(Measure point 6)*
First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>:	*(Measure point 7)*
Last point on face or [Corner/Extend/Undo/Mode] <Corner>:	*(Measure point 8)*
First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>: c	*(Enter 'C' for Close)*
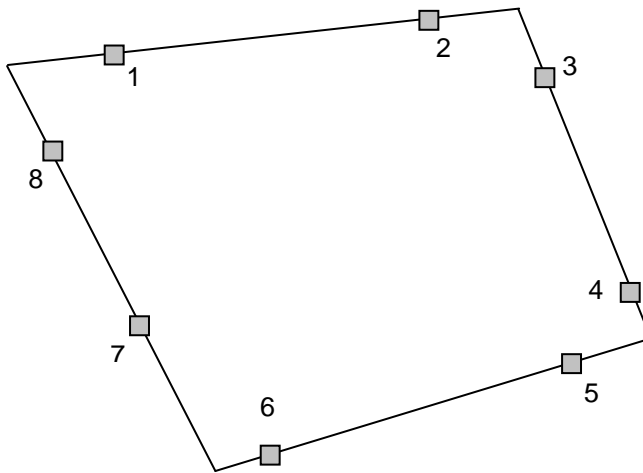
The following example shows possible uses of the command "outlineplan" in greater detail. In this case, we combine the measurement of wall and corner points with entering distance values.
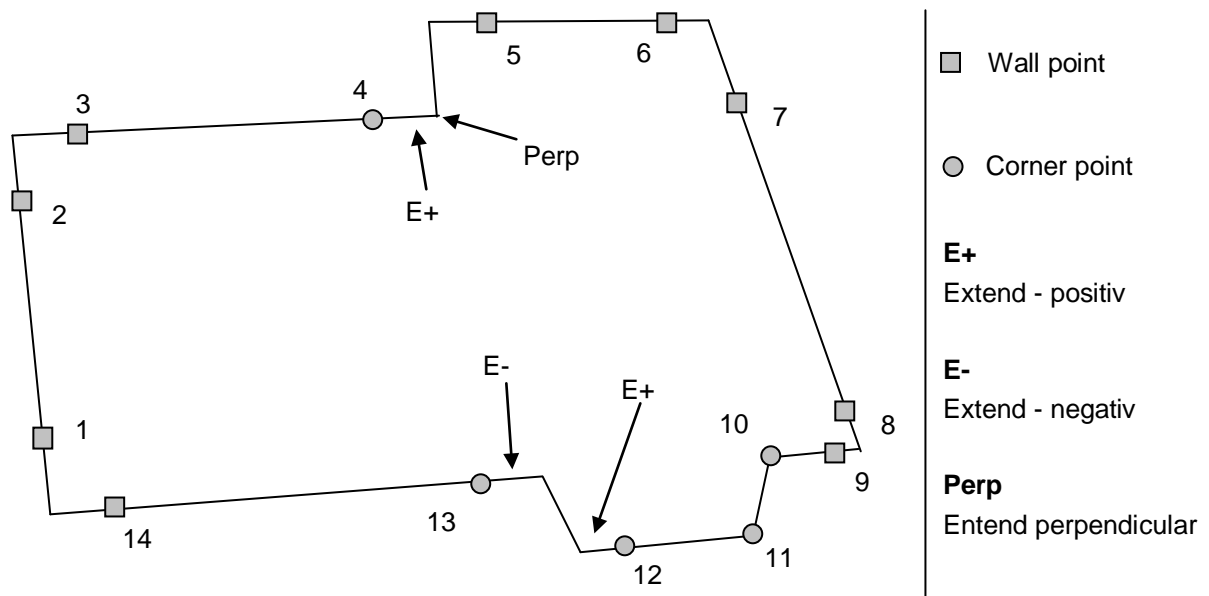
The command procedure looks as follows:

Command: **Outlineplan**

First point on face or [Corner/Extend/Mode] <Corner>:   *(Measure point 1)*

Last point on face or [Corner/Extend/Undo/Mode] <Corner>:   *(Measure point 2)*

First point on face or [Extend/Perp/Undo/Mode] <Extend>:   *(Measure point 3)*

Last point on face or [Corner/Extend/Undo/Mode] <Corner>: C

(Switch to corner mode with option 'C')

Corner point or [Face/Extend/Undo/Mode] <Face>:   *(Measure point 4)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>: e   *(Enter option 'E' for extend)*

Enter distance: **0.33**   *(Enter value for extension)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>: p   *(Enter option 'P' for perpendicular)*

First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>:   *(Measure point 5)*

Last point on face or [Corner/Extend/Undo/Mode] <Corner>:   *(Measure point 6)*

First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>:   *(Measure point 7)*

Last point on face or [Corner/Extend/Undo/Mode] <Corner>:   *(Measure point 8)*

First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>:   *(Measure point 9)*

Last point on face or [Corner/Extend/Undo/Mode] <Corner>: c

(Switch to corner mode with option 'C')

Corner point or [Face/Extend/Undo/Mode] <Face>:   *(Measure point 10)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>:   *(Measure point 11)*

Corner point or [Face/Extend/Undo/Mode] <Face>:   *(Measure point 12)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>: **e**

(Enter option 'E' for extend)

Enter distance: **0.20**   *(Enter value for extension)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>:   *(Measure point 13)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>: **e**   *(Enter option 'E' for extend)*

Enter distance: **-0.23**   *(Enter negative value for extension backwards)*

Corner point or [Face/Extend/Perp/Undo/Mode] <Face>: **f**   *(Switch to Face mode with 'F')*

Last point on face or [Corner/Extend/Perp/Undo/Mode] <Corner>:   *(Measure point 14)*

First point on face or [Extend/Perp/Close/Undo/Mode] <Extend>: **C**   *(Enter option 'C' for close)*

As the example above shows clearly, the command "outlineplan" offers numerous possibilities for the flexible measurement of outlines. The command only offers options that are possible in a particular situation.

With the option **Mode** you decide of which type the outline to be constructed should be. You can switch between lines and 2D polylines.

If you select "Lines", separate lines will be created three dimensionally in the room according to their survey points. The outline plan However, every wall stays an individual AutoCAD line object.

If you select "2D polyline", a two dimensional connected line will be constructed. This polyline has the Z coordinate (height) of the first given point.

The outline in the ground plan does not differ in both modes. Only the type of the outline is different.

Measure – Survey outline plan with fixed angles

	Command name:	TCOUTLINEPLANFIXED
	Ribbon tab	As-Built TS Design

The command "Outline plan with fixed angles" is for surveying wall outlines with perpendicular walls or with other fixed angles.

The way this command is used is similar to that of the command "Outline plan": Two points are surveyed on each wall. The individual wall lines are automatically filleted with each other, so that the room corners are created in an indirect way. The walls are created step by step whilst the command is running, additionally the wall lines are all aligned to a fixed angular grid, if they meet the tolerance conditions. One or more reference lines are essential to set up the angular grid.

"Outline plan with fixed angles" is ideal for creating building floor plans where the walls must be perpendicular to each other.

Command sequence:

The command starts with **the prompt for the reference line(s)**

> Command: **_KUBT_TC_OUTLINEPLANFIXED**
>
> Type = Line Increment angle = 45 Tolerance = 2
>
> Select line for reference direction or [last/Ucs]: *(Select one or more*
>
> *reference lines and confirm with ENTER.*
>
> *They must be single lines and not polylines!)*

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

Reference line 1

This part of the building (orange) will be aligned to reference line 1.

Reference line 2

This part of the building (green) will be aligned to reference line 2.

Whilst the command is running the **reference lines** are highlighted with blue arrows and a centre mark.
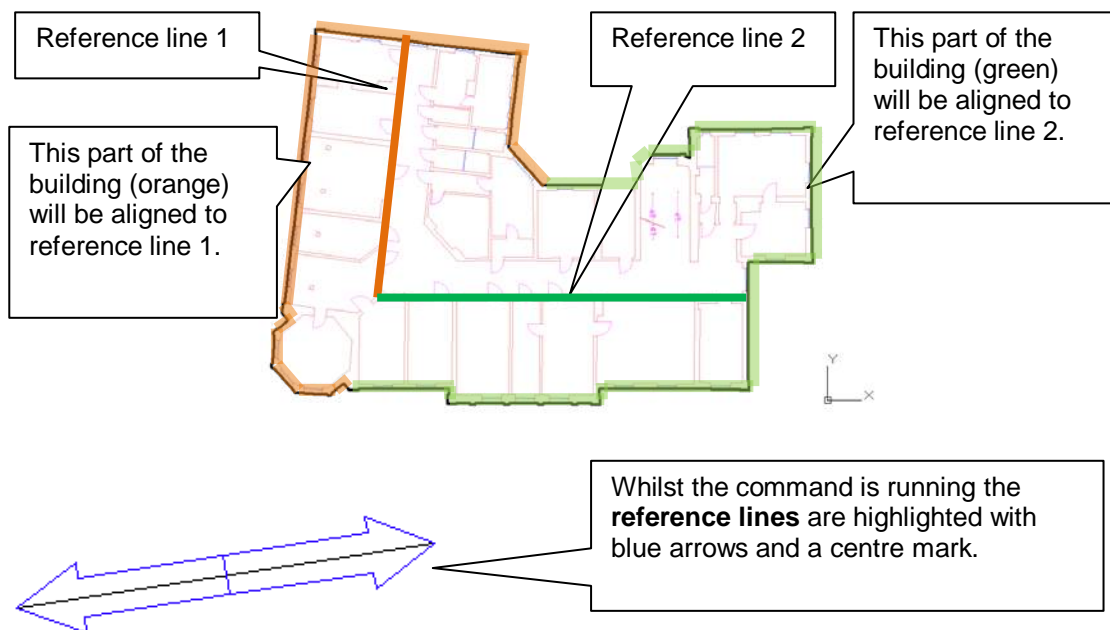
In the next step you can start observing points or go to the settings dialogue box:

Incremental angle for snap determines the angular grid for automatically aligning the wall lines. Angular steps of 90°, 45°, 30° and 22.5° can be set.

The **angular tolerance in degrees** sets the maximum angular deviation for the alignment. Should the surveyed line vary more than this tolerance from one of the reference lines, then it will not be aligned, but retain its surveyed direction.

With the option **Line type** you define what type of line the resulting contour shall be. You can switch between lines and 2D polylines.

"Lines": Single lines are created as 3D lines in space from their respective observed points. The contour is created by projecting these lines onto the XY plane of the current user coordinate system (UCS) and filleting them. Each side remains as an AutoCAD line entity.

2D -Polyline": A two dimensional connected line string is created. The polyline has as its Z coordinate (elevation) that of the first observed point.

In the floor plan there is no difference in the contour between the two variants.

The command sequence in a simple example:

The following room should be surveyed.

The three upper walls are almost perpendicular, in other words, parallel to the reference wall. The resulting walls should be drawn precisely perpendicular or parallel.

This is the reference wall, for example an important, load bearing wall of the building.

This wall is oblique and should be drawn as such.
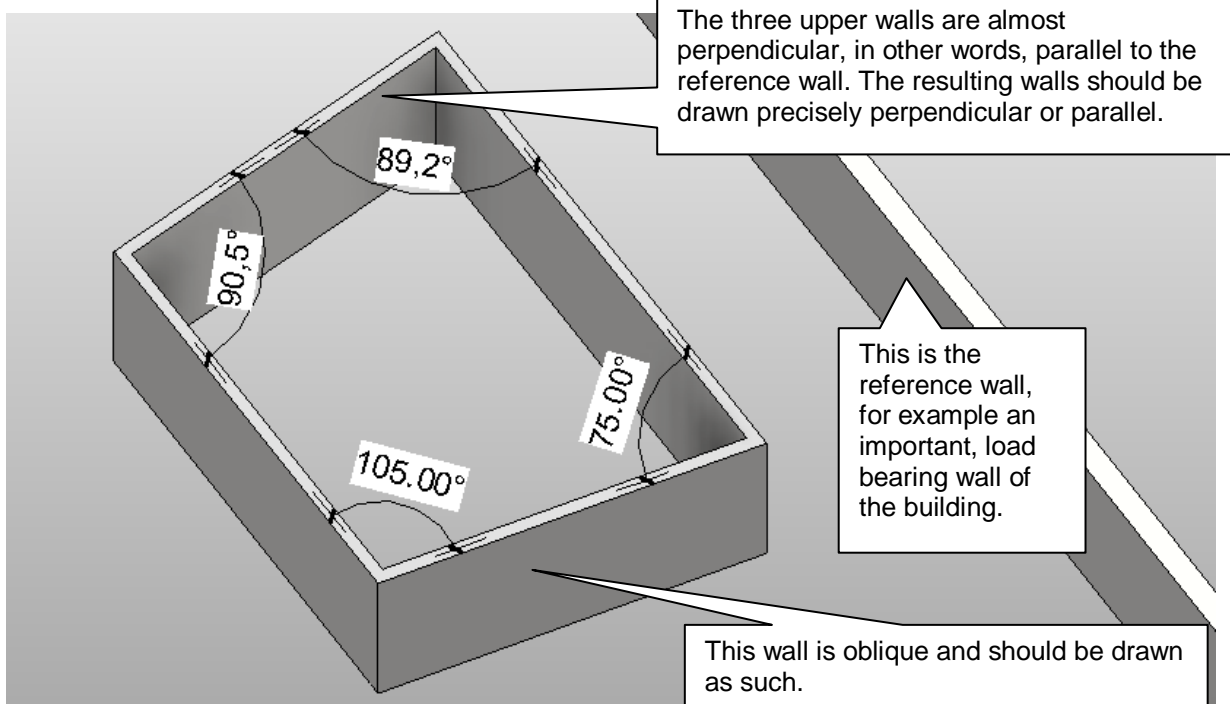
The command sequence for the four sided room 1, as shown above, runs as follows:

Command: _**KUBT_TC_OUTLINEPLANFIXED**

Type = Line Increment angle = 45 Tolerance = 2 *(the current settings)*

Select line for reference direction or [last/Ucs]:1 found

Select line for reference direction or [last/Ucs]: **ENTER** *(Select the reference wall*

and confirm with Enter)

Measure first point on wall surface or [Settings/FInish]: *(observe point 1)*

Measure last point on wall surface or [Settings/UNdo/FInish] <FInish>: *(observe point 2)*

Measure first point on wall surface or [Settings/FInish]: *(observe point 3)*

Measure last point on wall surface or [Settings/UNdo/FInish] <FInish>: *(observe point 4)*

Measure first point on wall surface or [Settings/FInish]: *(observe point 5)*

Measure last point on wall surface or [Settings/UNdo/FInish] <FInish>: *(observe point 6)*

Measure first point on wall surface or [Settings/UNdo/CLose/FInish] <FInish>: *(observe point 7)*

Measure last point on wall surface or [Settings/UNdo/FInish] <FInish>: *(observe point 8)*

Measure first point on wall surface or [Settings/UNdo/CLose/FInish] <FInish>: **C**

(closes outline)

The results look like this:

The upper, right and left walls have been aligned because the direction of the surveyed wall segments are within the angular tolerance to the reference line. They are precisely parallel or perpendicular to the reference line. Therefore the observed points lay a little to the side of the wall line.

The lower wall has not been aligned because its direction did not fit in the 90° angular grid. The observed points therefore lay precisely on the wall line.
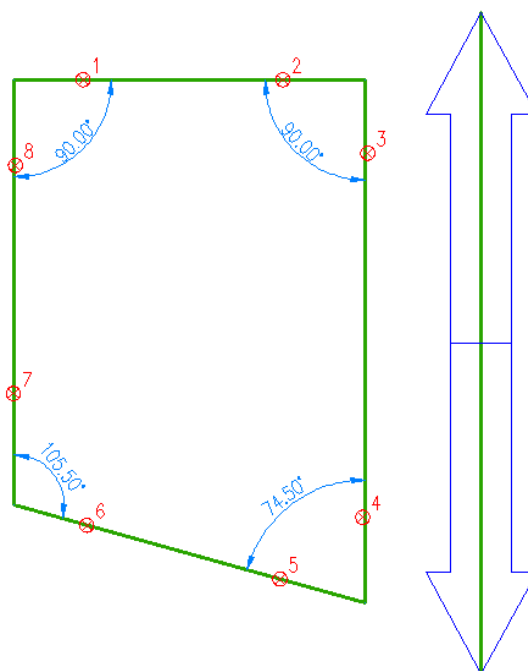
What happened in detail:

When a wall line is aligned to a fixed angle, the line is inserted in the middle of the two observed points. In this way the variances (deviations) are distributed evenly (see figure).

deviation 1 = deviation 2
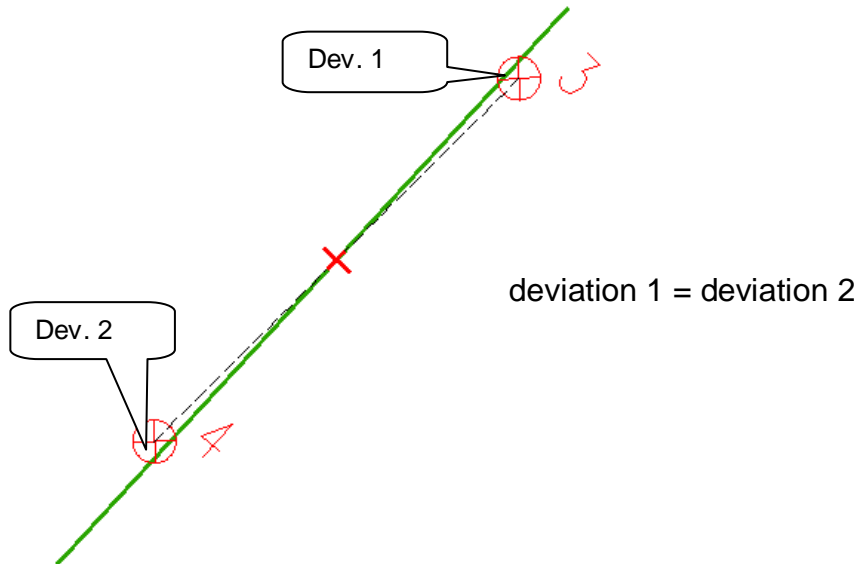
Measure – Polyline with curves

	Command:	TCADCURVEDPOLY
	Ribbon tab	As-Built TS Design

This command serves for creating organically shaped 2D polylines, where lineal and curved parts alternate. It is perfect for measuring curved wall contours, curb lines or similar.

The 2D Polyline is created on the XY-Pl deviation 1 = deviation 2

It runs in command line.

The option „Arc" switches the line from straight to curved course.

The option "Line" switches back to lineal mode.

The following example shows the command cycle:

At first tree points are measured in „line" mode. After that it is switched to "Arc" and two more points are measured. "Finish" leaves the command.

> Specify first point or [Arc] <Arc>:
>
> Specify next point or [Arc/Undo] <Arc>:
>
> Specify next point or [Finish/Arc/Undo] <Finish>:
>
> Specify next point or [Finish/Close/Arc/Undo] <Finish>:
>
> Specify next point or [Finish/Close/Arc/Undo] <Finish>: **A**
>
> Specify next point or [Finish/Close/Line/Undo] <Finish>:
>
> Specify next point or [Finish/Close/Line/Undo] <Finish>:
>
> Specify next point or [Finish/Close/Line/Undo] <Finish>: **F**

The result of this example could look like this:
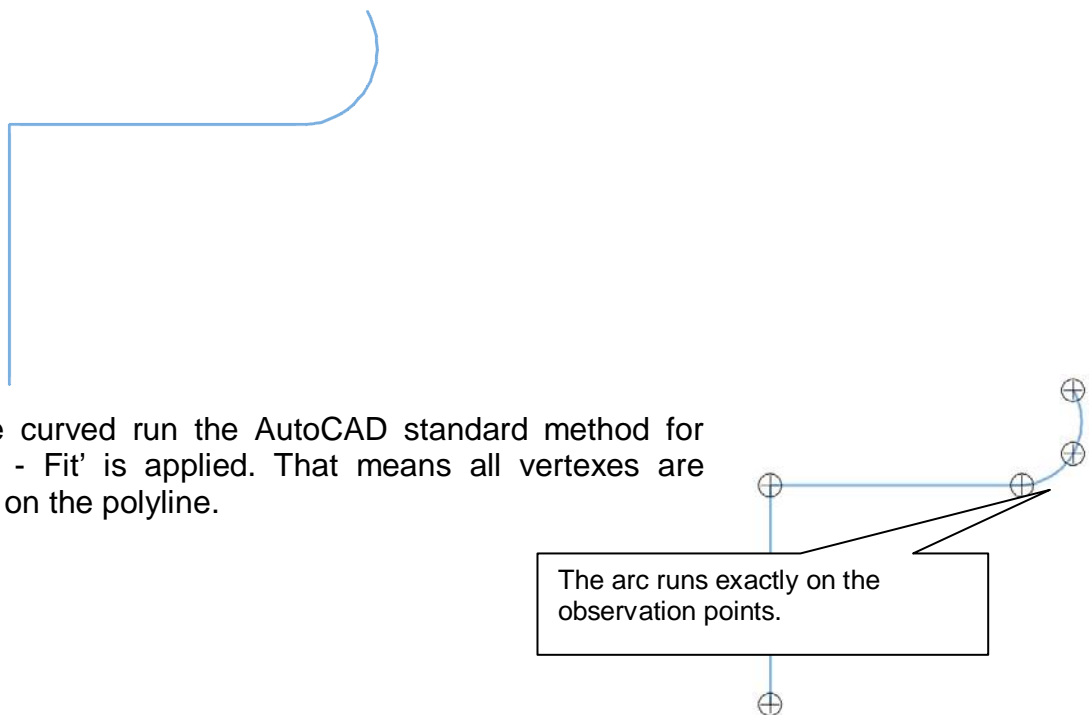
For the curved run the AutoCAD standard method for 'PEDIT - Fit' is applied. That means all vertexes are exactly on the polyline.

The arc runs exactly on the observation points.

Measure – Continue polylines

	Command:	TCPOLYCONT
	Ribbon tab	As-Built TS Design

With the "TCLINZUGFORT" command you can measure up several polylines with a total station. Existing lines or polylines will be continued by measuring new support points.

It is especially suited for the measuring of stretched objects like tracks (e.g. a street with several lines for the edge of carriageway, sidewalk, roadside ditch, etc.) with the reflector.

The measuring process has been optimised, so that the distance to be covered is as short as possible. The profile of the track will be passed through in zigzag pattern and a measuring point will always be added to a line.
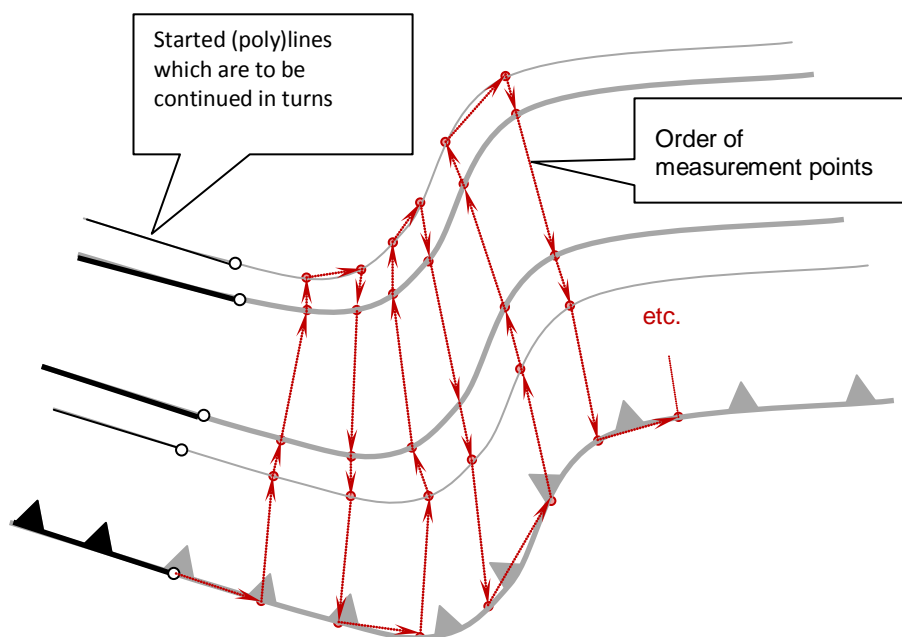
Instruction cycle:

In the run-up the first parts of all lines have to be generated. In order to do so, use any AutoCAD or As-Built command for the measurement or drawing of lines or polylines.

Now select "Continue polylines". At first you will be prompted for the line objects to be continued. Click to elements one after the other, namely:

- in the correct order (in that order they are to be measured afterwards)
- on the correct side (on that side where the drawing of the line is to be continued)

You may select 2D polylines, 3D polylines and lines.

If this command has been used before, you have the option to continue the last measurement. In that case the lines to be continued do not have to be selected again.

Select (poly)line 1:   *(Select the first line or polyline.)*

Select (poly)line 2 or [Continue/UNdo] < Continue >:   *(Select the second line or polyline.)*

...

Select (poly)line x or [Continue/UNdo] < Continue >:   *(Select the umpteenth line or polyline.)*

With the "Contiunue" option (predefined, selectable by the Enter key) you end the selection of polylines to be continued.

Now the new support points will be asked for in "zigzag order", that means 1-2-3-3-2-1-1-2-3-3-2-1-…:

New vertex for (poly)line 1 or [Skip/FInish] <FInish>:

(Measure the new support point of the first line or polyline.)

New vertex for (poly)line 2 or [Skip/UNdo/FInish] <FInish>:

(Measure the new support point of the second line or polyline.)

New vertex for (poly)line 3 or [Skip/UNdo/FInish] <FInish>:

(Measure the new support point of the third line or polyline.)

New vertex for (poly)line 3 or [Skip/UNdo/FInish] <FInish>:

(Measure the new support point of the third line or polyline.)

New vertex for (poly)line 2 or [Skip/UNdo/FInish] <FInish>:

(Measure the new support point of the second line or polyline.)

With the <Finish> option you finish the command.

With the "Skip" option it is possible to skip a support point.

Skip point
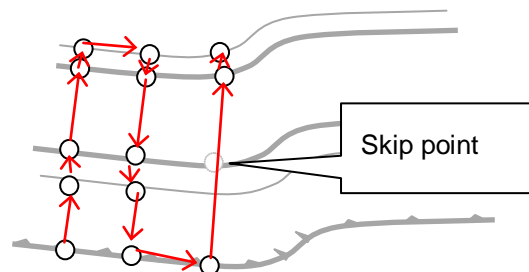
Measure – Automatic profile scan

	Command:	TCAUTOMEASURELINE
	Ribbon tab	As-Built TS Design

With the TCAUTOMEASURELINE command you can automatically survey the profile section of a room. Survey points will be scanned at regular intervals along a plane and then be connected with a polyline.

It can be used with any of the following motorised, reflectorless total stations:

- Leica Viva TS 15/TS16, TPS 1200

- Trimble S-Serien, VX

- Leica 3D Disto

- Flexijet

The results are polylines.

It allows the very precise definition of walls and ceilings.
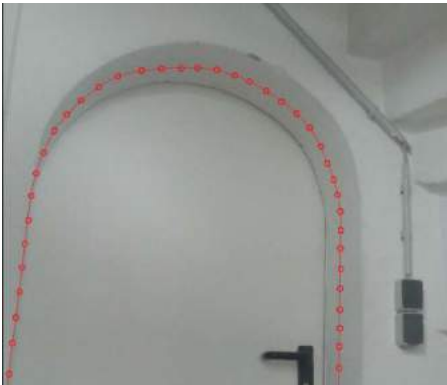
The sequence of steps in the command is roughly as shown here, each of the steps will be now be explained:
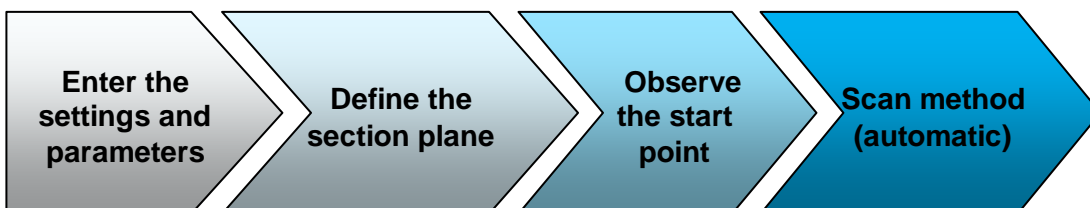
Settings and parameters

After starting the command a dialogue box is first displayed. The dialogue box is split into three sections.
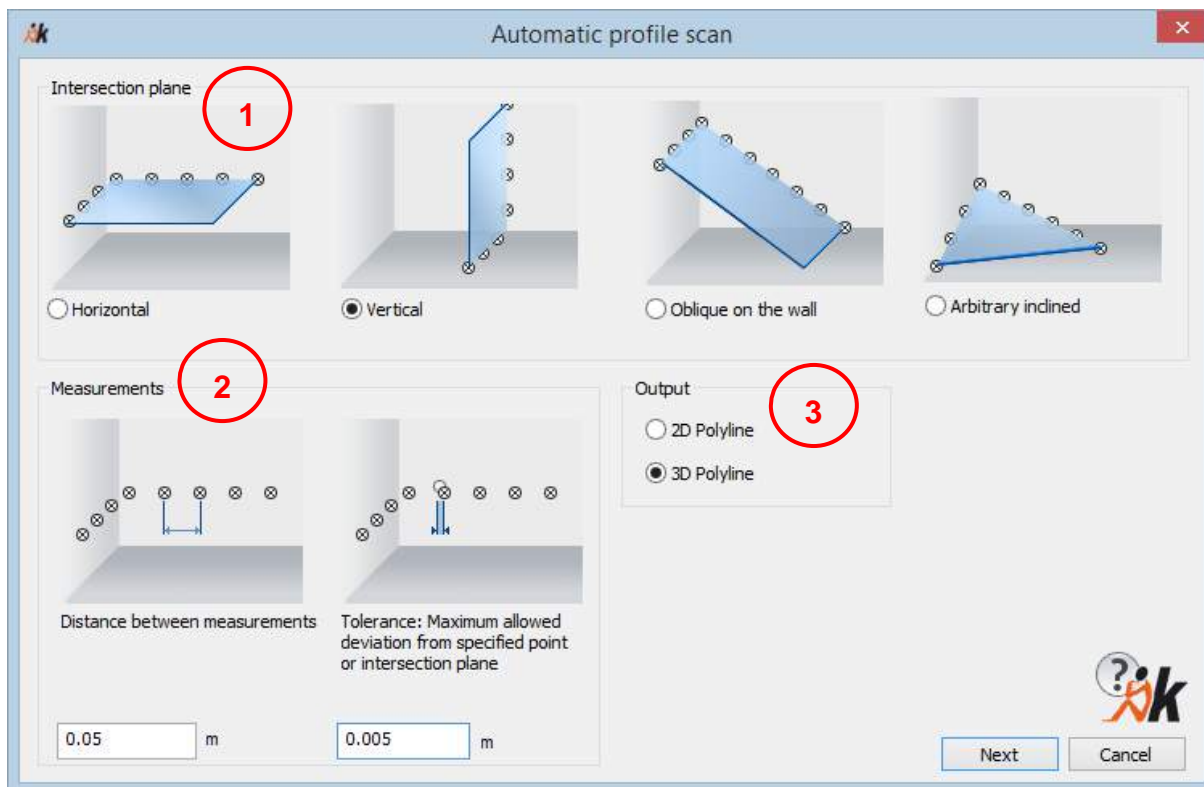
Scan plane (1)

Select the scan plane that is to be scanned. Here you can choose between:
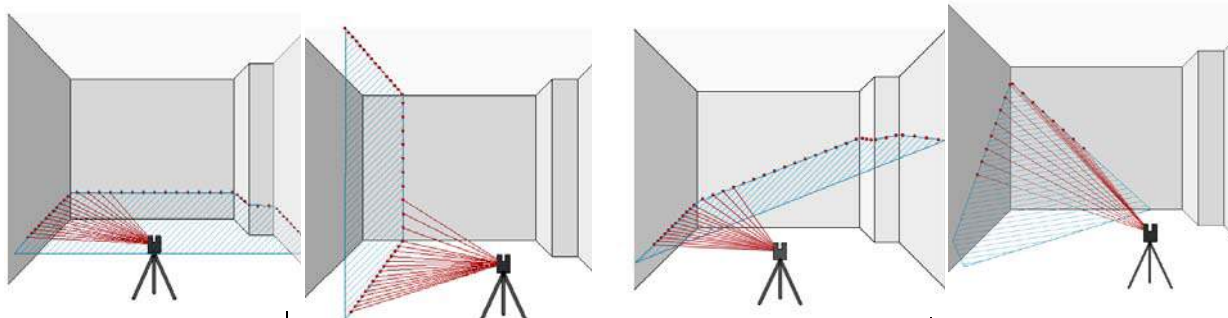
Horizontal profile

Observe a point that will be used to determine the level.

Vertical profile

Observe two points to determine the position of the vertical plane.

Oblique on the wall

Observe two points on a vertical wall to determine the position of the scan plane. The scan plane intersects the wall at right angles.

Arbitrarily inclined profile

Section plane with an arbitrary position.

Observe three points or select a kubit plane or align to a UCS

The observation of the points to define scan plane is done after the dialogue box is closed.

<u>Observed points (2)</u>

Distance between observed points: Determines the scan density, that is the distance between the scanned points.

Tolerance value: Determines how precisely the scan plane is to be observed. It is often the case that As-Built must make several observations, so that it can iteratively find the desired position. The smaller the tolerance value is, then the more precisely will the section plane be surveyed, but it will also take longer to observe the points [12].

<u>Output (3)</u>

The polyline that is created can either be a 2D or 3D polyline. With a 2D polyline the survey points are plumbed down onto the section plane

Close the dialogue box with [Next].

The settings must not be re-entered every time, As-Built remembers the values.

Define the section plane

In the next step the points are observed that are required to define the scan plane. This is dependant on the selected mmode:

- Horizontal:

   Point for the level of the horizontal scan plane or [Previous] <Previous>

   (Observation of a point for the level of the horizontal plane)

- Vertical:

   First point for the vertical scan plane or [Previous] <Previous>

   Second point for the vertical scan plane or [Previous] <Previous>

- Inclined on the wall:

   Select a point on the wall or [Previous] <Previous>

   Select a second point on the wall or [Previous] <Previous>

- Arbitrarily inclined: You have <u>three possible ways</u> of defining the section plane:

   Observe the first point for the section plane or [kubitplanE/UCs xy/Previous] <Previous>:

8. <u>Points</u>: To define the plane, observe three points with the instrument. This option makes sense when your section plane lays inclined in space.

   Observe the first point of the section plane:

   Observe the second point of the section plane:

   Observe the Third point of the section plane:

9. <u>UCS:</u> the XY plane of the current user coordinate system will be used as the scan plane.

10. <u>Kubit Plane</u> Select a kubit plane to use as the section plane. You must have previously created this special kubit object using one of the As-Built plane commands.

---
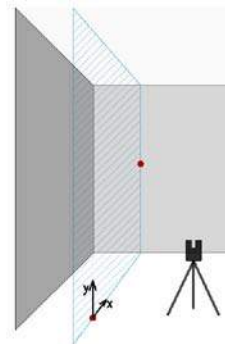
[12] As-Built checks how well the automatically observed point compares with previously calculated point. Should the difference exceed the tolerance value, then it will be re-sighted and re-observed until the difference value is met.

Select a kubit construction plane or [pOints/Ucs xy/Previous] <Previous>:

Observe the start point

Observe the start point *(1)*

Observe a point close to the start point for the scan direction: *(2)*

It is first prompted for Start point (1). That is the point from which the instrument will start the scan.

The point for the scan direction (2) defines in which direction the scan will be carried out (there are two possibilities)
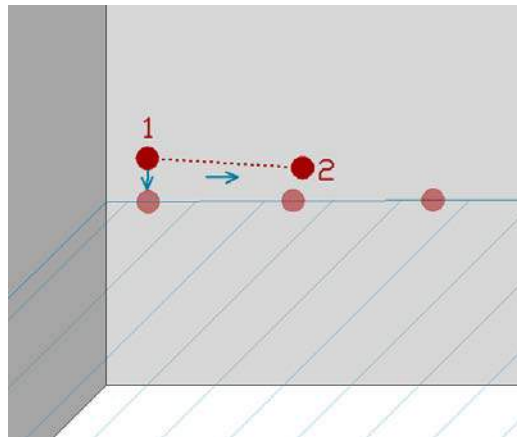
Scan method (Automatic)

Now the automatic observation procedure starts. Meanwhile an information dialogue box will appear which will alternately display "Sighting" and "Observation".

The scan procedure will continue until

- either the profile has been completely observed

- the user stops the observations by clicking the "Cancel" button in the information dialogue box. This may be useful when only part of the profile is needed or part of the profile is to be skipped.

- The scanning stops by itself when the instrument does not receive a valid distance measurement (e.g. Because the survey point lies out side of the area to be surveyed). In this situation you as the user have to intervene and skip the section which cannot be observed.

Should there be a jump in the surveyed contour (e.g. from a blind spot in an alcove), then the resulting polyline would be broken at this point.

Cancel and continue:

In order to do so click "Cancel" in the information dialogue box. On the command line you will be prompted:

Survey new wall plane?
[Yes/No] <No>:

If you select "Yes" you can skip the non observable section. You will have to observe three new start points and the survey will be continued from there.

If you select "No" the command will be cancelled.

Hint:

By default As-Built checks and corrects the distance between the points, so that it precisely matches the pre-set value.

It is possible to deactivate this accuracy check. The distance between the points will then still be used when sighting the survey points, but there will be no check to see if it has been achieved after the observation has been made. This deactivation of the accuracy check speeds up the observation procedure, but it may lead to unequal distances between the points at corners and edges.

You can find this setting in the → **As-Built settings**:
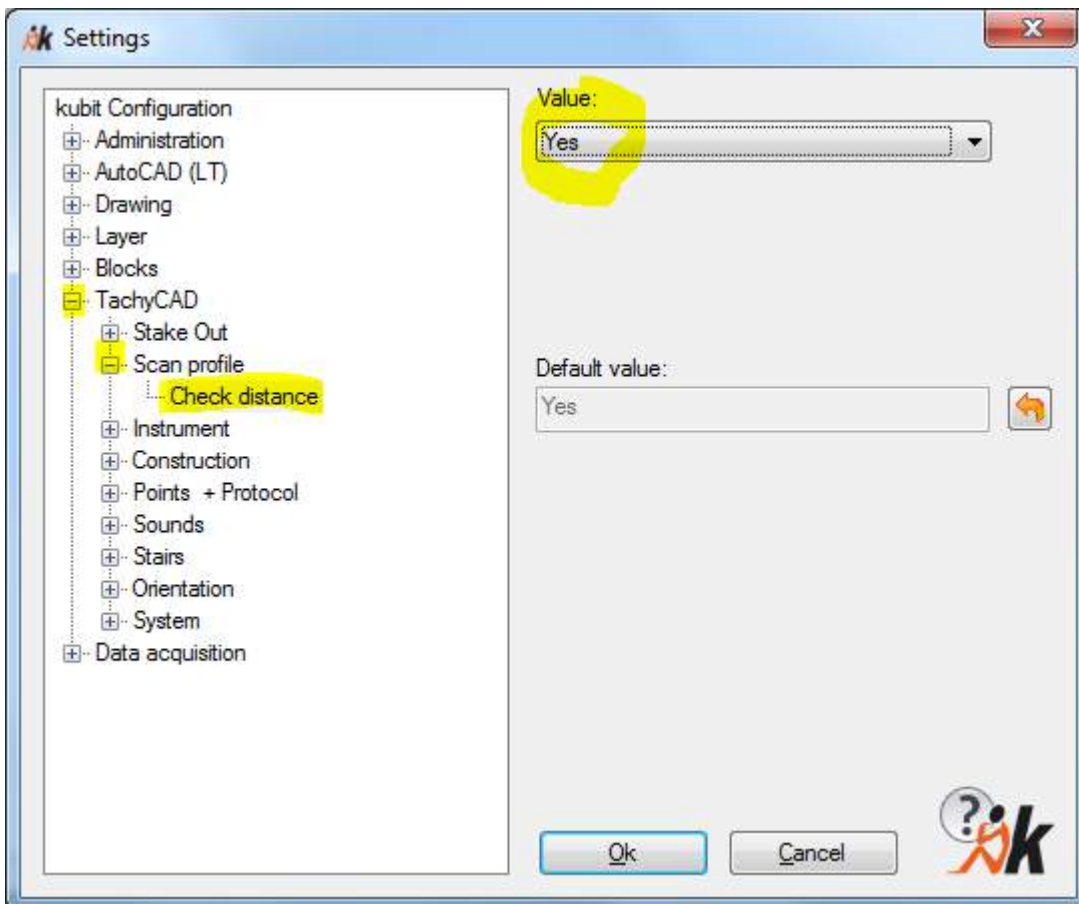
If the accuracy check stays active, the distance between the points will be precise but the observation procedure will be slower.

Measure – Perpendicular return

	Command:	PERPRETURN
	Ribbon tab	As-Built TS Design

The command "perpreturn" is used to drop a point perpendicular onto a line. It works similar to the command "Fix points to line".

> Command: **perpreturn**
>
> Select line: *(Select a line or another curve.)*
>
> Current modes: Projection=2D Break=On POints=On Line=On
>
> *(The current settings of the option 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 two determined points. 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 interrupted 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	
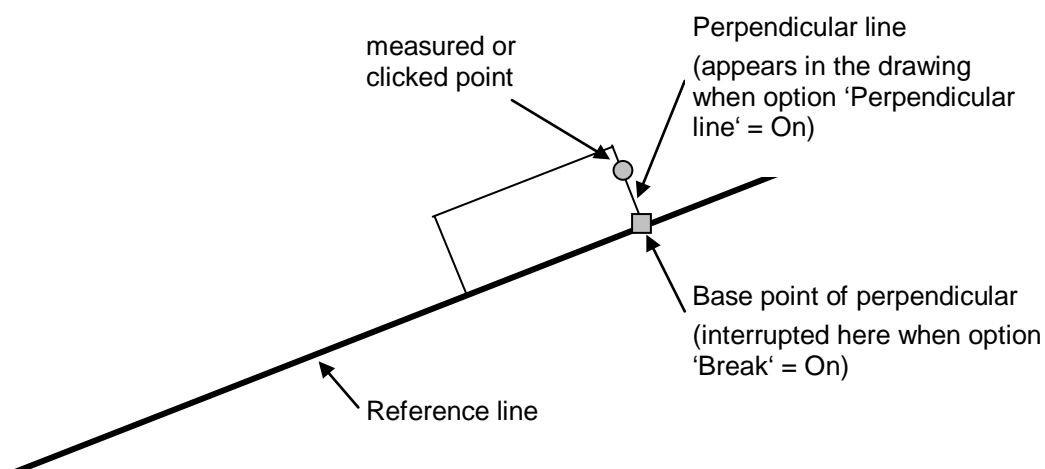
measured or clicked point

Perpendicular line (appears in the drawing when option 'Perpendicular line' = On)

Base point of perpendicular (interrupted here when option 'Break' = On)

Reference line

Measure – Fix points on a line

	Command:	FIXPOINTLINE
4.21	Ribbon tab	As-Built TS Design

The command 'fixpointline' is used to fix a progressive measurement (e.g. window or door corner, recess) on a reference line. After entering the command, you will be asked to select a reference line. It is important to select the reference line on that side from which the measurements were taken.

Command: **fixpointline**

Select line: *(Select a line or an arc)*

Current modes: Projection=2D Break=On POints=On

(The current settings of the option are displayed)

Enter distance from corner or [Projection/Break/POints/Undo]: **1.45**

(Enter the progressive measurement)

options

Option	Value	Description
Projection	2D	The measurement to be fixed corresponds to the projection of the line into the XY plane of the current UCS.
	3D	The measurement to be fixed corresponds to the line itself.
Break	On	The line is interrupted at the respective point.
	Off	
Points	On	An AutoCAD point is inserted at the respective point.
	Off	

Measure – Arc through 3 3D Points

	Command:	3POINTARC
	Ribbon tab	As-Built TS Design

The command "3Pointarc" 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 As-Built for AutoCAD command "3POINTARC" 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 ➔**Measure – Circle through 3 3D 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.

Important: If the geometries of the objects to be measured do not resemble an arc (e.g. vaults or barrel vaults), use the command "Rotarch". This command allows to measure arcs with more than three points (➔**Annotation - Create rotated arch plan**).

Measure – Circle through 3 3D points

	Command:	3POINTCIRC
	Ribbon tab	As-Built TS Design

The command '3Pointcirc' 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 As-Built for AutoCAD command '3PPointcirc' 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 measured. Its base should take the shape of a circle in the ground plan. Three points are measured on the surface of the cylinder, (usually at different heights). The following outline shows the difference in reaction between both modes.

3D mode:

The circle is constructed three dimensional and runs through the three points

The circle takes the shape of an ellipsis in the ground plan.

Thus, this mode should not be used for ground plans but can be of use in different situations.

GROUNDPLAN.DWG

2D mode:

The circle is constructed in a plane parallel to the XY plane of the current UCS (parallel to the ground plan). The two points measured last are set perpendicular to the height of the first point.

The circle takes the shape of a cicle in the ground plan.

This mode is useful for constructing 2D ground plans.

GROUNDPLAN.DWG

Construction 2points / 3points:

For 3points, any three points on the circle will be measured. With this the circle has been well-defined. This option is perfect for the measurement of columns and the like.
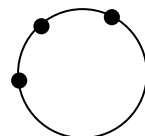
For 2points, two points have to be measured which are exactly opposite from each other. This option is advisable for the measurement of circular objects on the ground like covers or post holes (archaeology). This option is only available in the 2D mode.
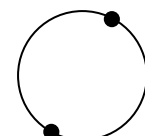
Measure – Measure segments

	Command:	STARTSTOPLINE
	Ribbon tab	As-Built TS Design

The command "startstopline" allows the simple **measurement of line and polyline segments**. With this command you can consecutively measure strings of lines, without any further input from your notebook PC.

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: _**TCADSTARTSTOPLINE**

Number of points per segment or <2>:

Select object type or [2D-Polyline/3D-Polyline/Line/UNdo] <Line>:

Measure segment point 1: or [UNdo]:

Measure segment point 2: or [UNdo]:

Measure segment point 1: or [UNdo]:

Measure segment point 2: or [UNdo]:

Measure segment point 1: 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:
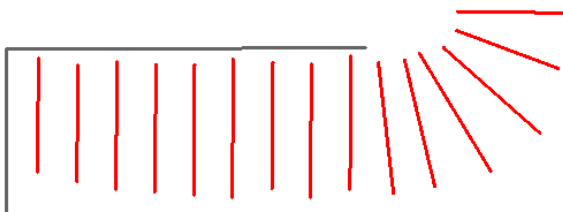
Measure – Measure rectangle

	Command:	TCADRECT
	Ribbon tab	As-Built TS Design

The command "tcadrect" allows you to measure a rectangle (e.g. sectional view of a pillar) directly.
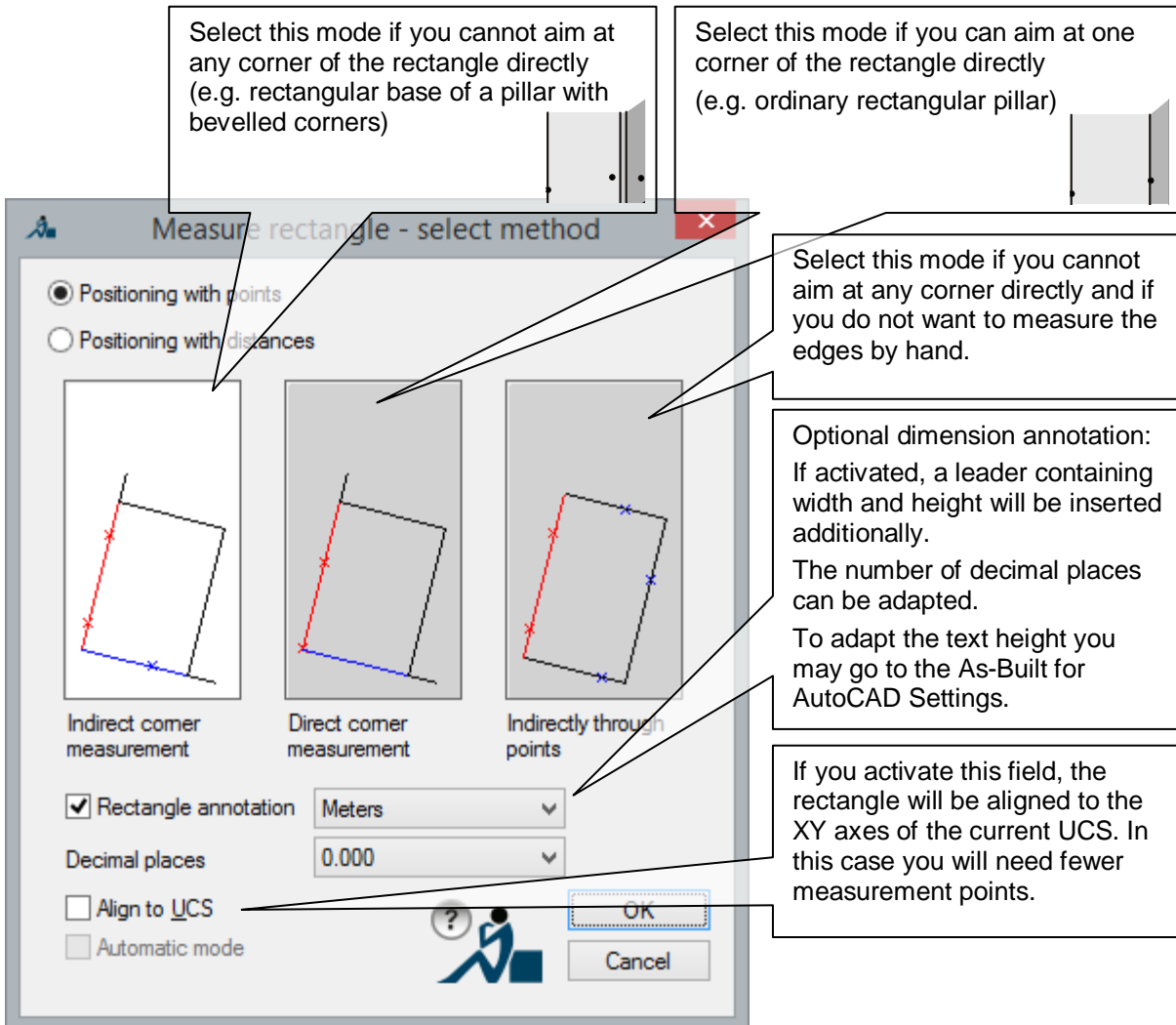
Previous hint: This command contains options for point measurement with a total station and options for distance measurement.

The rectangle always lies in the XY plane of the current UCS with the Z coordinate of the first measured corner point. When selecting the option "Positioning with distances" the Z coordinate will be set to Zero.

There are different measurement modes depending on the corners that you can aim at with the total station or a hand-held laser.

Command: **tcadrect**

Running the command opens following window:

Select this mode if you cannot aim at any corner of the rectangle directly (e.g. rectangular base of a pillar with bevelled corners)

Select this mode if you can aim at one corner of the rectangle directly (e.g. ordinary rectangular pillar)

Select this mode if you cannot aim at any corner directly and if you do not want to measure the edges by hand.

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 for AutoCAD Settings.

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.

Measure rectangle - select method

◉ Positioning with points
◯ Positioning with distances

Indirect corner measurement

Direct corner measurement

Indirectly through points

☑ Rectangle annotation   Meters
Decimal places   0.000
☐ Align to UCS
☐ Automatic mode

OK
Cancel

The option "<u>Positioning with points</u>"is recommended for the measurement with a total station. Select "Positioning with distances" if you use a hand-held laser for the measurement.

Select this mode
if you can indicate two lines parallel to the edges of the rectangle.

Select this mode
if you can indicate one parallel reference line and the distance to one reference point.
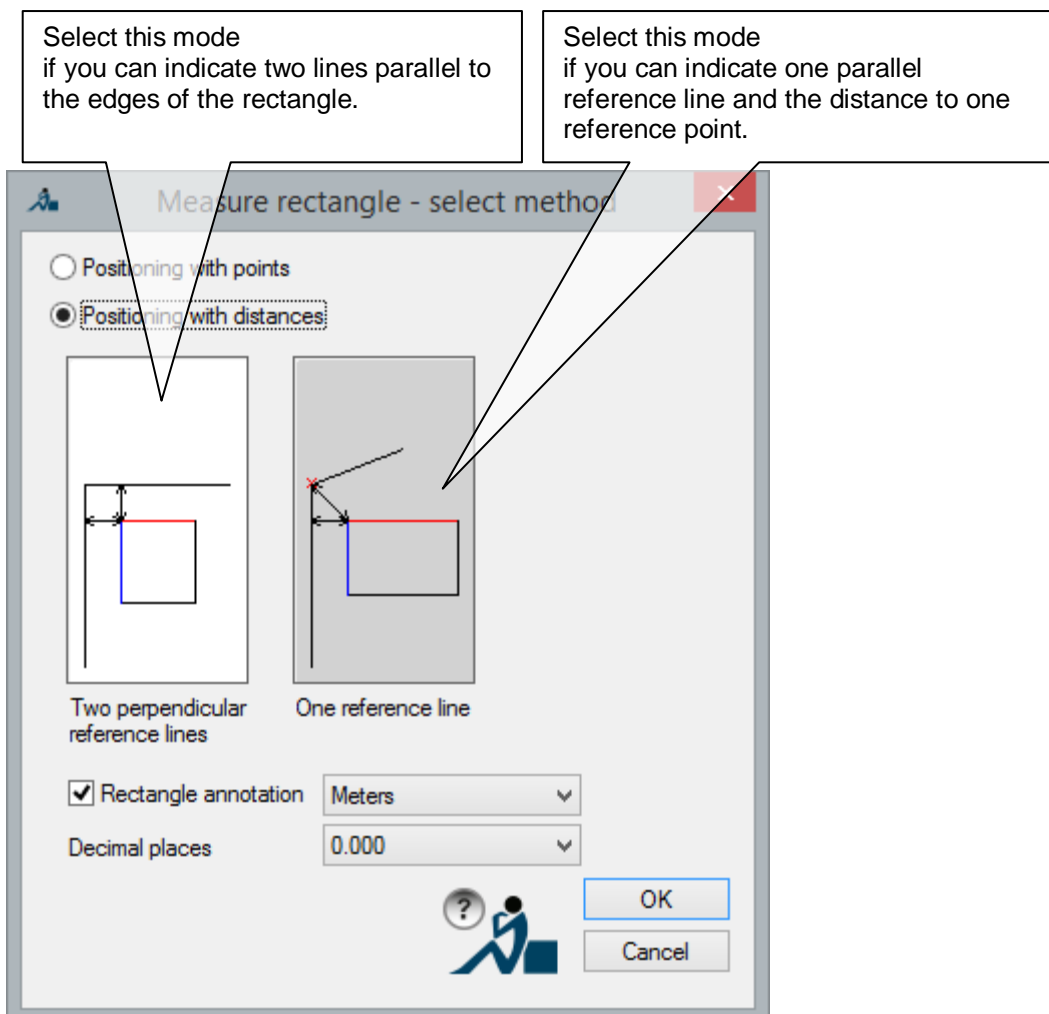
You select a mode by clicking on its icon. Confirm with **OK**.

Now you are asked to measure the points of the rectangle. Depending on the mode, the process differs:

Mode "Indirect corner measurement":

Mode = indirect corner measurement *(the current mode is displayed*

First point on first side or [Undo]: *(Measure one point on one side of the rectangle)*

Second point on first side or [Undo]: *(Measure a second point on this rectangle side)*

Point on the second side or [Undo]: *(Measure a point on an adjoining side of the rectangle)*

Following dialogue appears:

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 measurement":

Mode = direct corner measurement	*(the current mode is displayed)*
Corner of rectangle or [Undo]:	*(Measure one corner of the rectangle)*
Point on rectangle side or [Undo]:	*(Measure a point on one edge of rectangle)*

Following dialogue appears:

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.

Measure rectangle – define lengths

Red length   0.5   ☐ Calculate from measured points
Blue length   0.7   ☐ Calculate from measured points

Aspect ratio
◉ Rectangle
○ Square red length
○ Square blue length

Choice of side
◉ Draw to the right
○ Draw to the left

☐ Keep settings for next measurements (automatic mode)

OK
Cancel

Mode "Indirectly through points":

Mode = indirectly through points	*(the current mode is displayed)*
First point on first side or [Undo]:	*(Measure a point on one edge of the rectangle)*
Second point on first side or [Undo]:	*(Measure another point on this edge of the rectangle)*
Point on the second side or [Undo]:	*(Measure a point on an adjoining edge of rectangle)*
Point on third side or [Undo]:	*(Measure a point on an adjoining edge of rectangle)*
Point on fourth side or [Undo]:	*(Measure 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)*
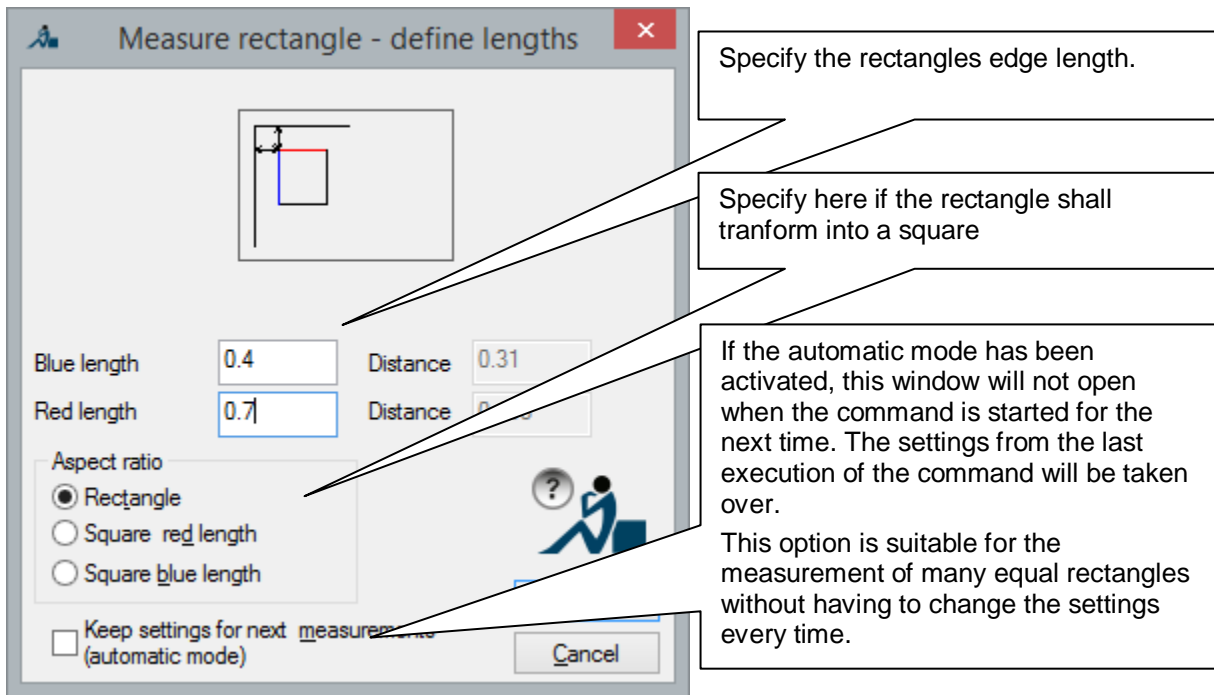
The following window will appear:

Specify the rectangles edge length.

Specify here if the rectangle shall tranform into a square

If the automatic mode has been activated, this window will not open when the command is started for the next time. The settings from the last execution of the command will be taken over.

This option is suitable for the measurement of many equal rectangles without having to change the settings every time.

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)*
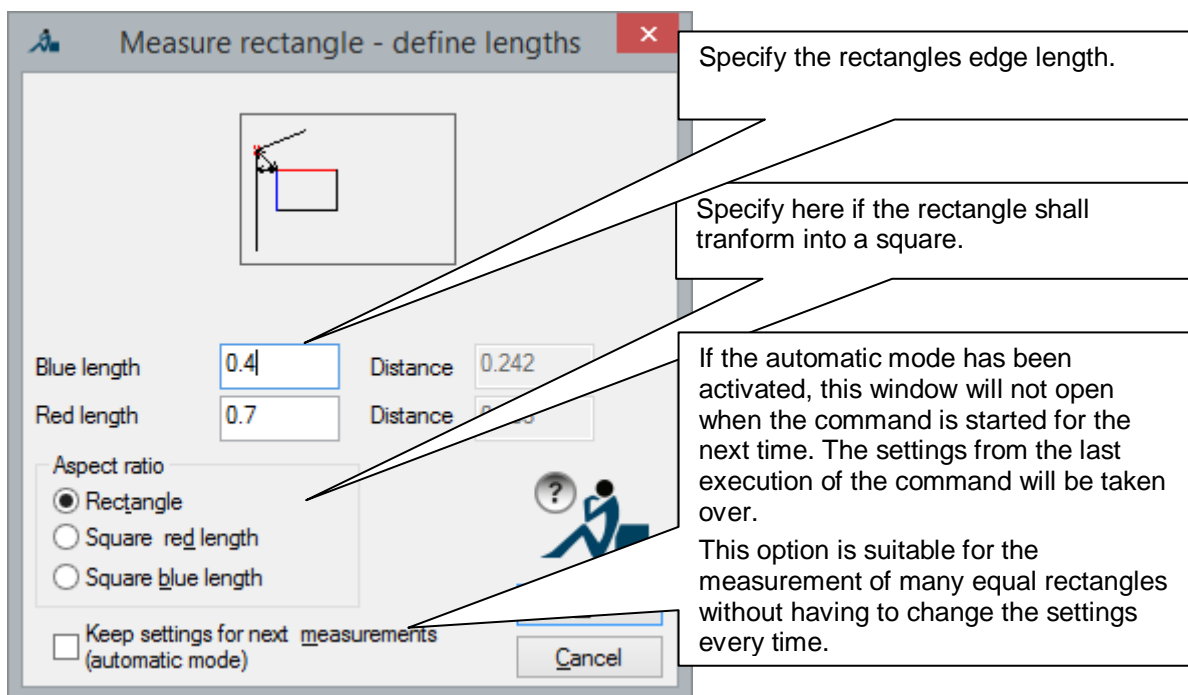
The following window will appear:

Specify the rectangles edge length.

Specify here if the rectangle shall tranform into a square.

If the automatic mode has been activated, this window will not open when the command is started for the next time. The settings from the last execution of the command will be taken over.

This option is suitable for the measurement of many equal rectangles without having to change the settings every time.

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)*
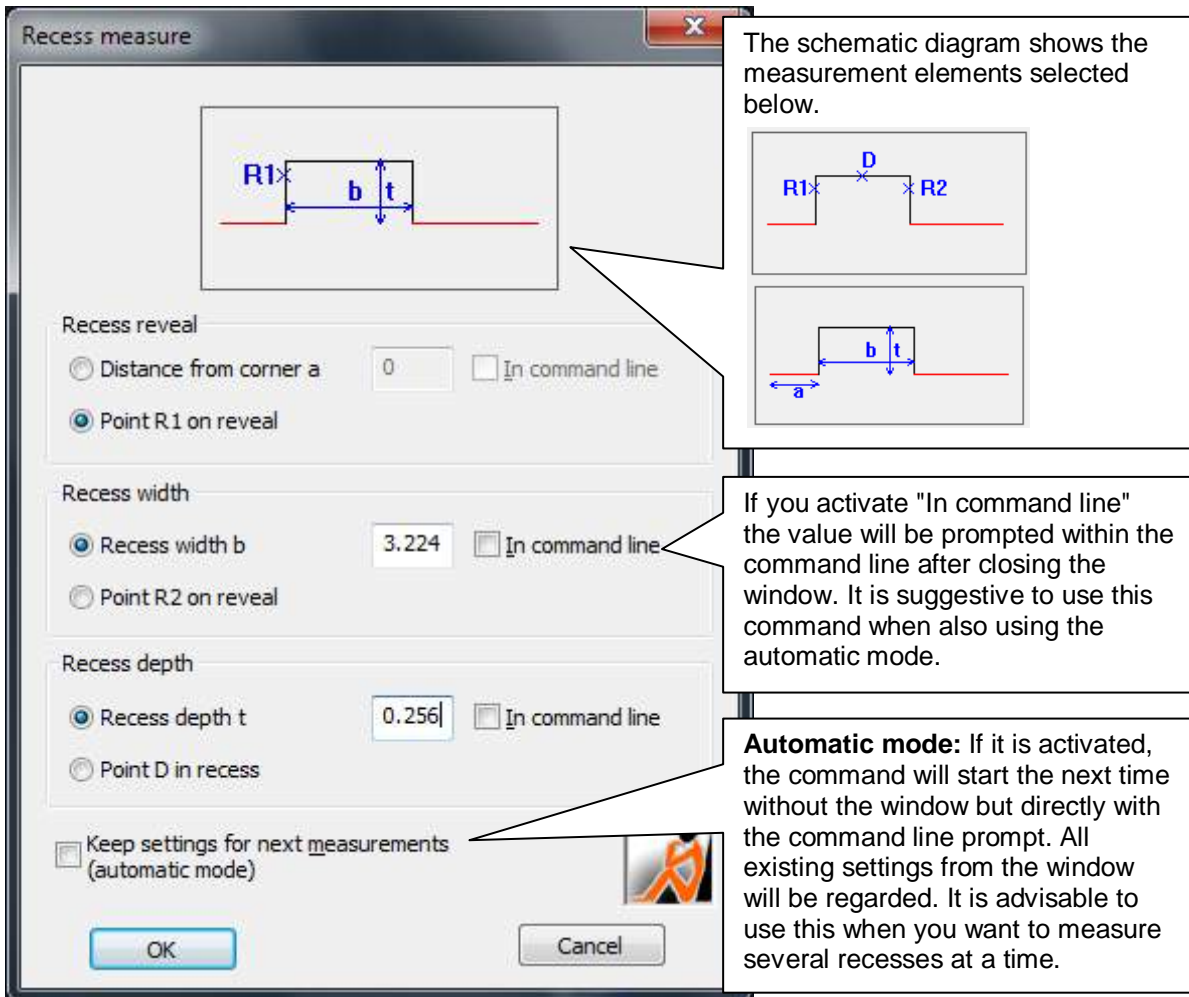
Measure – Measure Recess / Ledge

	Command:	TCADRECESSMEASURE
	Ribbon tab	As-Built TS Design

With the "tcadrecessmeasure" 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. You may also combine the manual and the total station measurement according to the spatial situation.

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 schematic diagram shows the measurement elements selected below.

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.

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.

The wall line should be clicked here...

...and therefore the distance will be measured from here.
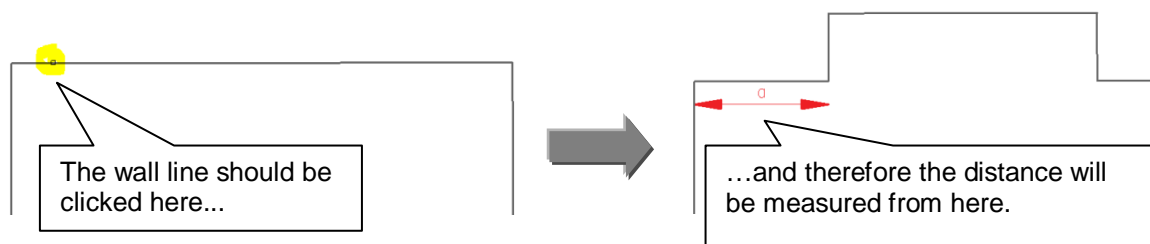
This value will either be sent into the field with a Bluetooth laser distance meter or entered manually.

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.

Measure – Measure and insert door

	Command:	INSERTDOOR
	Ribbon tab	As-Built TS Design
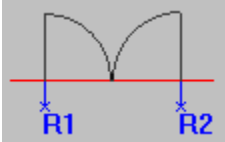
The command "insertdoor" draws a door symbol into an existing wall line and annotates it optionally with height and width of the opening.

Previous hint: 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:

The button **Change door type…** opens a selection dialogue:
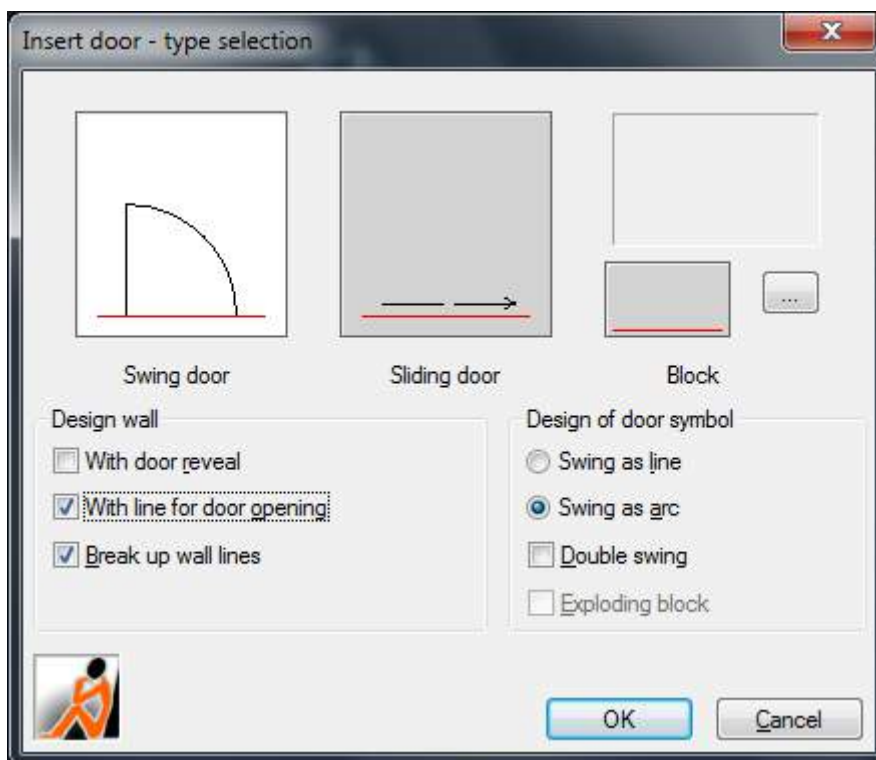
By clicking an image you select a type (white background). You can choose between swing doors and sliding doors but you also have the chance to select a door symbol you defined yourself (AutoCAD block). For the latter you have to design the symbol beforehand and save it in a separate drawing. With the 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 measure both reveals of the door choose this option. After closing the dialogue the following is prompted in the command line:

Select wall line or [two Points] <Points>:	*(Click the line)*
Select point on the reveal or [Undo]:	*(Measure point on first reveal)*
Select second point on the reveal or [Undo]:	*(Measure point on second reveal)*
Select point for direction of door swing or [Undo]:	*(Click point for opening direction)*

- Option **Point on one side (R1 or R2):** If you are able to measure only one reveal of the door 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>:	*(Click the line)*
Select point on the reveal or [Undo]:	*(Measure point on reveal)*
Enter value for door width or [Undo]: **1.1**	*(Enter the value)*
Select side or [Undo]:	*(Click side for opening direction)*
Select point for direction of door swing or [Undo]:	*(Click point for opening direction)*

- Option **Measure of length:** Here you do not have to measure any point with your instrument. 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.
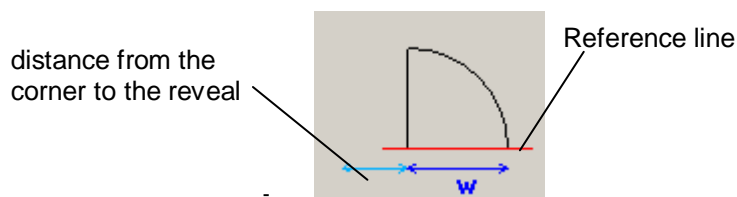
distance from the corner to the reveal

Reference line

w

-

After closing the dialogue the following is prompted in the command line:

Select wall line or [two Points] <Points>:	*(Click the reference line)*
Enter distance from the corner to the reveal [Undo]: **0.57**	*(Enter measured distance)*
Enter value for door width or [Undo]: **1.3**	*(Enter the value)*
Select point for direction of door swing or [Undo]:	*(Click point for opening direction)*

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 measure them.

- **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]:	*(Measure point on lower edge of the door)*
Measure point on the soffit or [Undo]:	*(Measure point on upper edge of the door)*
Height of opening: 2.34216	*(calculated door height)*
Select first point for opening width or [Undo]:	*(Point on one side edge of the door)*
Select second point for opening width or [Undo]:	*(Point on other side edge of the door)*

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 measurements (automatic mode)**". This option is intended to measure 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 measurements 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 FARO 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 **Measuerements** is clicked, a window opens where one can choose if the asymmetry shall be determined by measuring a point or entering a value for the length.
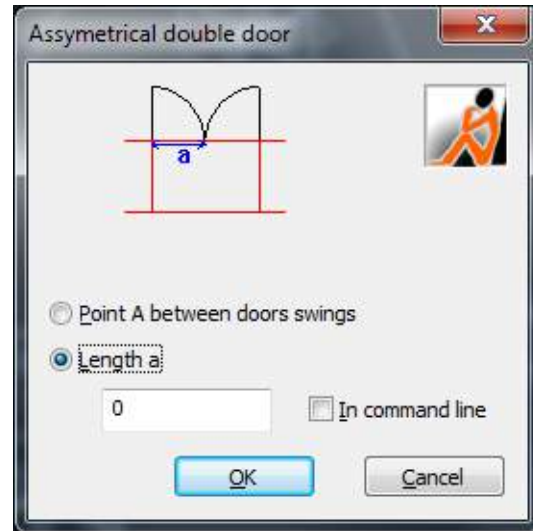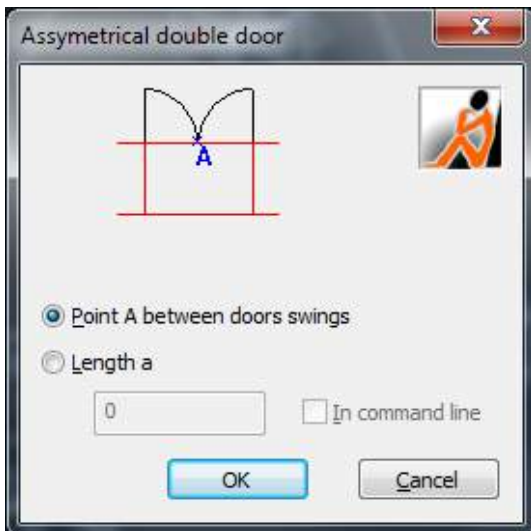
Create feature data objects

This option can only be applied, when you use the As-Built feature data management. If the used structure definition has a predefined class "Door", a feature data door object will automatically be generated additionally. 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.

Measure – Measure and insert window

	Command:	INSERTWINDOW
	Ribbon tab	As-Built TS Design

The command "insertwindow" inserts a window symbol into an existing wall line.

Previous hint: 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:
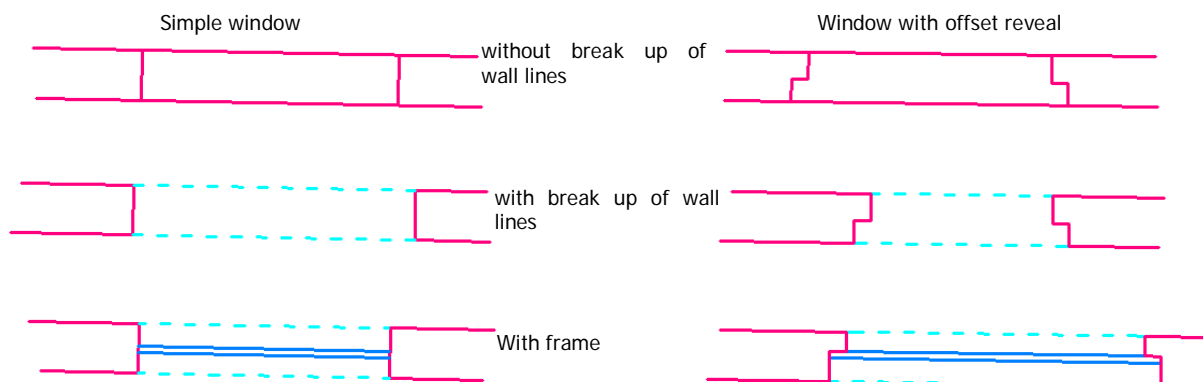
General rules for FARO windows

- A reference line has to be set. The **reference side** is that side of the wall from which the window is measured. (The side where you stand with your total station.) 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 As-Built for AutoCAD command ➔**Settings …**

Workflow

11. When running the command the following dialogue opens:
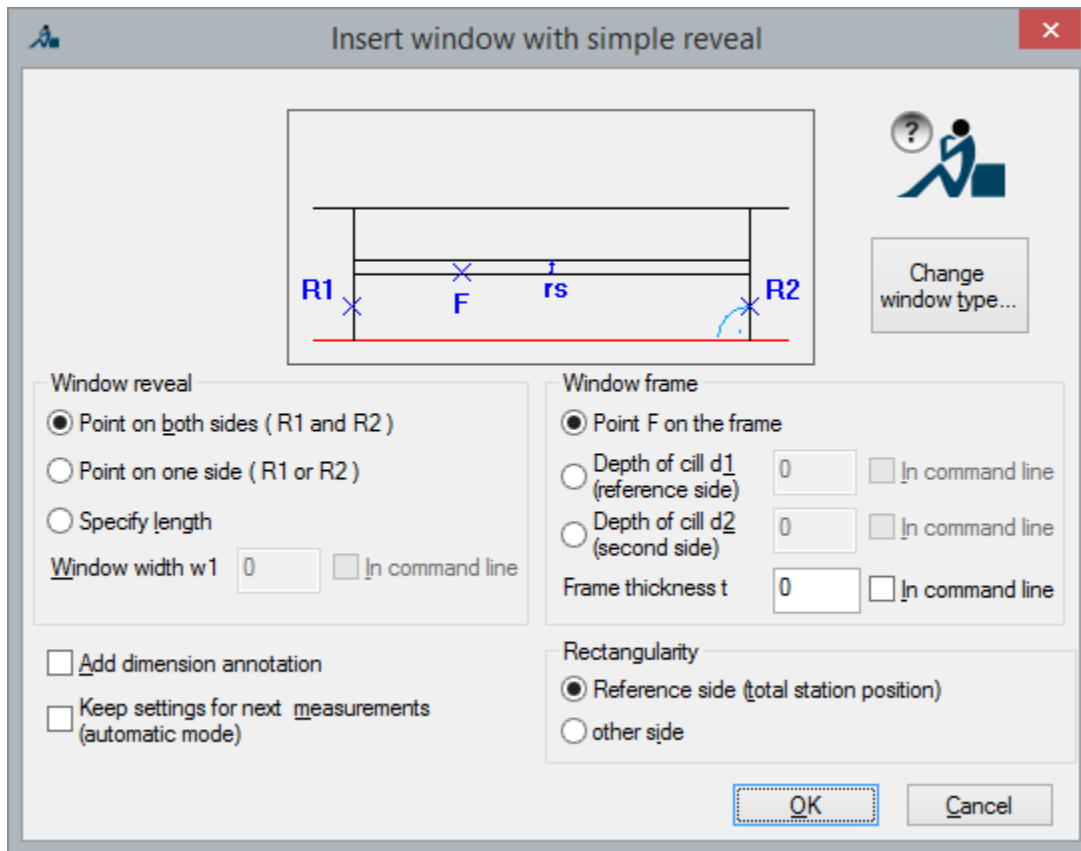
Here you set how the window is measured/inserted. The several possibilities are explained later on. With the button **Change window type** you can select the **representation of the window**:

Select the window type by clicking on one of the previews.

Here you define if the window shall be drawn with or without glass pane.

without frame

Here you decide whether the lines of walls are broken up or kept as they are.
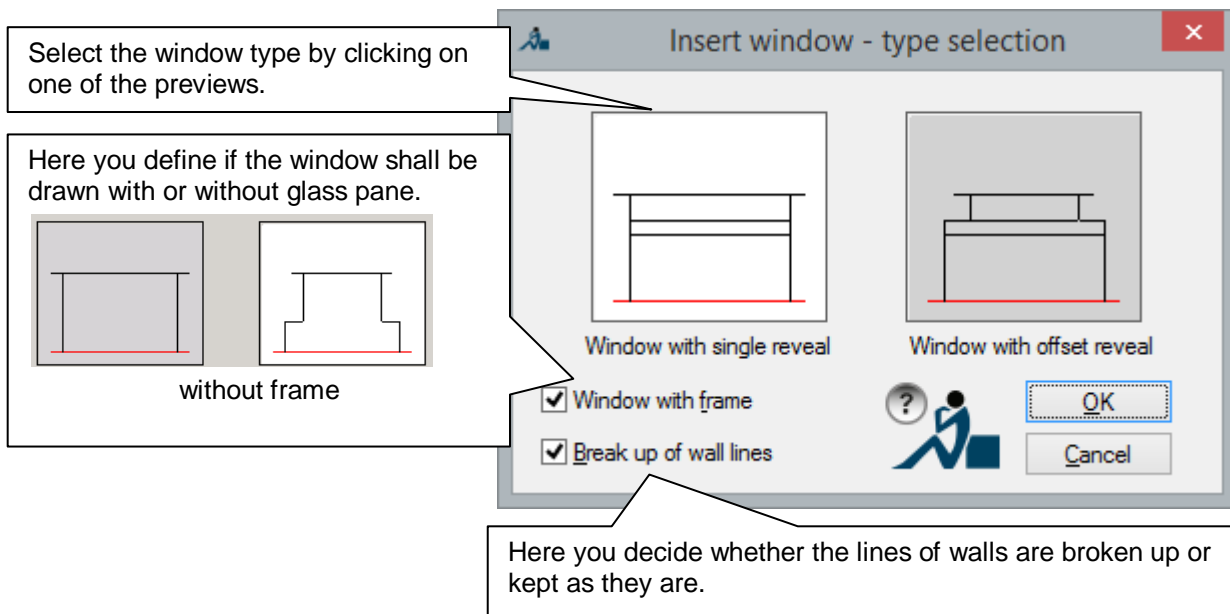
Confirm your selection with **OK**. You get back to the main dialogue.

12. 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: *(Click line or polyline for reference wall)*
> Enter wall thickness or [Line/Position/Undo] <Line>: **.24** *(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')*

13. Following the values and measured points are prompted according to the settings in the dialogue.

Simple Window

The window is assumed to be symmetric. Is has no rabbet and the reveal is rectangular to one of the wall lines.
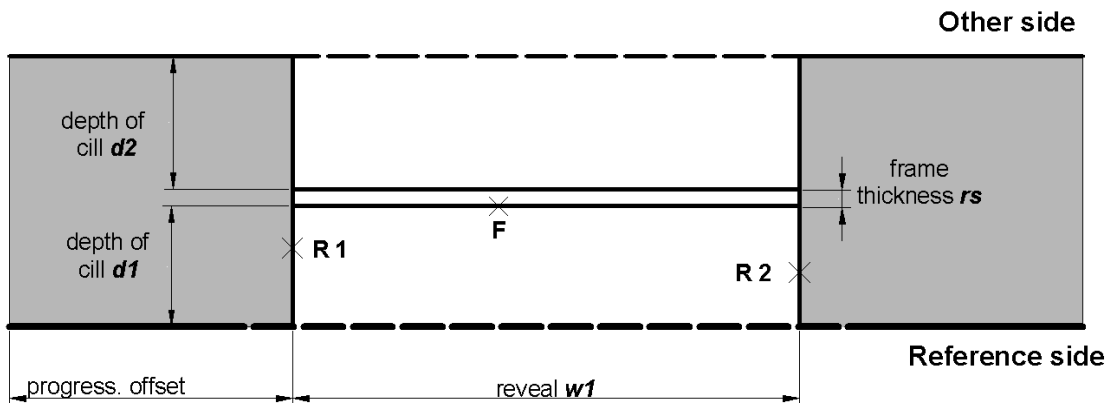
The reveal s are parallel to each other.

Window with offset reveal

The window is assumed to be symmetric. The following measurement options are available:
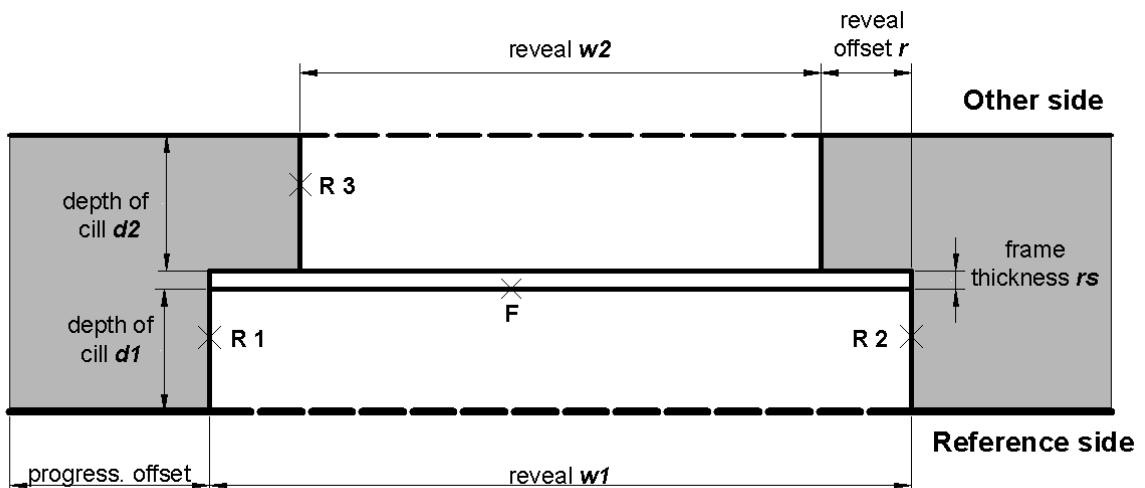
The window with offset reveal has two different reveal widths. The rabbet is rectangular. The measurement of the reveal on the reference side is similar to the simple window.

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 aim at both sides of the reveal with the total station, select this option. After finishing the dialogue, following queries appear in the command line:

  Select reveal point (R1) or [UNdo]: *(Measure point R1 on one reveal side)*
  Select second reveal point (R2) or [UNdo]: *(Measure point R2 on second reveal side)*

- **Point on one side:** If you can measure only one side of the window reveal with the total station, 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]:
  (Measure point R 1 or R2 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 use the total station at all. You can either enter the window width w*1* 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:** Measure 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 measure 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 measure the front of the reveal directly, choose this option. If you have selected a window type 'with frame' you will be prompted to measure 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]:

- **Depht of cill d1 (reference side) or d2 (second side) :** If you cannot measure 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.

Frame thickness (t) = 0

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 measurements'. This option is intended for measuring 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 ➔ **Annotation - Window dimension annotation** is called up after this command.

Measure – Measure Staircase

	Command:	TCADMEASSTAIR
	Ribbon tab	As-Built TS Design

The **TCADMEASSTAIR** command enables you to measure a staircase and to generate an according floor plan illustration.

Initial note: This command can be used with tacheometric point observations as well as with point cloud data and manual measurements.
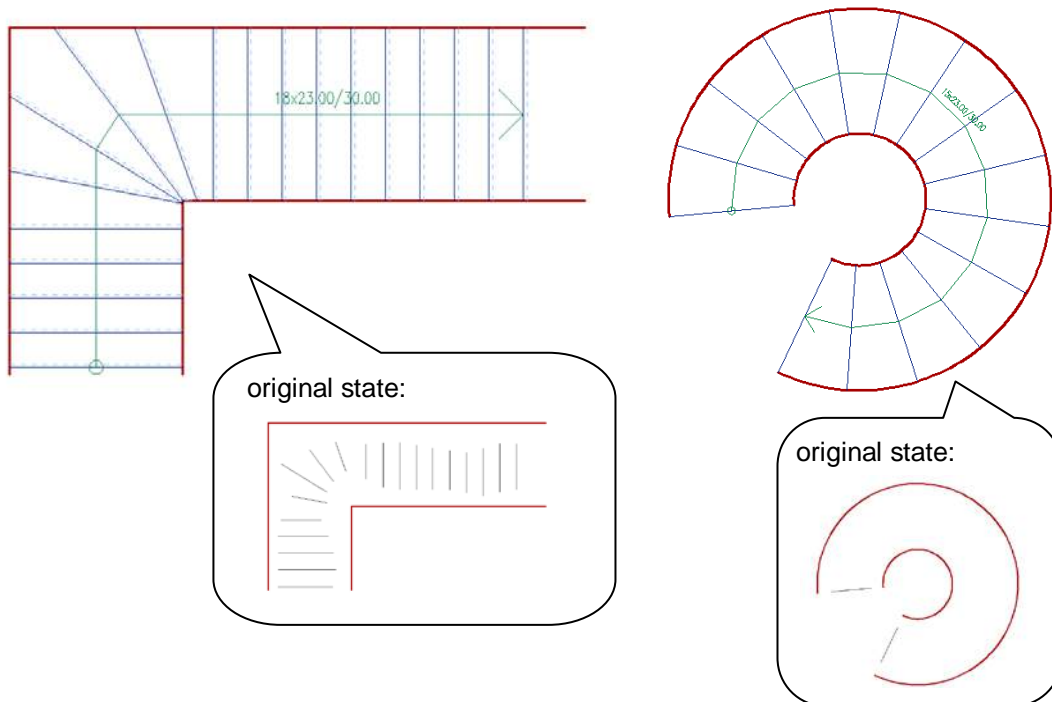
Requirements

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 the software 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:
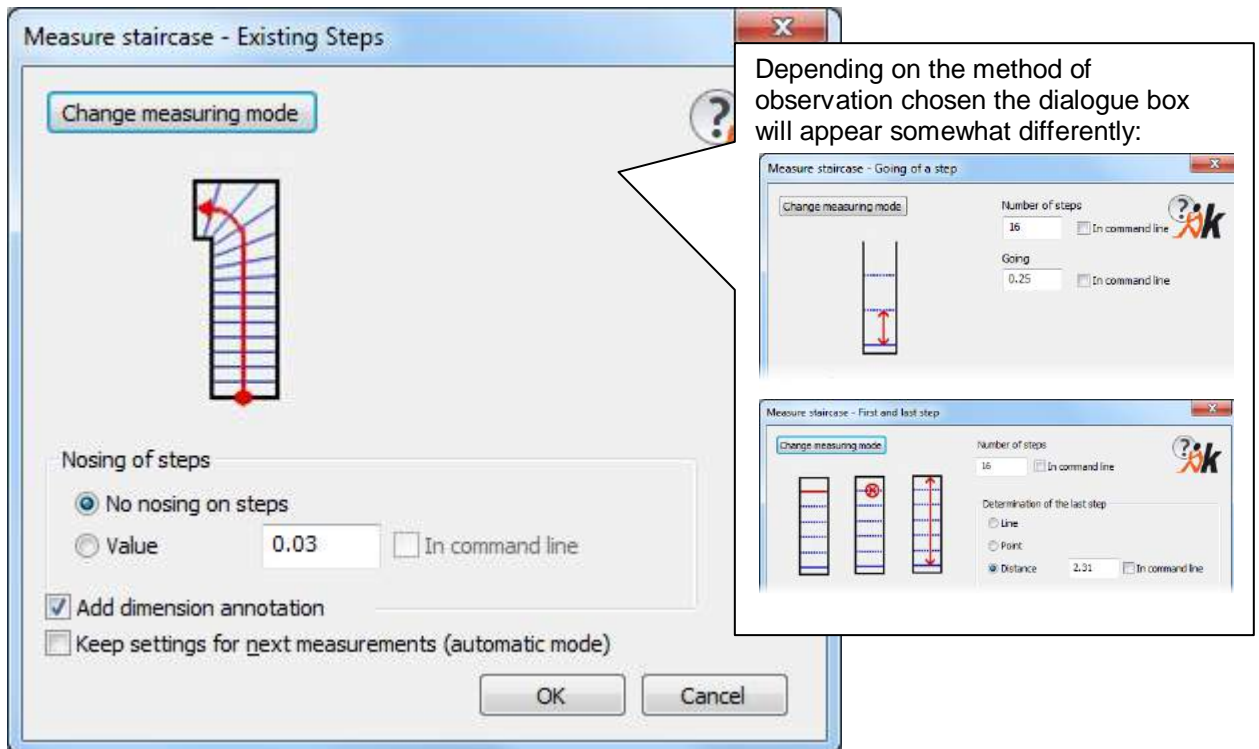
The button [Change measuring mode...] opens a selection dialogue box:
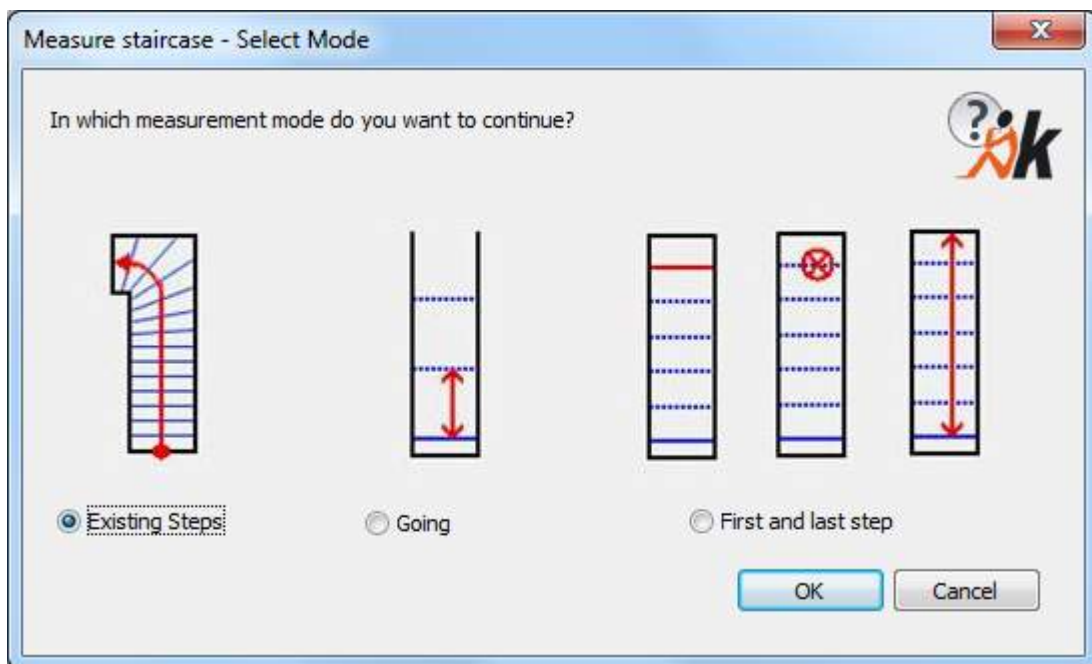
You can choose one of three methods of observation:

- 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.

This area is empty.

It is not at all necessary to enter any further information for the risers and goings.

They result from the existing line work.

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

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:

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".
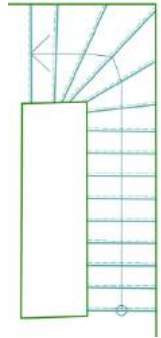
Going:

Here, a line for the first step will be requested, as well as a value for the going (depth of a step).

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:
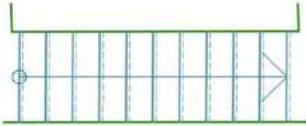
First and last step:

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.
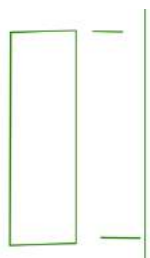
Line and second line

Line and point
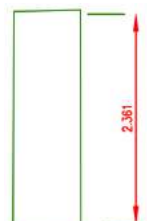
Line and length

Measure staircase - First and last step

Change measuring mode

Number of steps

16   ☐ In comm

Determination of the last s

◉ Line

○ Point

○ Distance   2.31

Nosing of steps

○ No nosing of steps

◉ Value   0.03   ☐ In command line

☑ Add dimension annotation

☐ Keep settings for next measurements (automatic mode)

Based on the number of steps entered the going (step depth) for a step will be automatically calculated.

"**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:

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 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:
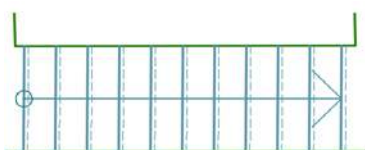
Stair dimensions

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 - Staircase – Arrow size" in order to adapt the size of the arrows.

Measure – Survey Grid

	Command:	TCADDRAWGRID
	Ribbon tab	As-Built TS Design

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.

First a tip: 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?

<u>Line:</u> Select a line with which the grid should be aligned parallel to.

<u>2 Points</u>: Click or observe two points, where the grid should be aligned parallel to the line joining the two points.

<u>UCS:</u> The grid should be aligned parallel to the XY axes of the UCS.
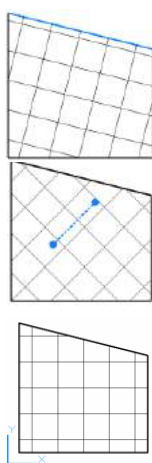
Node:

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.
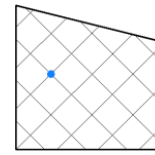
Line intersection: Select two known lines and two parallel distances from these lines. The intersection point of the parallel lines defines the node.
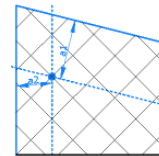
Arc intersection (Bilateration): The node is defined by selecting two known points and giving two radial dimensions The intersection point of the two arcs is the node.
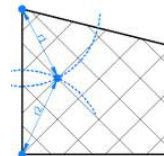
Grid spacing:

The spacing of the grid lines in two directions. If "In command line" has been selected, then the value will 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.
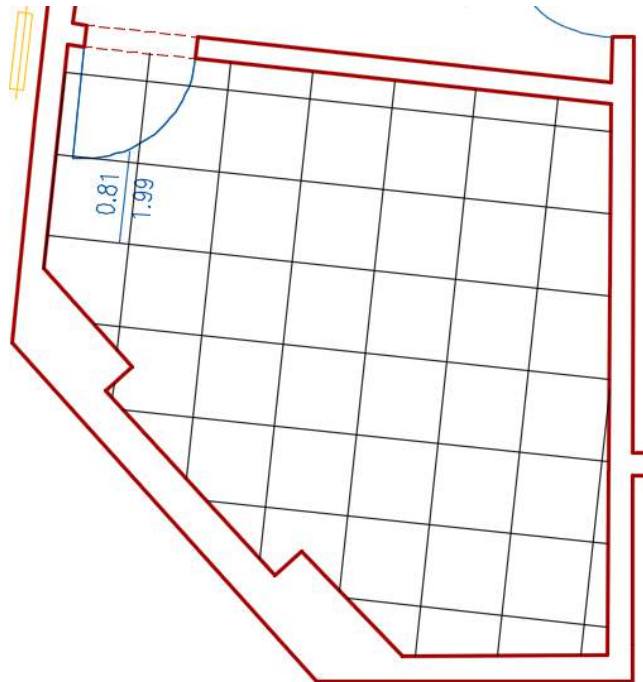
Measure – Show tool palette (symbols)

	Command:	_KUBT_TC_TPTOPOSHOW
	Ribbon tab	As-Built TS Control

This command displays the As-Built tool palettes. There is one with topographical symbols (As-Built Topo) and one with symbols for in-house connections, e. g. for sockets and water connections (As-Built Connections).

The As-Built Topo tool palette contains a number of block symbols for the design of your topography plan.

The symbols may easily be selected by clicking on them. Within the command line you will then be prompted for the insertion point. You can define it by measurement or by clicking within the drawing. The symbol will then be inserted at the given position.

The predefined symbols are sorted into the following categories

- Vegetation
- Water/sewage/electrics
- Signs and posts
- Coordinate dimensioning
- Line signatures

Important hint: For the symbols to be displayed in the correct size, you will have to state the units for your drawing. You can do so with the AutoCAD command "UNITS".
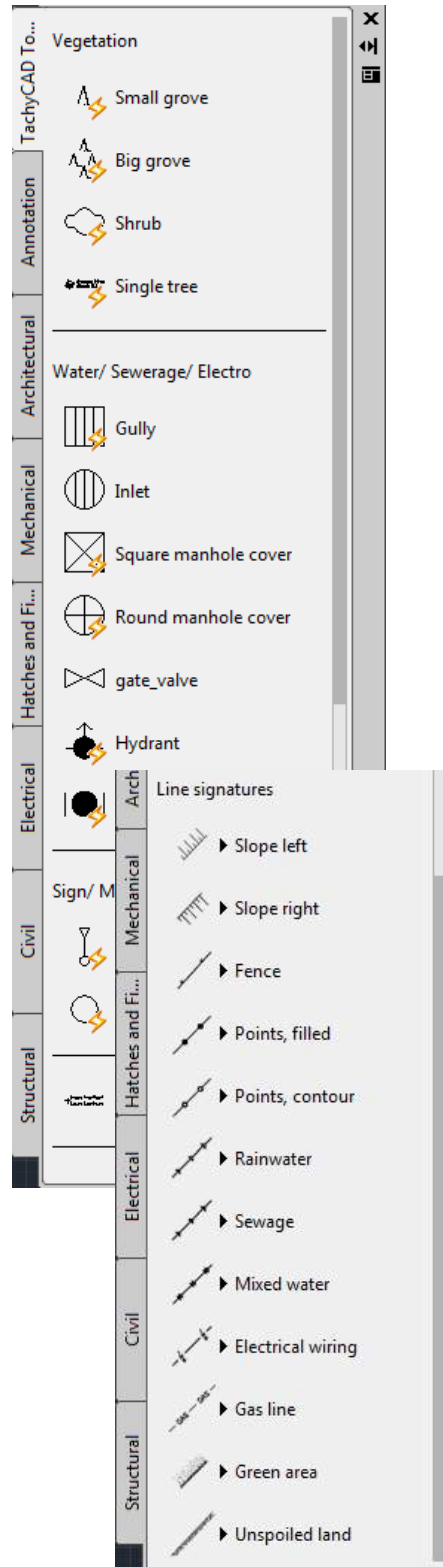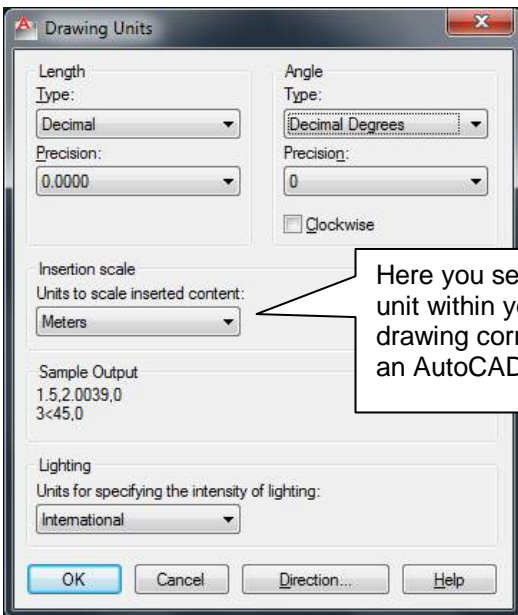
The As-Built Connections tool palette contains a number of block symbols for the design of your plan for the interior work.

You may simply select the symbols by clicking on them. You will then be prompted for the insertion point within the command line. You may determine that by measurement or clicking in the drawing. The symbol will then be inserted into the drawing at exactly that position.

The predefined symbols have been structured within the following categories:

- Electro
- Water/sewage
- Gas
- Ventilation
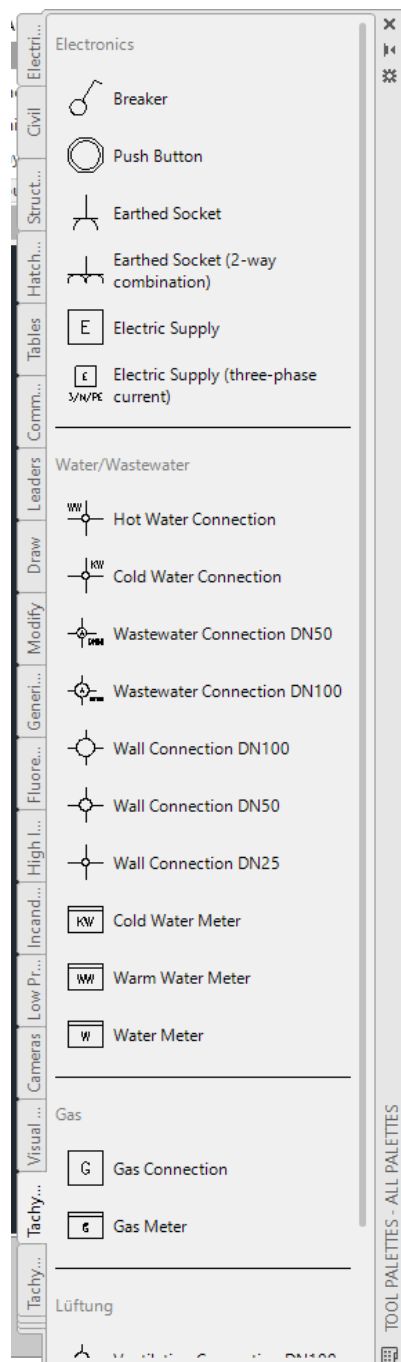
Adapting or extending the tool bar

The tool bars may individually be adapted or extended fairly easy.

<u>Adapting of a tool</u>

You may execute different settings for every tool via the context menu of the tool (right mouse button):

Here you may access different characteristics of the tool, like

- layer
- color
- scaling factor
- …

Especially for the line signatures you may set which line types can be generated via the characteristic "Flyout Option".

You can state your own icon picture for the tool.

Blocks

The samples for all block symbols offered here are saved within the drawing "Symbols_Topo.dwg" or "Symbols_Connections.dwg". You will find this file within the installation folder of As-Built in the sub folder **TOOL PALETTE**.

- Open this file, add your own blocks or change the existing ones.

- Save the file.

- Now pull the new block symbols into the tool bar by "Drag&Drop" with the mouse. If this should not work (There are difficulties with AutoCAD 2012) simply use the **AutoCAD design center**. (Command ADCENTER, tab "View" – Group "Palettes")

You call up the design center here.
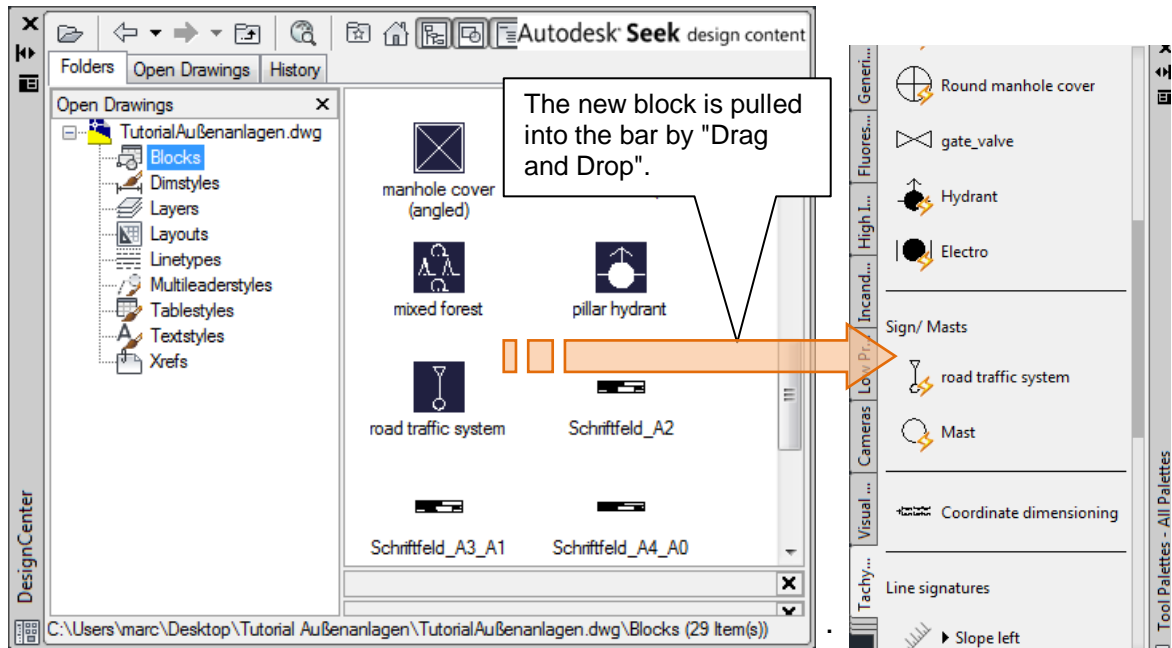
Line types

For the line types it is similar to the above. Add your own line types into the "Symbols_Topo.dwg" and pull them into the bar using the same method.

Measure – Measure profile section

	Command:	TCSECTIONPIN
	Ribbon tab	As-Built TS Design

The TCSECTIONPIN command is for the calibration of a profile section line between 2 points, as is typical for archaeological excavation documentation. Optionally the profile section may get a numeration labeling.
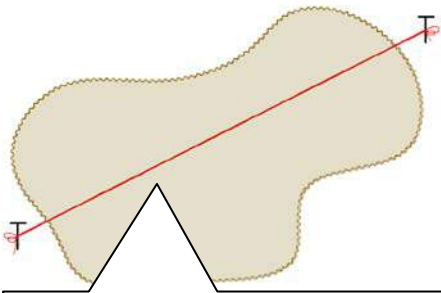
A feature has been mapped. With the help of two pins a string will be stretched across the feature outline, where the section is to be excavated.
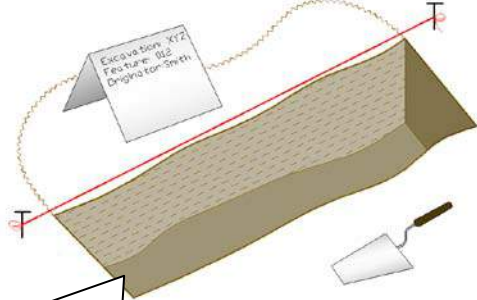
Here you can see the excavated section.

This is what the result looks like within the excavation plan, after the command has generated it. The short lines stubs show the direction of the excavation.
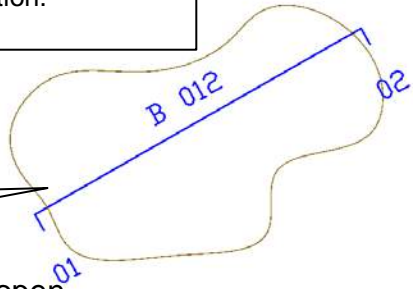
After calling up the command the settings dialog will open.

The profile section can be generated with or without direction. The symbols may be configured according to your own requirements.

Optional labeling of the complete profile section, it will be displayed centered above the outline.

Optional labeling of the profile section pins. It will be displayed underneath the profile section pin symbols. "Include profile name" means that the name of the profile will be written in front of the name of the pin (divided by a slash).

With his option several profiles may be generated one after the other.

The numeration labeling will, depending on the selected name, counted up automatically.

For example: When entering the name P_01 the next profiles will be named P_02, P_03 … etc. When entering the name "Feat.", the next profiles will be named "Feat.1", "Feat.2", "Feat.3"… etc.

After confirming the dialog with OK, the profile pins will prompted within the command line,

Command: **TCSECTIONPIN**

Enter first point:

Enter second point:

The order in which the points are entered determines the profile direction, if the option "With direction" is being used. When the points will be entered from left to right, the pins will point "downwards", when they will be entered from right to left they will point "upwards". The labeling will automatically be aligned to good readability.

The command may generate the following display by default:
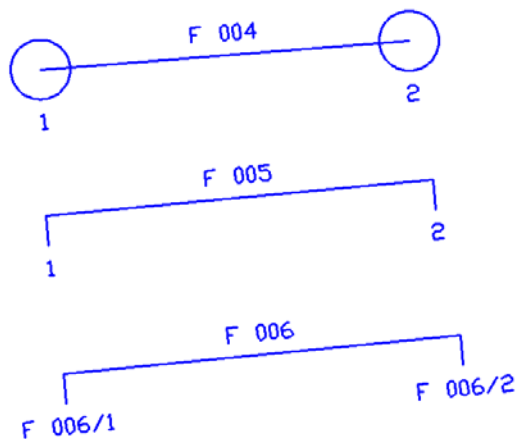
Design – Fix lines to corner

	Command:	CORNERFIX
	Ribbon tab	As-Built TS Design

The command 'cornerfix' is used for intersecting lines, polylines and arcs. It also works for skewed lines, polylines and arcs. 'Cornerfix' 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: **cornerfix**

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.

Design – Sharpen corner

	Command:	CORNERSHARPEN
	Ribbon tab	As-Built TS Design

The command "cornersharpen" combines the commands "intersect" and "delete". This command is useful when you have measured an outline plan, but the corner could not be measured exactly. An example:
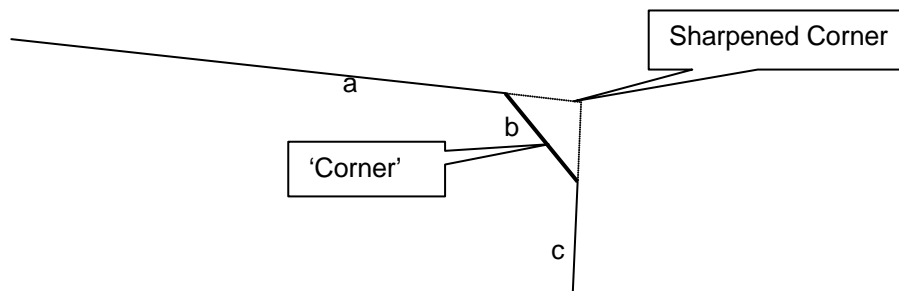
After you have called up the command, you will be asked to select the line to be sharpened. In this example it is line **b** which is marked off by two lines (**a,c**). "Sharpen" extends **a** and **c** until their projections intersect in the XY plane of the UCS, and then deletes **b**.

An alternative to the command "cornersharpen" offers the more complex command →**Measure – Measure outline plan**. This command can also clip.

Design – Align cursor

	Command:	ALIGNCURSOR
	Ribbon tab	As-Built TS Design

The command "aligncursor" 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 "aligncursor", following prompt appears:

Command: **aligncursor**

Select object or <Return> for previous cursor state:

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

Cursor aligned at a line.
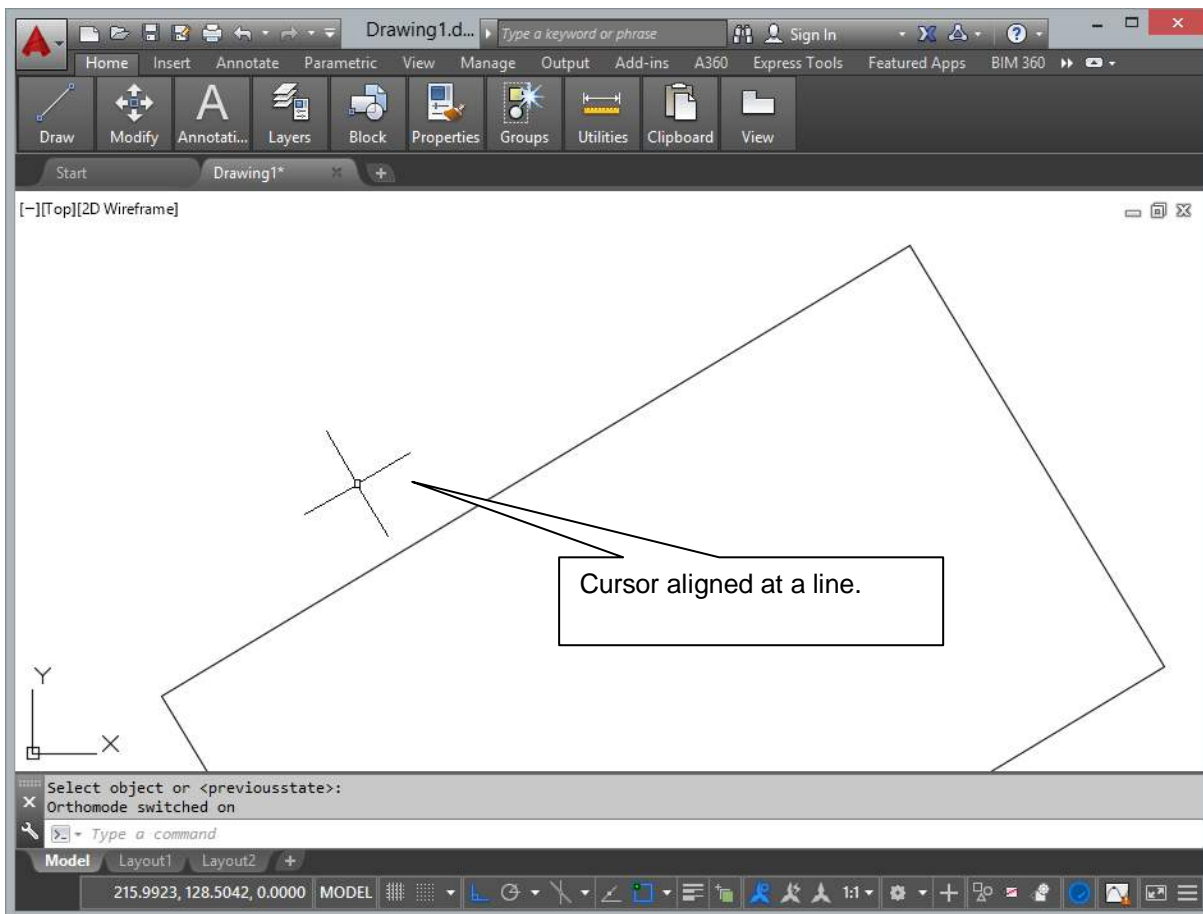
To reset the cursor to the normal orthogonal position, just answer with Return.

Design – Multiple trim and extend

	Command:	MULTITRIM
	Ribbon tab	As-Built TS Design

The command "multitrim" combines the AutoCAD commands "extend" and "trim". After the selection of the boundary / cutting edges...

> Command: **multitrim**
>
> Select boundary/cutting edges:
>
> Select objects: *(Select boundaries / cutting edges; confirm with RETURN)*

... you need to select the objects that are to be trimmed or extended at the boundary / cutting edge. You also have to select the side on which the trimming is to be carried out.

> Select objects to trim/extend: *(Select object; confirm with RETURN)*
>
> Select point for direction: *(Pick point on side)*

This command is of advantage when many lines are to be trimmed and / or extended at the same time. A typical example in building surveying are rafters in a ground plan:

The wall lines are determined as boundary / cutting edges.

The measured rafters are to be extended onto the wall.
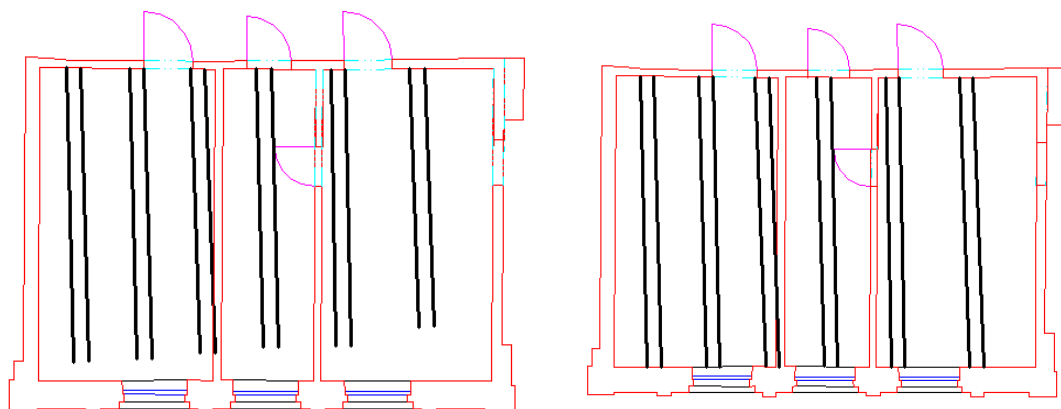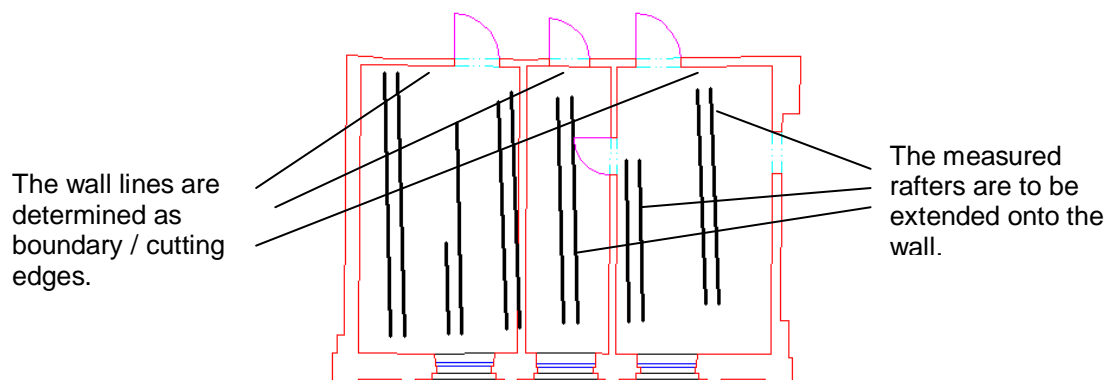
1st step with MULTITRIM   2nd step with MULTITRIM

Boundaries / cutting edges can also be construction planes.

Design – Horizontal extend

	Command:	'EXTEND+ Option [Horizontal]
	Ribbon tab	As-Built TS Design

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: **tcadextend** *(Calls up the command 'Extend')*
>
> Select the line: *(select the line to be extended)*
>
> Line horizontal extend by [By/Total/Horizontal/3D/to Point]: **0.20** *(Enter extension measurements)*
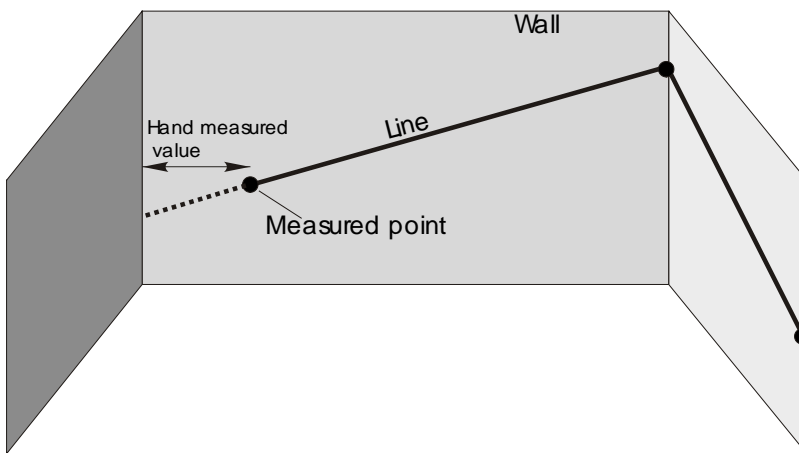
The line is extended into the direction that lies closer to the selected point.

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 ➔**Design – 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]: **'tcadextend**
>
> *(transparent call-up of the command 'Extend')*
>
> 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 'Extend' 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.

Design – 3D extend

	Command:	'EXTEND+ Option [3D]
	Ribbon tab	As-Built TS Design

This command is used for extending a line in a **three-dimensional** way.

Command: **tcadextend**   *(Call up of the command 'Extend)'*

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 ➔**Design – 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]: **'tcadextend**

(Call up the command 'Extend' 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 'Extend' 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.

Design – Perpendicular extend

	Command:	PERPEXTEND
	Ribbon tab	As-Built TS Design

The command 'perpextend' is used for the perpendicular, horizontal (in the current UCS) extension of a line from the last measured line end point perpendicular to the last line. Just measure the missing measurement (e.g. the depth of a recess) with a tape measure. The direction of the line (e.g. left or right) and its length are to be entered or to be picked.

The call-up of the command can be processed within the command 'Line' as transparent command.

Specify first point:	*(e.g. measure first corner)*
Specify next point or [Undo]:	*(e.g. measure to recess)*
Specify next point or [Undo]: **'perpextend**	
	(transparent call-up of the command 'Perpextend')
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 measuring)*

In case you do not call up the command 'perpextend' within the command 'line' 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: **perpextend**	*(Call-up of the command 'Perpextend')*
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.

Design – Measure height difference

	Command:	'HTDIFF
	Ribbon tab	As-Built TS Design

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: **'htdiff**
>
> 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).**

Design – Measure Z difference

	Command:	'ZDIFF
	Ribbon tab	As-Built TS Design

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: **'zdiff**

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).**

Design – Measure horizontal distance

	Command:	'HZDIST
	Ribbon tab	As-Built TS Design

The command 'hzdist' 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: **'hzdist**
>
> 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.

Design - Vertical view (UCS) through 2 points

	Command:	VUCS
	Ribbon tab	As-Built TS Design

With the command "vucs" 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 XZ coordinate origin of the new UCS is set to the first point. The Y value of this point matches the Z Value of the previous coordinate system.
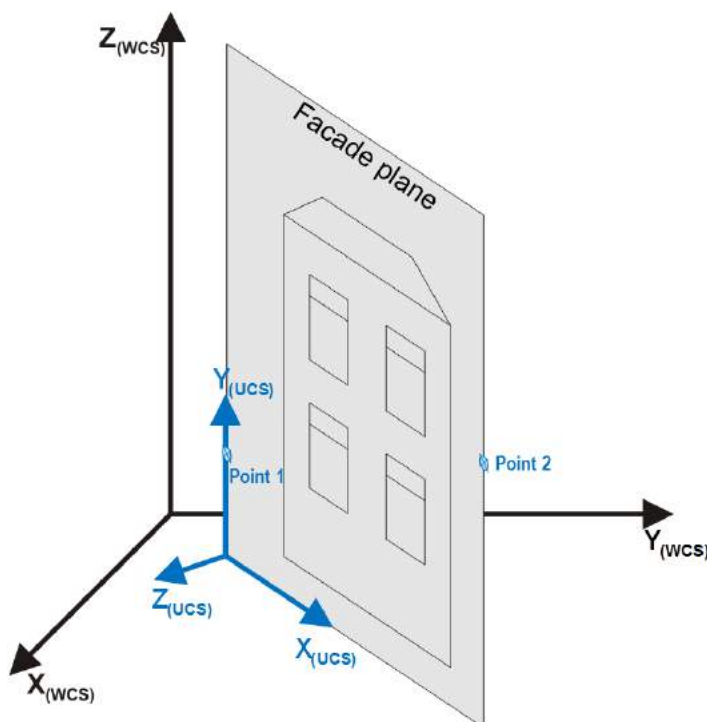
Command: **vucs**

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, As-Built for AutoCAD 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).

Design - New UCS through 3 or more points

	Command:	MUCS
	Ribbon tab	As-Built TS Design

With this command you can define a new user coordinate system (UCS) at any alignment through any number of points.

By selecting at least three points, you determine the XY plane of the new UCS. Then you select a point which describes the origin of the new UCS.

If you enter more than three points, an average plane through these points will be calculated. The first two points determine the direction of the X axis of the UCS.

Command procedure:

Command: **mucs**

Defines new UCS through 3 or more points.

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

1. point for XY plane of new UCS: *(select at least three points)*

After the point selection has been completed with the option 'Finish', you are asked for another point for the origin of the UCS.

Select point for origin of UCS: *(enter point)*

UCS aligned for [Elevation/Plan] <Elevation>:

UCS name: *(Enter name for UCS)*

The options 'Elevation' and 'Plan' determine the alignment of the X axis of the new UCS. If 'Plan' is chosen, it will be aligned parallel to the XY plane of the WCS; whereas if 'Elevation' is chosen, it will be aligned parallel to the XZ plane of the WCS. The option 'Elevation' is used for almost vertical facades, that you want to look at from the front. The option 'Plan' is used for approximately horizontal planes, that you want to look at from top.

At the end of the command, the programme automatically changes to the plan view of the new UCS.

Note: If you want to put the UCS on the screen later, use the AutoCAD command '**Tools → named UCS**' The dialogue 'UCS' appears with the register 'Named UCS'. Select the respective entry from the list and click 'Current'.

Design – Fit plane

	Command:	KPFIT
	Ribbon tab	As-Built TS Design
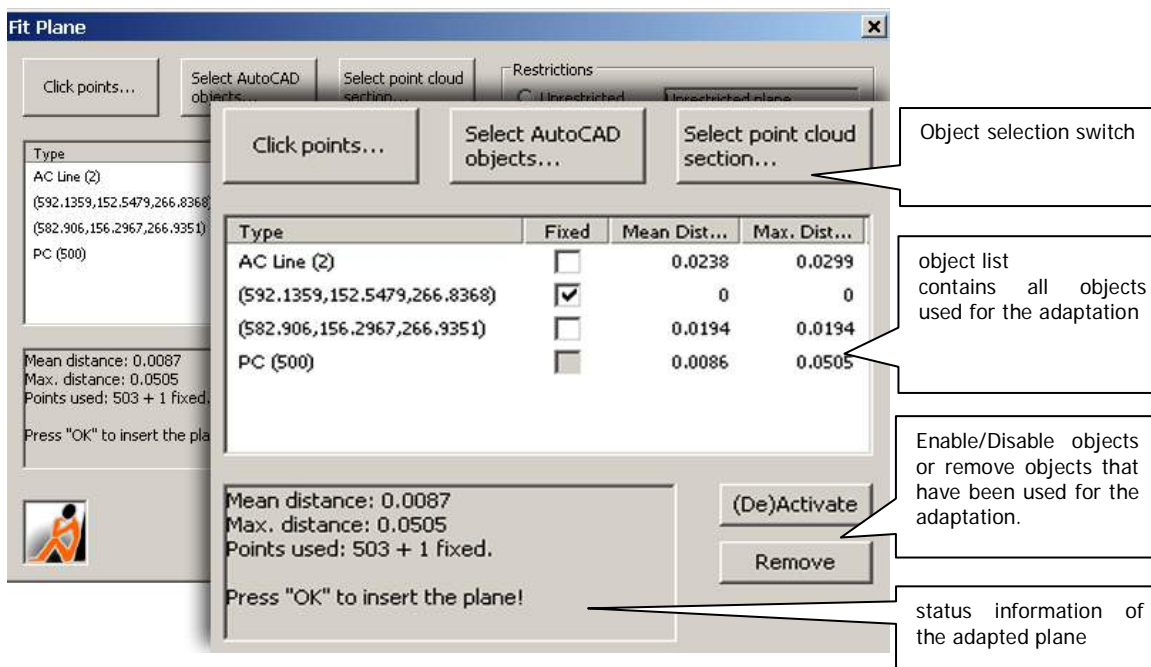
An adjusting plane will be calculated over a set of points. The plane minimizes the total of quadratic distances (orthogonal) of all points to that plane.

For a better orientation when working with a lot of planes it is recommended to give each a unique plane name.

Under As-Built for AutoCAD Settings… the object layer, on which the plane will be created, can be defined. The default value for LayerKubitPlane is KUBITPLANE. The current layer will be used, if the field is empty.

By clicking **OK** you will close the window and the plane will be inserted into the AutoCAD drawing.

The plane is indefinite by default and affixed on its supporting plane.

Object selection for the plane

You may use two different kinds of objects for the plane adaptation:

- Clicked points
- AutoCAD points, lines and block references (insertion point)

The object selection is controlled by two corresponding object selection switches.

The objects, used for the adaptation are shown within the object list. The error information (mean and maximum distance) will be displayed for each object and are given 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 adapted. To do so activate the object in the column "Fixed" in the object list. The fixed objects will become bases of the plane (error value=0). All

other objects will be used for the adaptation process. The number of points that can be fixed depends on, among other factors, whether a restriction has been defined and if so which.
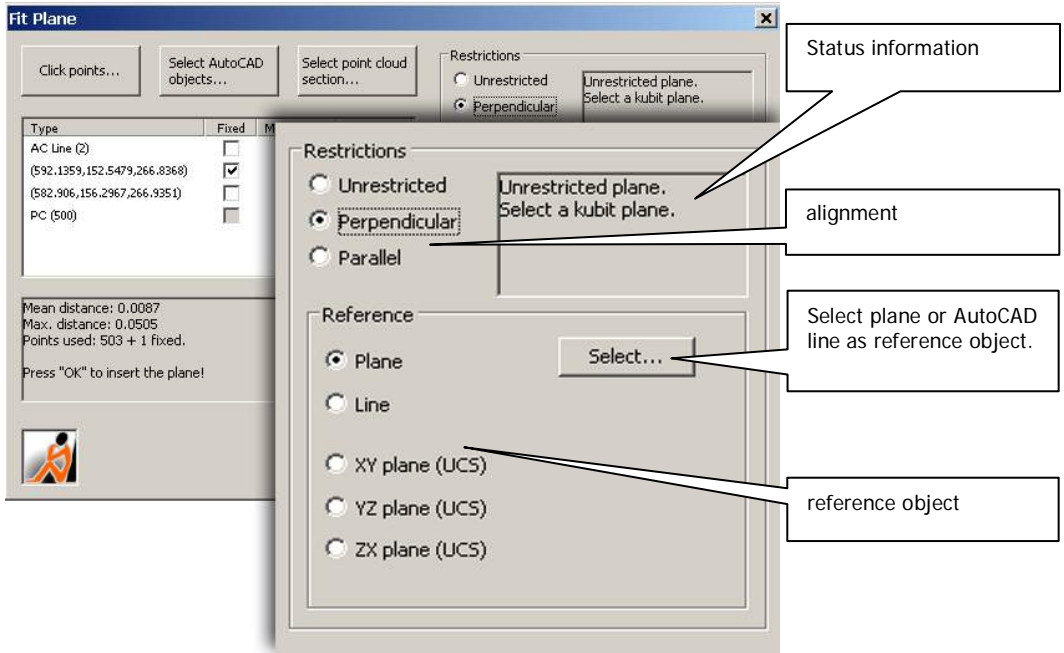
Restrictions

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

Restriction	equivalent restriction	necessary number of points
		max number of fixed points
free plane		3
perpendicular to a plane	Parallel to a line whose direction vector is the plane normal.	2
parallel to a plane	Vertical to a line whose direction vector is the plane normal.	1

Design – Draw plane

	Command:	KPCREATE
	Ribbon tab	As-Built TS Design
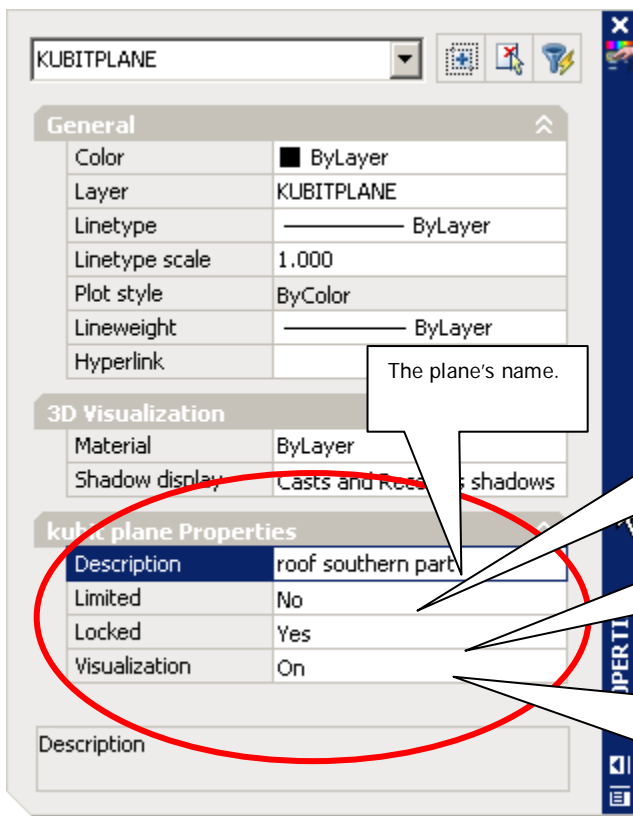
With this command you may draw a plane by clicking points. The first three selected points define the supporting plane. All further points refine the boundary.

Select first point:	*(Select the first support point)*
Select second point:	*(Select the second support point)*
Select point:	*(Select the third support point*
	and more if required for the boundary.)

Complete the point selection with [**ESC**]or by clicking the right mouse button.

The resulting plane is indefinite and fixed on its supporting plane.

The planes – also called construction planes or Kubit planes – are 3D surfaces. They have an infinite mathematic supporting plane and a boundary which visualizes this plane. You may display and change the characteristics of a plane. You may do this either within the AutoCAD properties window or by manipulation of the nodes of the plane.

The plane's name.

The plane is either indefinite ("No") and therefore in its circumference consistent with its supporting plane or it is limited ("Yes") and thus its circumference is represented by its boundary.

The supporting plane is either locked against editing ("Yes") or it may be edited by manipulation of its nodes ("No").

Here you controll the display of the symbol within the plane.
When exploding a plane (AutoCAD command "Origin") a 2D polyline results from the current boundary.

You will find a symbol within the plane boundary, which changes its appearance according to the current plane properties.

Plane is infinite (infinite extension) and fixed on its supporting plane (default).

Plane is infinite and *not* fixed on its supporting plane.
Caution: This plane may be changed freely!

Plane is delimited by its boundary and fixed on its supporting plane (locked).
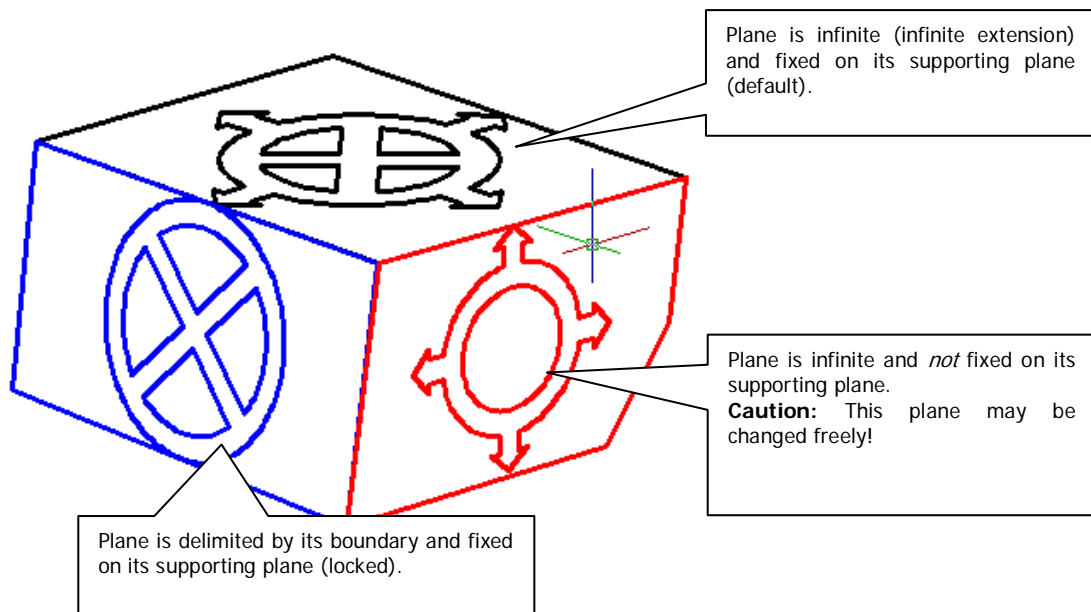
You may name the plane by selecting it, running the AutoCAD command "properties" and in the section "kubit plane Properties" entering a description. A unique name makes sense in order to ease the work with many planes in one drawing.

Under As-Built for AutoCAD Settings… the object layer, on which the plane will be created, can be defined. The default value for LayerKubitPlane is KUBITPLANE. The current layer will be used, if the field is empty.

Design – UCS from plane

	Command:	KPUCS
	Ribbon tab	As-Built TS Design

You may define an AutoCAD user coordinate system with this command.

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 will be defined as follows:

- The Z axis points in the direction of the plane normal.

- The Y axis points in the direction of the Z axis of the WCS.

- The X axis points preferably to the right, referring to the current AutoCAD view port.

- The UCS origin is the point closest to the origin of the WCS on the plane and on the WCS XY plane.

For planes, which are parallel to the XY plane of the WCS, only the origin of the UCS will be moved to the supporting plane.

Design – Modify plane – Extend (2 planes)

	Command:	KPEXTEND
	Ribbon tab	As-Built TS Design

This command extends or trims 2 planes up to their common intersection edge. The extension or trimming is processed symmetrical, therefore it does not matter, which plane is selected first.

> Select first plane to trim or extend:

> Select second plane:

Remark:

You may also use planes in order to extend or trim line-like AutoCAD graphics (lines, arches, polylines). Please use the AutoCAD commands "extend" and "trim" in order to do so.

Design – Modify plane – New rim

	Command:	KPNEWRIM
	Ribbon tab	As-Built TS Design

You can draw the boundary of a plane with this command. The supporting plane and all characteristics of the plane will be maintained.

Select a kubit plane: *(Select a plane for that a new boundary is to be drawn.)*

Select first point:

Select second point:

Select point: *(Draw a new boundary.)*

Design – Modify plane – Modify rim

	Command:	KPMODIFYRIM
	Ribbon tab	As-Built TS Design

You may modify the boundary of a plane with this command, "cut" a corner or extend the boundary on one side for example.

> Select a kubit plane:
>
> Click point on plane's rim: *(Click a point at the boundary.)*

The following error message will appear in case the clicked point is too far away from the boundary:

> Point is not on rim!
>
> Click point on plane's rim: *(Click closer to the rim.)*

Otherwise you may continue with the changing the boundary:

> Select point for plane's rim or [Finish]:
>
> Select point for plane's rim or [Finish/Undo]:   *(Draw the new part of the boundary.*
>
>   *Select finish in order to close the command)*

If the point clicked last is already close to the boundary, the command will be closed. Otherwise you will be prompted again to click a point on the rim.

> Select point on rim to finish!
>
> Click side:

Now you have to decide, which part of the plane you want to keep. To do so, move the crosshairs within or outside of the corresponding boundary. Depending on where the crosshairs are at that moment, one of the three possible boundarieswill be displayed. Confirm your selection by clicking the left mouse button.

crosshairs outside -> the outer boundary will be selected

crosshairs in lower part

crosshairs in the upper part
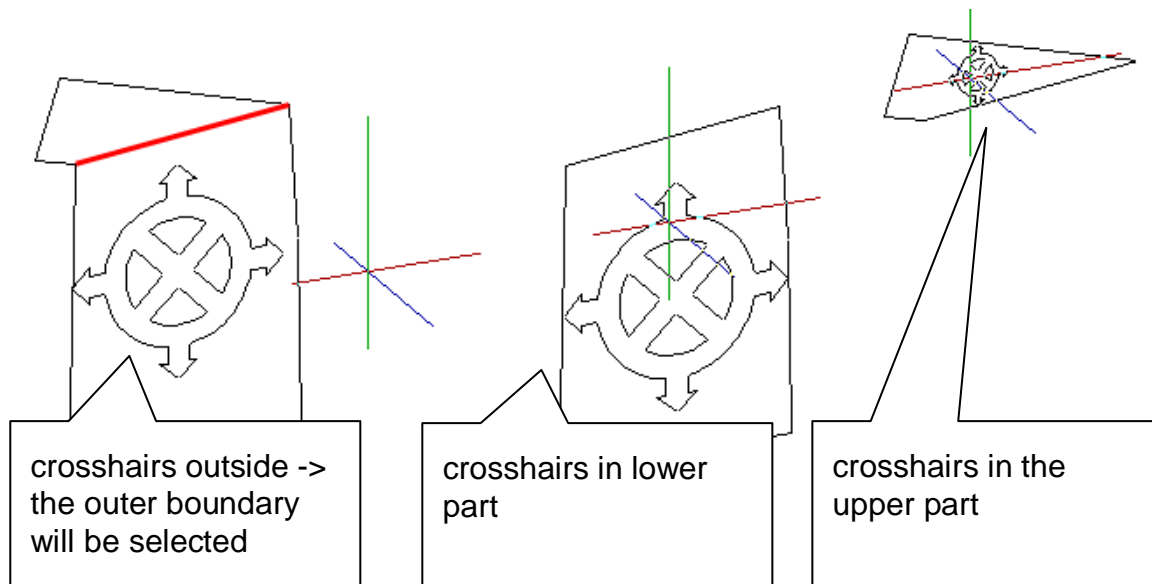
Design – Intersect plane – Intersection line (2 planes)

	Command:	KPINTERSECT
	Ribbon tab	As-Built TS Design

This command generates the intersection line of two planes.

Select first plane for creating intersection line:

Select second plane:

The intersection line will be inserted on the layer which has been entered under → **Settings** under the "Layer – plane" entry. The current layer is entered by default.

Design – Intersect plane – Intersection point (3+ planes)

	Command:	KPINTERSECTPOINT
	Ribbon tab	As-Built TS Design

This command will create the intersection point of 3 or more planes. The planes should not be pairwise parallel for this.

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

In case more than 3 planes were selected, a mean intersection point will be generated. That will result from the balance point of the intersection points of three planes at a time. Error values will be displayed in this case:

> mean distance between the planes and the intersection point: 0.03

> max distance between the planes and the intersection point: 1.12

The intersection line will be inserted on the layer which has been entered under →
Settings under the "Layer – plane" entry. The current layer is entered by default.

Design – Intersect plane – Intersection line (3 planes)

	Command:	KPINTERSECTTRIAD
	Ribbon tab	As-Built TS Design

This command generates an edge tripod in the intersection point of three layers. The layers may not be pairwise parallel.

This command is for the construction support of edge models or surface models (for further information please refer to chapter → **Edge and surface modeling**)

Command cycle:

Select first plane:

Select second plane [Undo]:

Select third plane [Undo]:
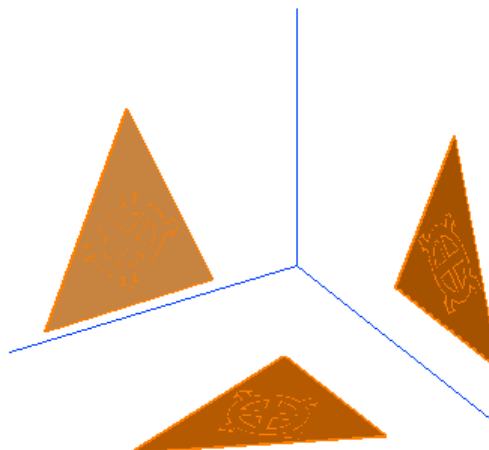
Three lines for the section lines of the respective planes will be created. The length of those lines will be determined by the expansion of the visible boundary of the kubit plane. It may also occur that the edges of the tripod will not end in the intersection point, as you may have expected → See figure below.

The 4. leg will be sustained. It will not be shortened.
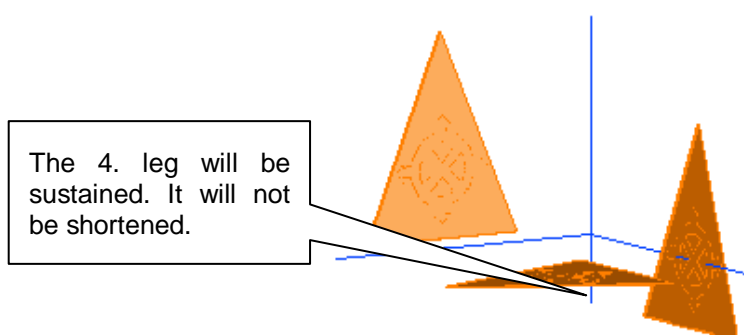
The tripod will be placed on the layer which has been entered under → **Settings…** under the "Layer – plane" entry.

Design – Intersect plane – Intersection with Lines, Planes, 3D-Solids, PNets

	Command:	INTERSECTPL
	Ribbon tab	As-Built TS Design

The command "intersectpl" allows you to intersect construction planes with other objects.

> Command: **intersectpl**
>
> Select section plane: *(Select the construction plane that is to be used as intersection plane)*
>
> Select object to be subtended: *(Select the object(s) to be intersected)*

Possible objects to be intersected are:

- Lines – an AutoCAD point will be created

- Other construction planes – a line will be created

- Solids – two solids will be created

- Poly face nets (PNet) – an intersection line (polyline) will be created

Design – Intersect plane – Perpendicular projection of points onto plane

	Command:	KPPERPRETURN
	Ribbon tab	As-Built TS Design

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*

Design – Convert old TachyCAD construction planes

	Command:	TCCONVERTPLANES
	Ribbon tab	As-Built TS Design

You may convert old TachyCAD construction planes (till version 5.0) into new construction planes (kubit planes) with this command.

Command: **tcconvertplanes**

Select TachyCAD planes to be converted:

Select objects: *(Click TachyCAD planes)*

Converted ...TachyCAD planes into construction planes.

Design – Cylinder unrolling

	Command:	TCADUNROLLING
	Ribbon tab	As-Built TS Design

This is the command for the unrolling of CAD graphics on cylindrical surfaces into the plane.

This function may be used for example in order to generate an unrolled view from a cylindrical building façade.

Initial situation:
The graphic is on the cylinder surface.

The cylinder will be "sliced" across from the reference point ...

… and unrolled into the plane.

Reference point

Result:
The graphic is now on the unrolled plane.
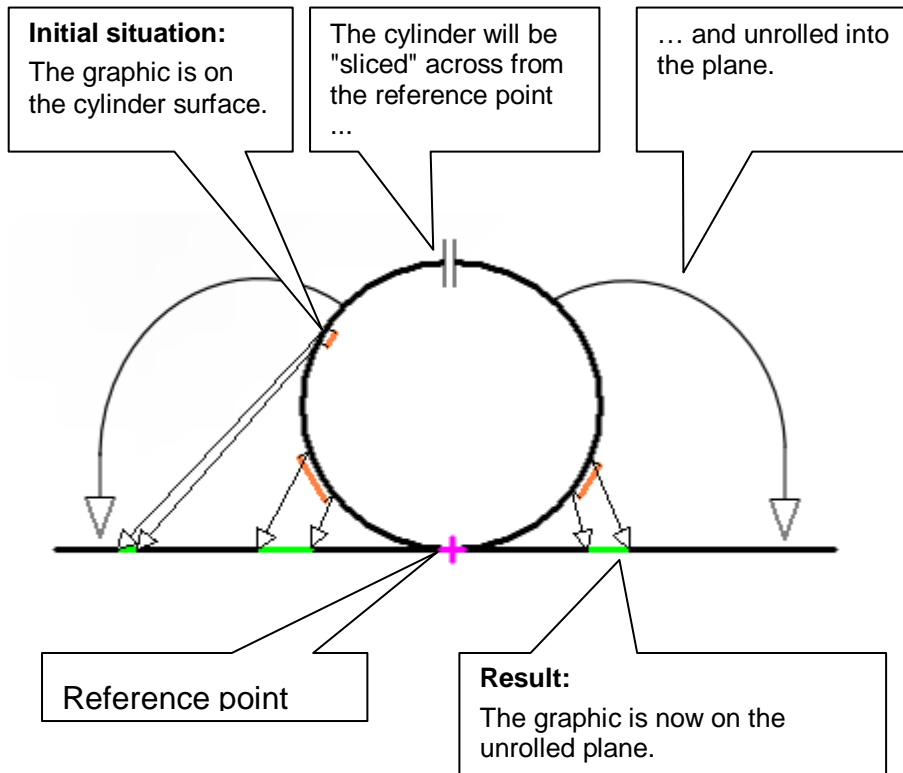
After you have started the command the following dialog will appear.

Fill in the details. Then you may start the unrolling with "Compute projection".

kubit cylinder

Select a kubit cylinder from your drawing. It will represent the cylindrical lateral surface to be unrolled. You will have to generate the kubit cylinder previously. In order to do so, use the →"**Generate adapted cylinder**"[13] command.

Side to be unrolled

Here you will state, if you have measured from the inside or the outside that means if you want an interior view or an exterior view of the cylinder.
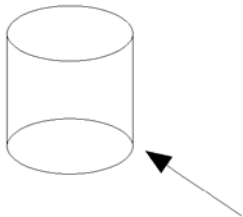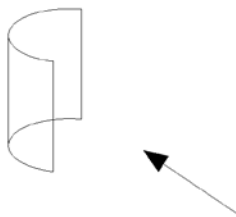
14. outer face   **15. Inner face**
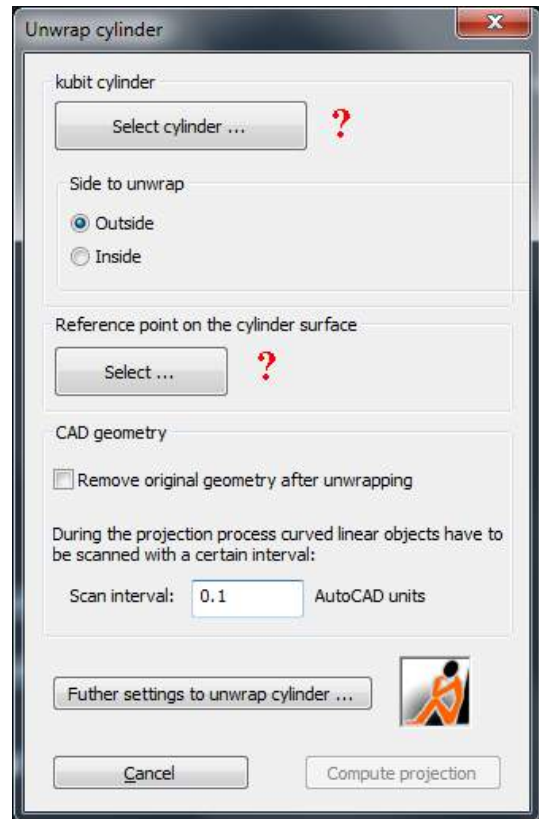
Reference point on the cylinder surface

The reference point states that point on the cylinder barrel on which the unrolled view will be fixed. Across from the reference point the cylinder barrel will be sliced and unrolled into the image plane.
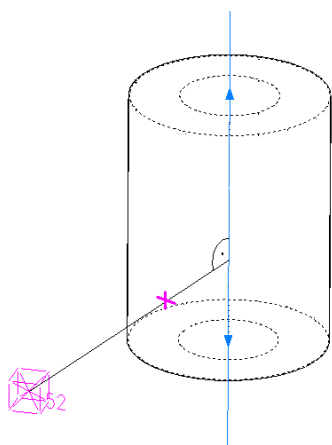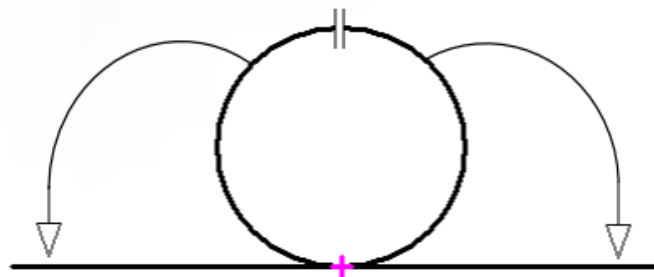
In order to state it, click a point close to the cylinder surface. That point will now be projected orthogonally onto the cylinder axis. The piercing point on the cylinder surface is the reference point.

---

[13] The vertical solution "plant surveying" provides even more possibilities to generate a cylinder.

CAD geometry

This is where you can set the scan width of the unrolling of geometry for curve-like objects (lines, polylines). The smaller you choose the value, the more detailed the unrolled curve elements will be, that means the more support points you will have.

Furthermore you can determine, whether the initial geometry is to be deleted after the unrolling.

Further settings ...

A click on this button will open the following dialog, where you will determine the position and the alignment as well as the width of the result of the unrolling.
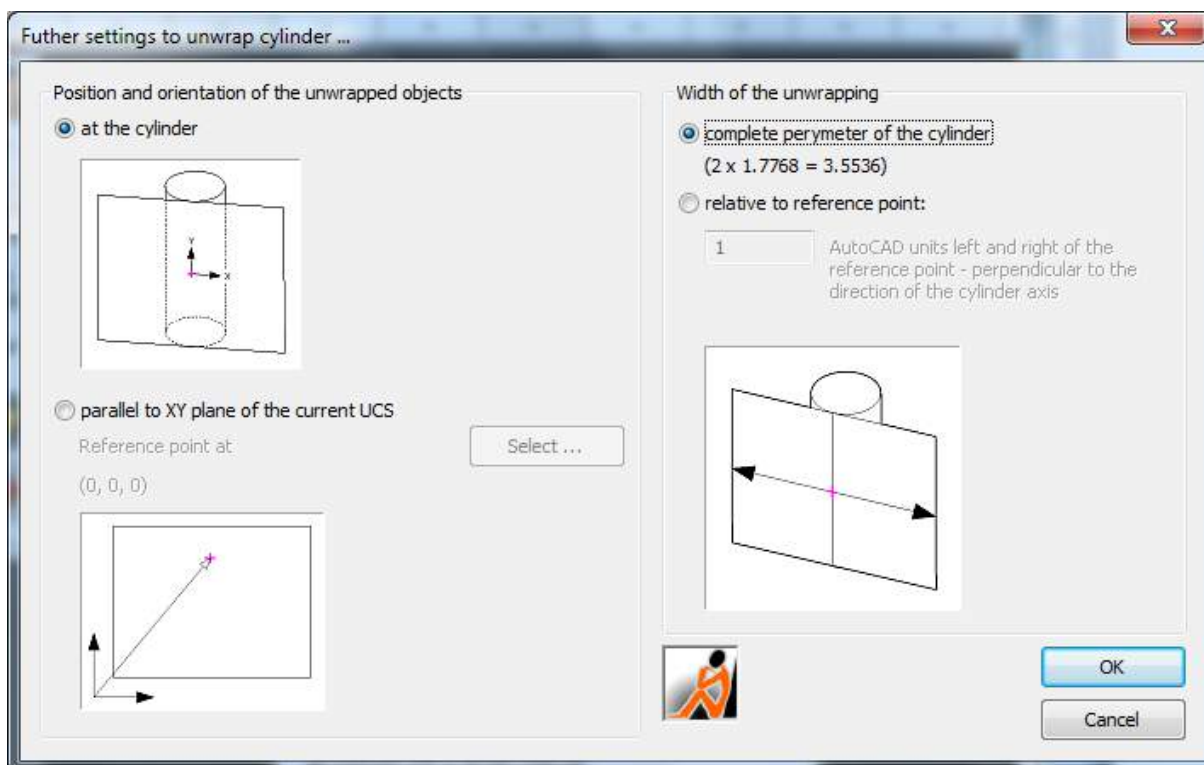
Position and orientation of the unwrapped objects

The result of the unrolling can be aligned and positioned *at the cylinder*. When doing so, a new UCS will be set. Its origin will be in the previously defined reference point. The Y-axis runs parallel to the cylinder axis through this point on the barrel, the X-axis perpendicular to it, as shown in the picture above. With this, the unrolling has been defined.

Alternatively the result of the unwrapping may be generated *parallel to the XY plane* of the current UCS within the drawing. With "Select …" you will define a point, at which the unrolled CAD geometry will be inserted into the current UCS. After that, the unwrapped graphic will usually not be "attached" to the cylinder anymore.

Width of the unwrapping

By default, the complete cylinder will always be unrolled. But since this is not necessary or wanted for many applications, it is possible, to determine the width of the unwrapping manually. The reference for this specification is the reference point on the cylinder barrel.

Unwrap Cylinder

After starting the unrolling (click on the button "Compute projection"), you will be prompted for the geometry to be unrolled. Select it from your drawing. The following object types are supported:

- AutoCAD points

- AutoCAD curve objects (lines, polylines, 3D polylines, splines, arches, circles…)

These will be scanned according to the scan width determined under "CAD geometry", projected onto the cylinder barrel and then be unrolled exactly into the image plane.
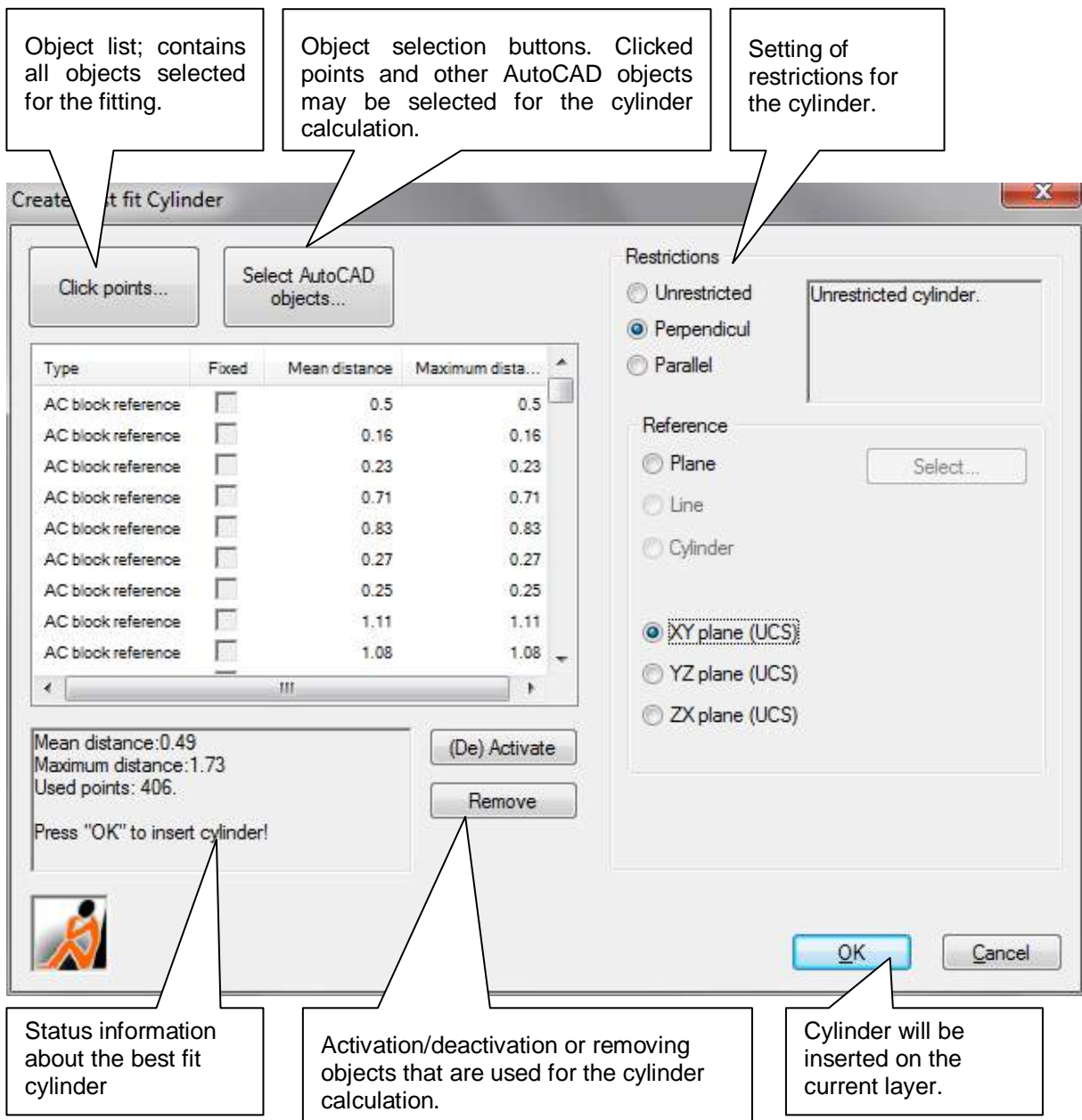
Hint:

The unrolled geometry will automatically be placed on the layer, which you have defined with the help of the As-Built for AutoCAD command "Settings..." under layer "LayerUnwrappedGeometry". If no layer has been stated there (default), the geometry will be placed on the current layer.

Design – Create best fit cylinder

	Command	KCFIT
	Ribbon tab	As-Built TS Design

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

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. At least 9 Points are necessary for the calculation of the cylinder. Selected line objects provide two points, the starting and the end point, the insertion point will be used from the blocks.

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 about the best fit cylinder

Activation/deactivation or removing objects that are used for the cylinder calculation.

Cylinder will be 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)

The object selection is controlled by the according 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 included in 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).
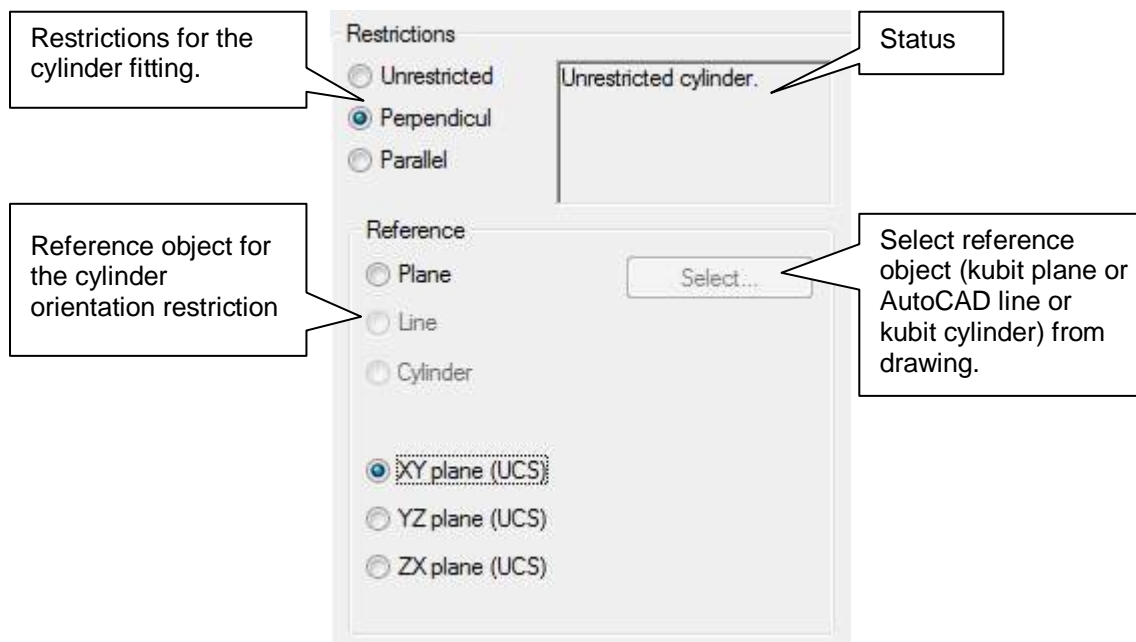
Convert Cylinder into solids: To create an AutoCAD solid from a kubit cylinder, just use the AutoCAD command „EXPLODE"
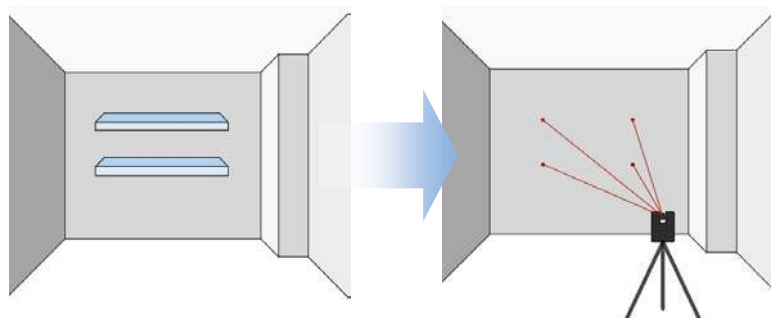
Projector

	Command:	TCPROJECT
	Ribbon tab	As-Built TS Design

This command is used to project points from the drawing, with the help of the visible laser point of the total station, onto a flat surface.

Components constructed in CAD (e.g. bore holes, section planes or similar) can be transposed locally to the site.

The process is optimised for a one-man operation. All of the points that are to be projected are first of all selected and then the sighting of the laser point follows automatically.
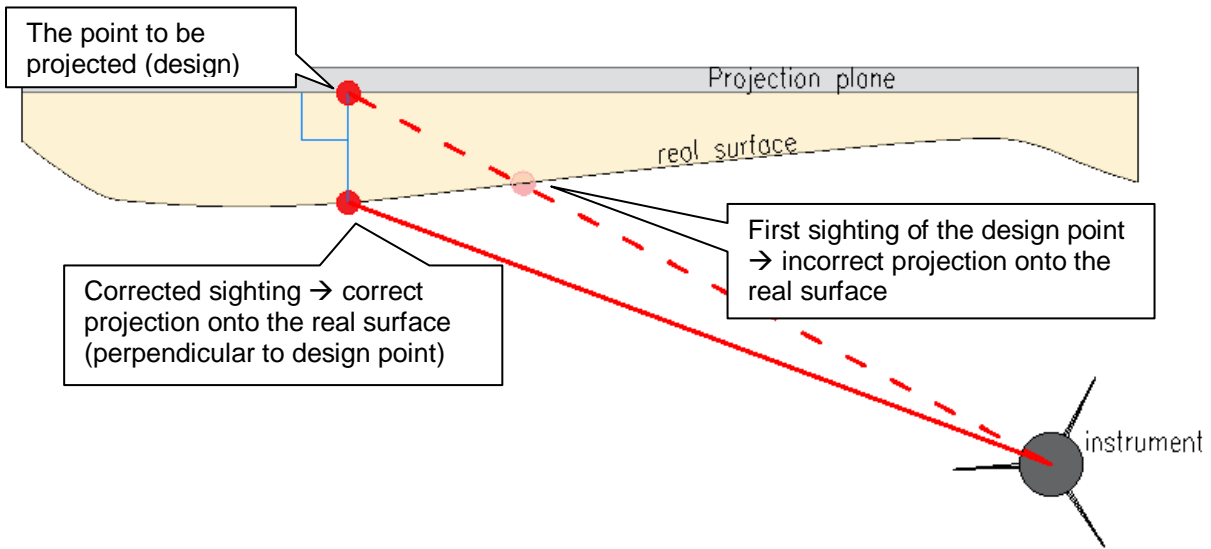
A requirement is a motorised total station with reflectorless functionality.

The following instrument types are supported by this command:

- Leica Viva TS 15/TS16, TPS 1200
- Trimble S-series, VX
- Leica 3D Disto
- Flexijet

The way this command is used:

The command works from an intelligent algorithm: The flat surface, on which the points that are to be projected lie, is first selected (=Projection surface). It is often the case, that the real surface area varies a little or a lot from the selected projection plane (e.g. when the wall is uneven). As-Built determines from multiple observations the real surface, and then sights a point onto the real surface that is a perpendicular projection of the design point on the projection plane. Thereby, even with some unevenness, it does not lead to incorrectly positioned sighted points.

The point to be projected (design)

Projection plane

real surface

First sighting of the design point → incorrect projection onto the real surface

Corrected sighting → correct projection onto the real surface (perpendicular to design point)

instrument

Detailed command sequence

After starting the command the following dialogue box opens, in which a series of entries have to be made:
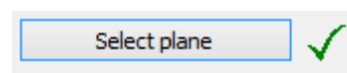
Projection plane:

Select the plane onto which the points should be projected. It can be, for example, a wall, or also a floor or a ceiling. The plane should be surveyed in, that means, it should mirror the real surface on site. There are several possibilities on how the projection plane can be defined.

- From a kubit plane: Previously create a kubit construction plane with one of the commands available in As-Built

- By entering or observing three points

- From the XY plane of the current UCS
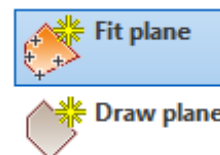
Click on "Select plane" to perform one of the three options. As soon as the plane has been selected a green tick is displayed next to the button, to indicate that this important task has been carried out.

An **offset** can be optionally selected, when the real plane has a parallel displacement. An example: The wall was surveyed as a kubit plane in the building shell. Now boreholes have to be projected onto the wall, that in the meantime has been plastered. Enter the thickness of the plaster as the offset. The value can either be entered as a value or determined by making an observation of a point.

Projection points

This is the list of points, that are to be sighted (projected).

The points are displayed in the list with a sequential number and various other pieces of information:

X | Y | Z: Coordinates in relation to the current UCS

Status: active: Point should be sighted

            inactive: Point has been deactivated and should not be sighted

            invalid: Point lays outside of the tolerance A(max) and will not be sighted

R: radial deviation of the point from the design point (is first available after a successful projection)

A: Distance of the point from the projection plane. If the distance is greater than A(max), the status changes to "invalid"

Add points: Select objects in the drawing, that should be used for projected points. These can be end or intermediate points of lines and polylines or also point objects or blocks (insertion point).

Remove points: Deletes the highlighted point from the list.

(de)-activate point(s): Sets the status of the highlighted point to "inactive". The points remain in the list but they will not be sighted. With the same function points can also be made active again.

Reverse order: the order of the points is reversed.

Up/Down: The highlighted points can, in their sequence, be moved either upwards or downwards.

Sight points: All of the points in the list with a status of "active" will be sequentially sighted. Waiting time before the next point: This is the length of time that the laser point stays in this location after it has been sighted. Select a time in which you can comfortably mark / bore or do other tasks and then move on to the next point. The time is shown in seconds.

Hint: Should you want to **sight just a single point in the list**, just double-click the row in the list.

Projection accuracy
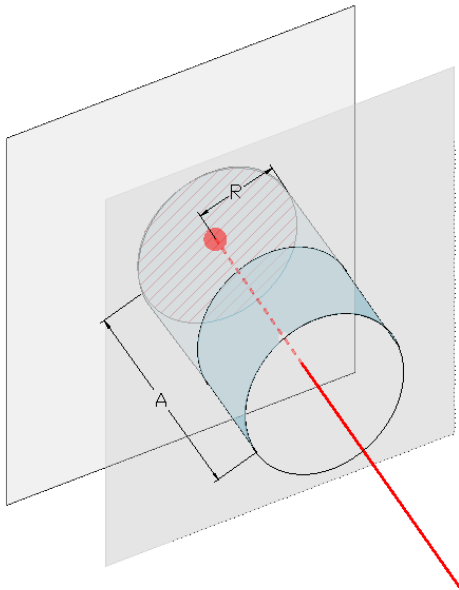
R(max) – maximum radial variance on the projection plane: It defines in what radius around the design point the projected point can move. Or in other words, how well the projected point must agree with the given point. The smaller the value, the more precise is the projection, but the longer the time needed to sight the point.

A(max) – maximum distance from the projection plane: It defines by how much the projected point may deviate from the projection plane. With other words: How far before or behind the wall the projected point may lie.

Manual measurement – Laser distance meter box

	Command:	OPENLASERBOX
	Ribbon tab	As-Built TS Design

This command opens the As-Built – Laser distance meter box

You will find further information regarding the laser distance meter box in the description of the next command → **Manual measurement – Open laser meter Box.**

Manual measurement – Connect laser distance meter

	Command:	STARTLASER
	Ribbon tab	As-Built TS Design

This command initializes the communication with a Bluetooth compatible laser meter.

The laser distance meter box opens:

The **Button** ⟩⟩ opens the dialogue ‚Laser distance meter settings'

Tab 'Connection'

Via this dialogue the laser distance meter is connected with As-Built for AutoCAD. It requires a working Bluetooth communication between the computer and the laser. For setting this up please refer to this manual or the documentation of your Bluetooth instrument.

Select the type of the laser distance meter from the list.

Select the device with which you want to connect via Bluetooth

Displays the status of the connection.
Red – problem with COM port
Yellow – no connection
Green - connected

Connect As-Built to your laser distance meter.

Tab 'Sending options'

You can determine the way you want to process the values during or after the measurement.

Depending on the selection in the setting menu, the values will be removed from the list or remain there in order to be re-used.

Here you can set the precision of the measured values.

Keep or remove values from the list that have been used once.

Send a TAB automatically after the measurement value has been transferred to a dialogue field in order to switch automatically to the next entry

Send RETURN to the command line after the measurement value has been transferred in order to end the command automatically or confirm the entry.

Leica Disto D810: Switch on/off the direction arrow function

A minimum or maximum measurement is automatically ended when after X seconds no further measurement is received from the hand held laser.

Tab 'Units'

Here you set the dimensions for your drawing. The most common units can be selected, like:

- Meter, Millimeter, Centimeter
- Feet, Inch, Yard (International)
- Feet, Inch (USA)

This setting means that one AutoCAD unit equals 1 Meter.

If the values measured with the laser distance meter are send in a different unit they are converted immediately.

Tab Protocol:

Here you will set, whether the lengths, measured with the laser distance meter, are to be recorded within the drawing. At every position where there has been a measurement with the laser distance meter, linear dimensioning will be inserted on a special layer.
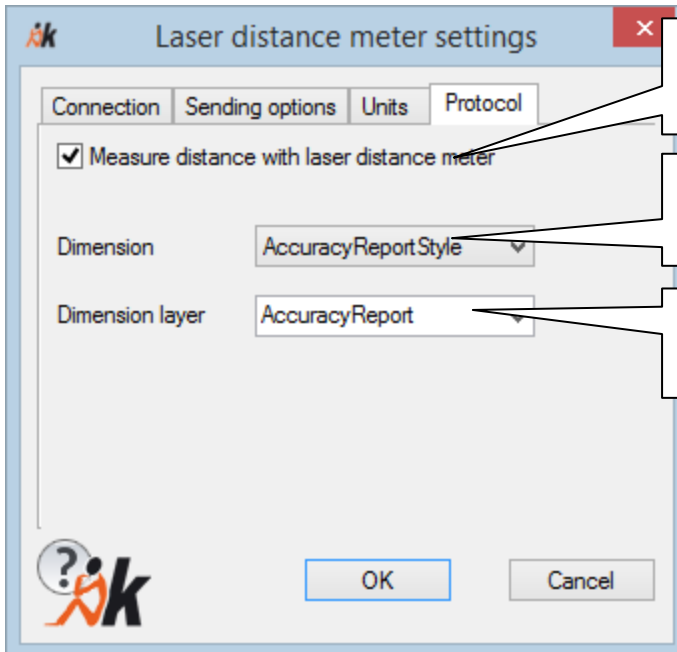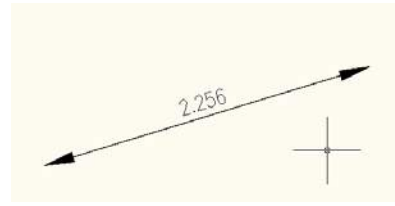
Here you can set, whether the measurements of the laser distance meter are to be recorded.

Select which style of dimensioning the dimensioning is supposed to have.

Select on which layer the recordings are to be written.

Working with the laser distance meter box

After you have measured the desired distance with the laser distance meter, you will transfer it to As-Built for AutoCAD by pushing the according button. The laser distance meter box collects all measured values within a list.

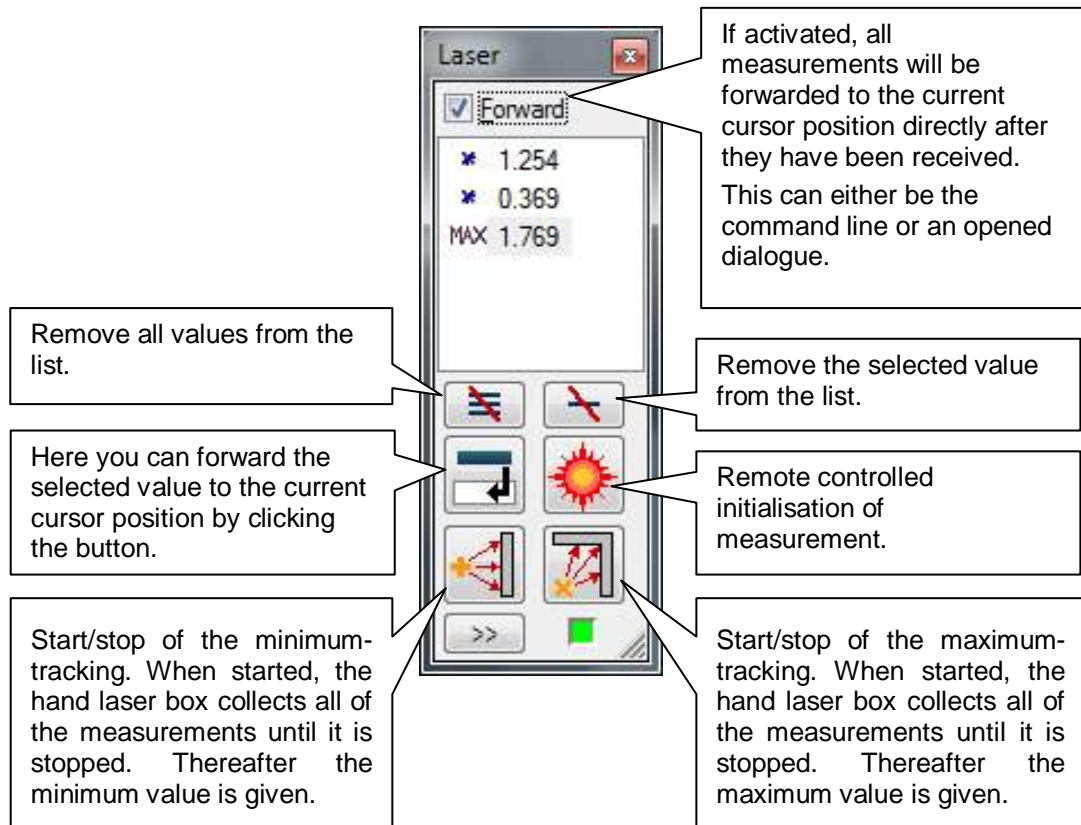If activated, all measurements will be forwarded to the current cursor position directly after they have been received.

This can either be the command line or an opened dialogue.

Remove all values from the list.

Remove the selected value from the list.

Here you can forward the selected value to the current cursor position by clicking the button.

Remote controlled initialisation of measurement.

Start/stop of the minimum-tracking. When started, the hand laser box collects all of the measurements until it is stopped. Thereafter the minimum value is given.

Start/stop of the maximum-tracking. When started, the hand laser box collects all of the measurements until it is stopped. Thereafter the maximum value is given.

There are two different possibilities with which the measured values can be worked with:

- Option "Forward": The measured value is immediately after receipt forwarded to the currently active command (to "where the cursor blinks"). That can be a command line prompt or also an input box in a dialogue box. So before the measurement is made the appropriate command must be started.
- Without "Forward": A stream of measured values are collected in a list. Then afterwards starts the drawing work. You start a drawing command and when you are prompted for a distance, fetch the value from the list by double clicking it or

  by marking it and clicking on ⬛. If you, for example, create a line, you can also set the length of the line by using the selected value via the context menu.

Processes selected value with current command.
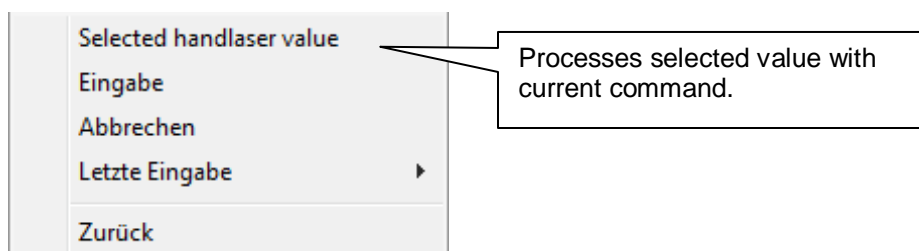
Manual measurement – Disconnect laser distance meter

	Command:	ENDLASER
	Ribbon tab	As-Built TS Design

This command ends the connection to the laser meter.

See also ➔ **Manual measurement** ➔ **Disconnect laser distance meter**

Manual Measurement – Measure perpendicular room

	Command:	TCADROOMOUTLINEPERP
	Ribbon tab	As-Built TS Design

This command is made for efficient construction of rectangular rooms using a laser distance meter. The command calculates compensation of measurement imprecision and reports original lengths versus adjusted lengths graphically. It directly supports the arrow keys on Bluetooth Distos (Leica laser distance meter).

After starting the command you have to define the starting point and the reference direction first.

> Command: **TCADROOMOUTLINEPERP**
>
> Current mode: UCS
>
> Select start point or [oPtions]: **p**

After entering the "p" a window will open, where the following methods will be available:
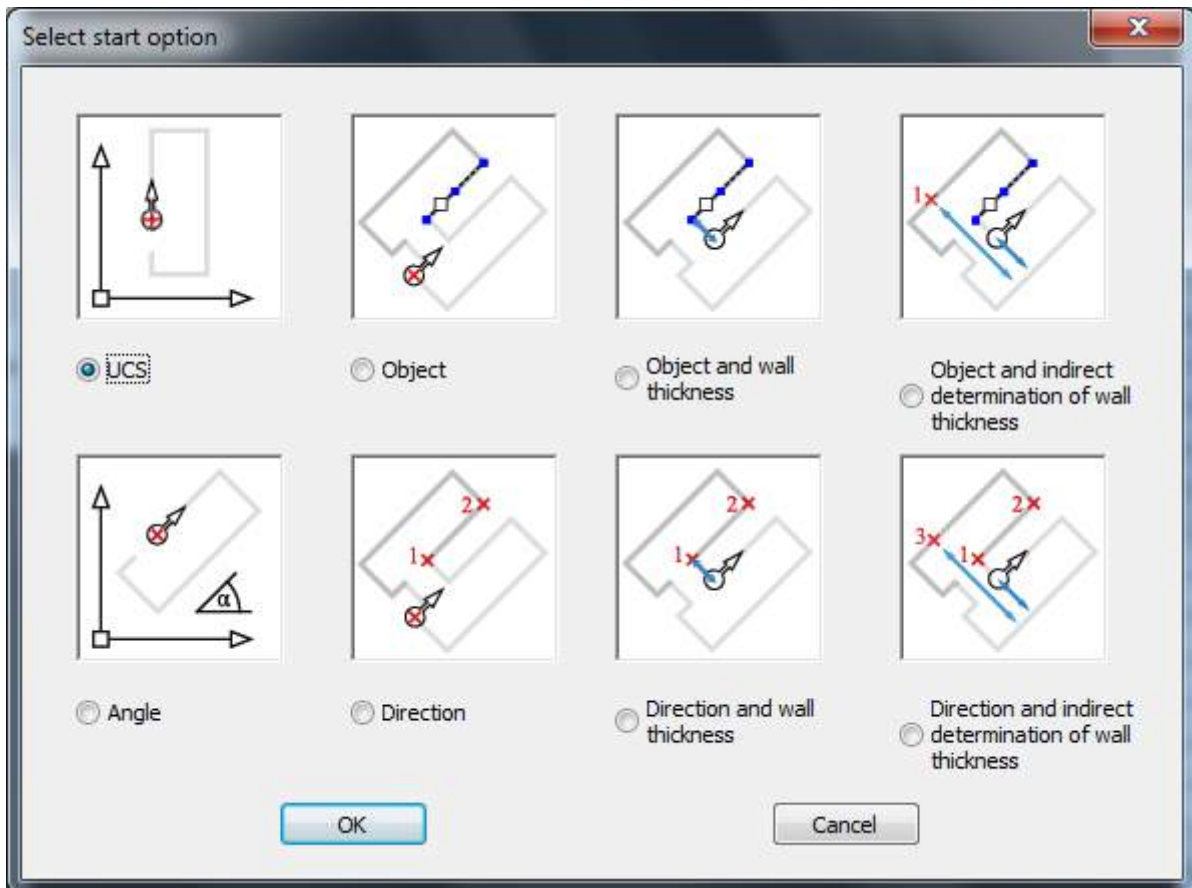
- **UCS**: The directions of the arrows refer to the axes of the current UCS.

- **Object**: The directions of the arrows will be oriented at a selected line object.

- **Object and wall thickness**: Like "Object" but additionally, the starting point will be moved parallel by a certain amount.

- **Object and indirect determination of the wall thickness**: Like "Object", but the wall thickness results from the difference of two distance measurements. Ideal, when the wall thickness may not be measured directly.

- **Angle**: The directions of the arrows will be oriented to the current UCS according to an entered angle.

- **Direction**: The direction of the arrow will be determined by stating two points.

- **Direction and wall thickness**: Like "Direction" but additionally, the starting point will be moved parallel by a certain amount

- **Direction and indirect determination of the wall thickness**: Like "Direction" but the wall thickness results from the difference of two distance measurements. Ideal, when the wall thickness may not be measured directly.

As soon as the starting position and the reference direction have been selected, the following window will open:

Length of the last drawn part of the wall in CAD units.
Either send directly via Bluetooth laser distance meter or entered manually.

Arrow keys in order to state the drawing direction for the wall.
When using the direction keys on the Leica Disto you don't have to use these keys.

Tab length from drawing.

AutoCAD pan function

Parameter for the calculation of the adjustment calculation.
Tolerance: Maximum size of a gap.
Structure size: Minimum length for the automatically generated wall parts.

Current situation regarding "Close..."

Current length error.(size of remaining gap) In this case closing is possible.

Setting options for the tolerance, structure size, reference direction for the direction arrows.

Completing the measurement without closing the room (line ends at last point).

Completing the measurement and closing the room by adjustment of the remaining gap (if possible).
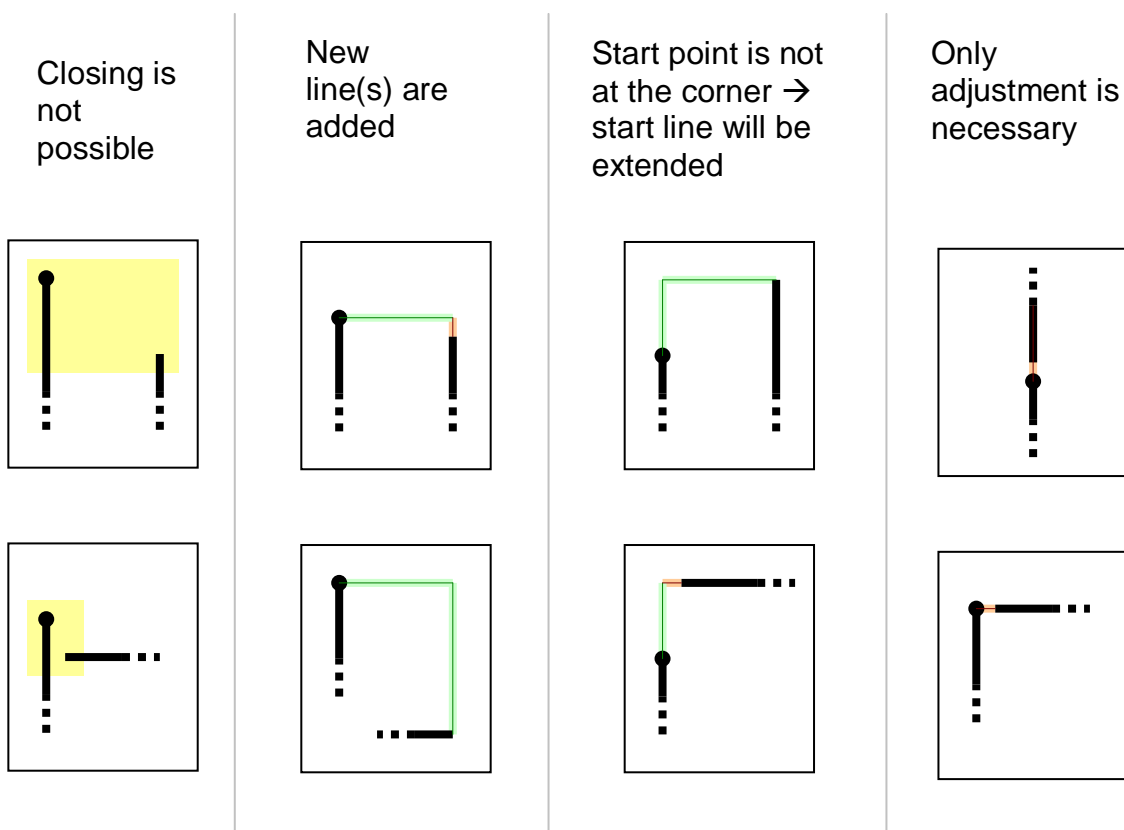
The user can add length and direction by pressing the appropriate keys on the laser distance meter Disto or in the dialog. These steps are repeated until the outline is

complete and the user selects either "**Close outline and finish**" or "**Finish**". The "**Undo**" option removes the last input.

The "Finish" button finishes the command and leaves the open polygon in the drawing. "Finish" makes no adjustments. "Close…" creates a closed rectangular outline with compensation of measurement imprecision. The "Close…" function is only available if a given tolerance (configured in "Options") has not been exceeded and a closed outline would be topologically consistent.

If applicable, a new line will be added. This only occurs if the gap exceeds the minimal structure size. A green line, in the "Current Situation" dialog figure, symbolizes the new line. A red line illustrates the compensation of measurement imprecision.
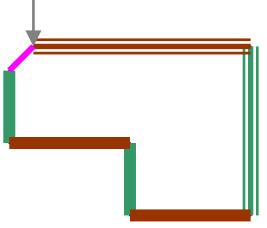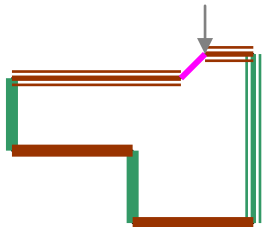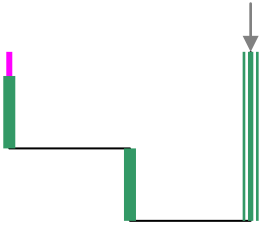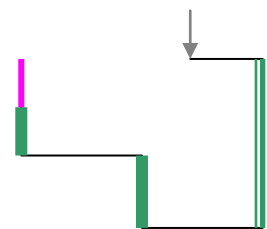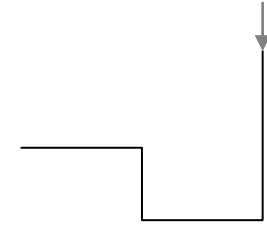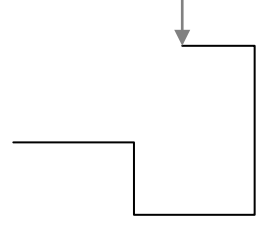
The following situations distinguish this command:

Closing is not possible	New line(s) are added	Start point is not at the corner → start line will be extended	Only adjustment is necessary

Here are some examples for the effect of the "Close outline and finish" option.

On the next page we present six different ways for measurement of an L shaped outline. The final contour will *always* look like this:

	The grey arrow symbolizes the start point. **This position will never change**. The magenta line symbolizes the imprecision of measurements. If imprecision is smaller than the configured tolerance the green and red lines will be recalculated after "Close" in a way that the imprecision gap will disappear (analogue next case)
	16. In this important case (start point is a door) the software will apply the assumption that the red multi lines are co-linear. The green lines will be adjusted and so will the red lines. The result is 17. Sum(▬) = Sum(≡) and 18. Sum(▬) = Sum(≡) and Length(▬) = 0.0
	19. Here the command has to add a line for closing the outline. This way there is no redundancy for the horizontal direction. The green lines will be adjusted. The result is 20. Sum(▬) = Sum(≡) and Length(▬) = 0.0
	This case is similar to the previous one, but the start point is at the opening of a door or marked by a total station, which is situated in an adjacent room.
	21. Here the software has to add two lines and there is no redundancy for an adjustment.
	This case is similar to the previous case, but the start point is again in an opening, of a door or marked by a total station, which is situated in an adjacent room.

In the following situations the option "Close…" is not possible.

After "Close…" or "Finish" the (adjusted) outline appears on the current layer. The adjustments are reported as follows:

Besides that a measurement reading protocol will be generated on the "AccuracyReport" layer (layer may be configured). Every measured distance will be displayed by a dimensioning arrow with labeling.

- Value top: actually measured length

- Value middle: adjusted result length

- Value bottom: difference between actually measured and adjusted length
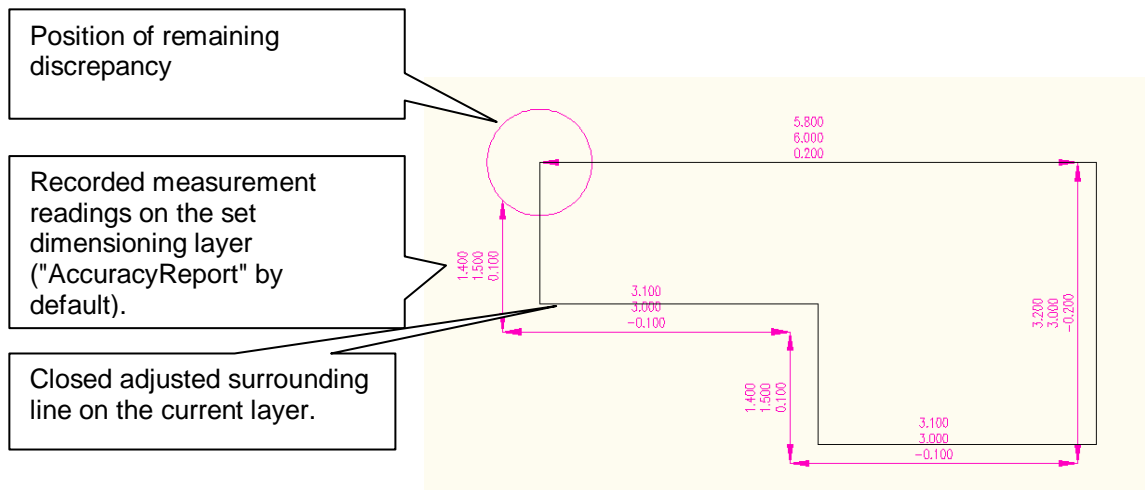
The recording will be an "aligned dimensioning". The dimensioning style and the layer can be selected within the laser distance meter settings → **Laser distance meter – Connect**.

Manual Measurement – Measure variable room

	Command:	TCADROOMOUTLINE
	Ribbon tab	As-Built TS Design

With this command you can measure non-perpendicular rooms with the help of a laser distance meter. The arrow keys on the Leica Disto are supported directly. In contrast to the "Measure perpendicular room" command, this command supports any angle. Therefore you have to do without the adjustment.

After starting the command you have to define the **starting point and the reference direction** of the room first.

Command: **TCADROOMOUTLINE**

current mode: UCS

Select starting point or [oPtions]: p

Upon entering "p" a window will open where the following methods will be available:
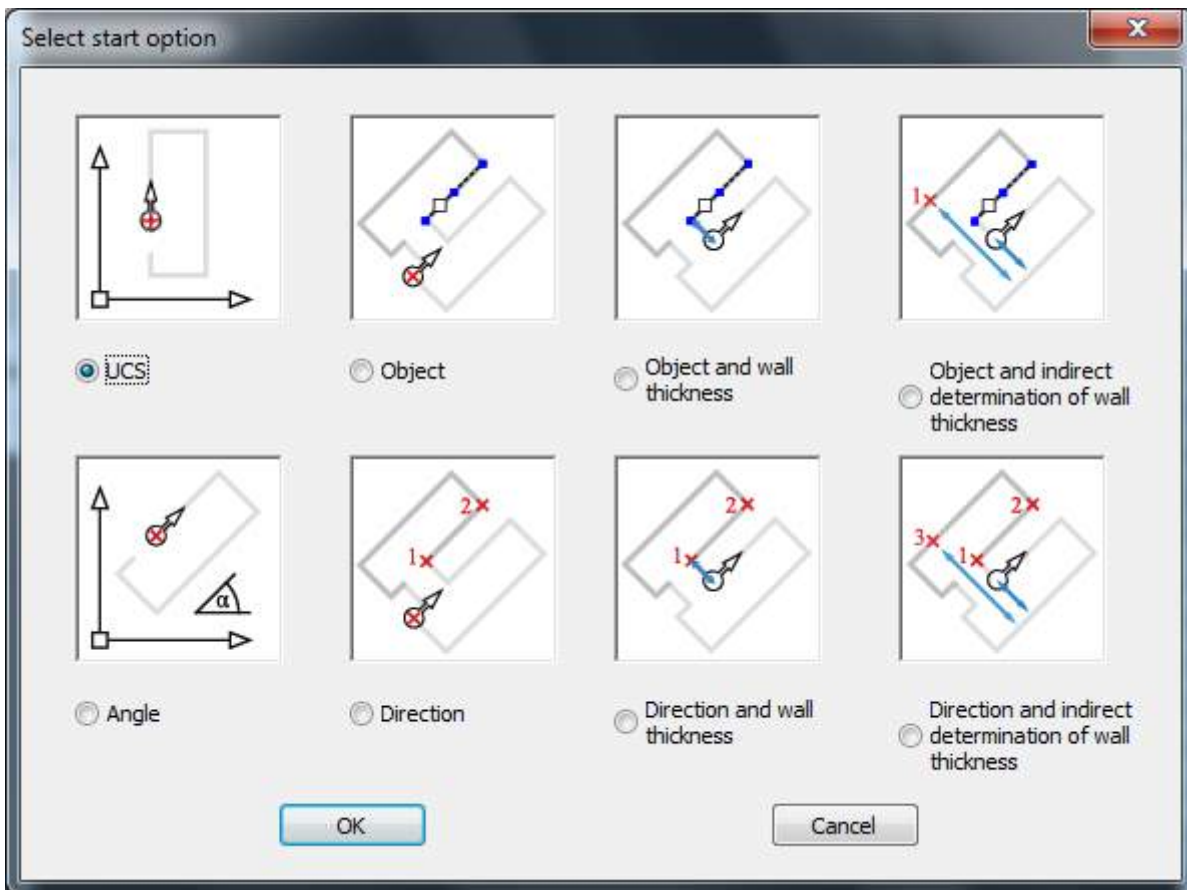
- **UCS**: The directions of the arrow keys refer to the axes of the current UCS.

- **Object**: The directions of the arrow keys will be oriented at a selected line object.

- **Object and wall thickness**: Like "Object" but additionally, the starting point will be moved parallel by a certain amount.

- **Object and indirect determination of the wall thickness**: Like "Object", but the wall thickness results from the difference of two distance measurements. Ideal, when the wall thickness may not be measured directly.

- **Angle**: The directions of the arrow keys will be oriented to the current UCS according to an entered angle.

- **Direction**: The direction of the arrow keys will be determined by stating two points.

- **Direction and wall thickness**: Like "Direction" but additionally, the starting point will be moved parallel by a certain amount

- **Direction and indirect determination of the wall thickness**: Like "Direction" but the wall thickness results from the difference of two distance measurements. Ideal, when the wall thickness may not be measured directly.

As soon as the starting position and the reference direction have been selected, the following window will open:

Length of the last drawn part of the wall in CAD units.
Either send directly via Bluetooth laser distance meter or entered manually.

Arrow keys in order to state the drawing direction for the wall.
When using the direction keys on the Leica Disto you don't have to use these keys.

Tab length from drawing.

Stating any angle for the next wall part.

Reference of direction for the arrow keys:
Global: Arrows will always be oriented on definite, adjustable direction (UP arrow).
Non Global: No reference direction direction. Arrows will always be oriented on line measured last (→means to the right", ←means "to the left"…)

AutoCAD PAN function

Completing the measurement and closing the room outline. There will be no geometry adjustment.

Different options in order to adapt the reference direction.

Completing the measurement without closing the room (line ends at last point).

Bluetooth distance meter with arrow keys:

Perpendicular and 45° wall runs will be drawn by sending the measured distance directly from the instrument via the arrow keys. Then As-Built for AutoCAD instantly draws a line according to the direction of the arrow and the set reference direction.

Wall runs with oblique angle: The measurement values will not be transferred via the arrow keys but via

the Bluetooth key ⦿. A window will automatically open for the entry of an angle or an additional segment. If you now send an additional segment from the laser distance meter, the "length from another point" will automatically be filled in and the window will be closed. Now a reference point will be prompted within the drawing (the point where this segment starts) and then the determined wall line will be drawn. This procedure corresponds to the arc section procedure.

Alternatively you may enter an angle value.
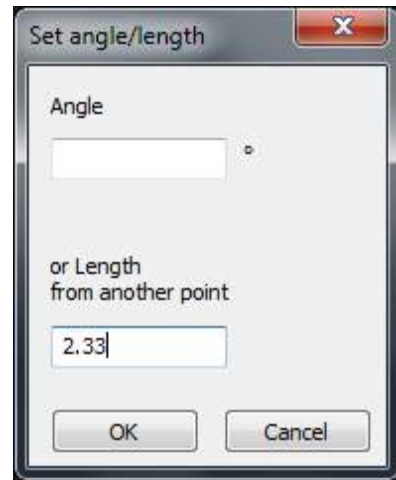
Manual data entry:

Perpendicular and 45° wall runs will be drawn by entering the measured distance into the "length" field. Then click on the corresponding arrow key within the window and the according part of the wall will automatically be drawn. Now you will enter the measurement value, etc.

Wall runs with oblique angle: The measurement value will be entered into the

"length" field as well. Then click the cross key ✖. Now the window "Set angle/length" (see above) will open as well and you may either enter a value for an angle or an additional distance from a known point (arc section). Confirm your entry with OK and the run of the wall will be drawn.

Measurement reading protocol

Besides that a measurement reading protocol will be generated on the "AccuracyReport" layer (layer may be configured). Every measured distance will be displayed by a dimensioning arrow with labeling.

The recording will be an "aligned dimensioning". The dimensioning style and the layer can be selected within the laser distance meter settings → **Laser distance meter – Connect**.

Manual Measurement – Check Distance

	Command:	TCADROOMCHECKDISTANCES
	Ribbon tab	As-Built TS Design

"Check distance" is a command to prove and document the accuracy of a drawing. The command starts with a request in the AutoCAD command line:

Select Line or [2P] <2P>:

The option [2P] stands for an input of two points instead of selecting a line object. If selected, the request changes to:

Select first point or [Object] <Object>:

After specifying a length in the drawing you will be asked to measure the true distance using a distance meter.

The result will be a dimensioning arrow with labeling:

- Value top: actually measured length

- Value middle: adjusted result length

- Value bottom: difference between actually measured and adjusted length
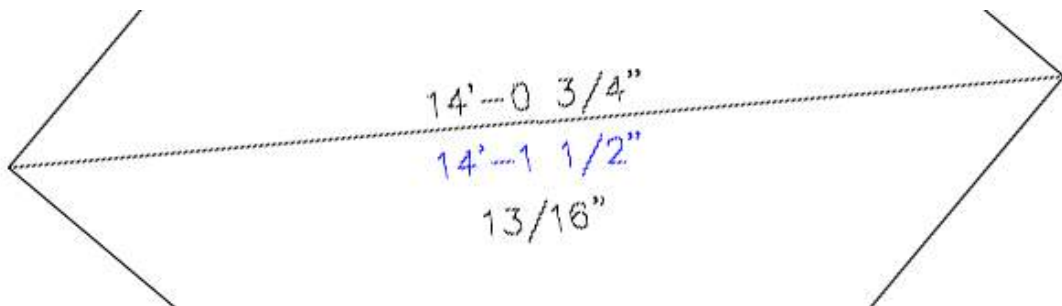
The reported distances are 2D-distances. Any 3D input will be projected on the UCS XY plane.

The dimensioning arrow is of the "Aligned dimensioning" object type. The dimensioning style and the layer can be selected within the laser distance meter settings → **Laser distance meter – Connect**

Manual Measurement – Height dimension

![icon]	Command:	DPHEIGHT
	Ribbon tab	As-Built TS Design

You may insert height symbols into the drawing and dimension the height with this command. After starting the command enter the value for the height first. This might be a value from your hand laser or the distance between two points that you click with your mouse or a manually entered distance. The following window will open
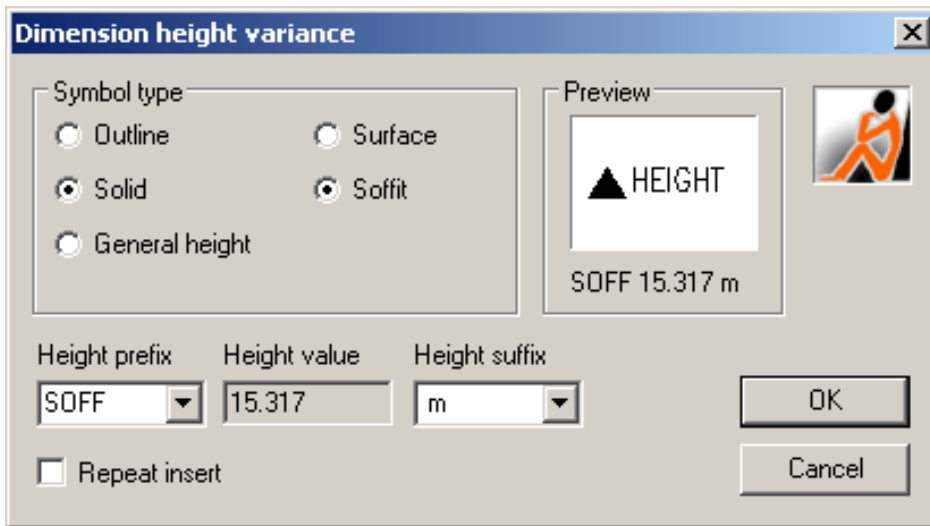
Change the appearance of your height symbol according to your needs by selecting the appropriate symbol option. When you have finished the editing, click OK and select the insertion point for the height symbol within the drawing.

If you have selected "Repeate insert" you will be prompted for a new height right after having entered the first value. The configuration window will not open again. The settings for the first height symbol will be used. To end this process just press ESC.

Remark: The offered symbols are stored as drawings files in the installation folder of As-Built for AutoCAD under \DWGBASE. If you like to change the geometry and the attribute text permanently we recommend to make these changes in those drawing files.

Manual measurement - Arc intersection

	Command:	ARCINTERSECT
	Ribbon tab	As-Built TS Design

The command 'arcintersect' allows to measure a point by hand that cannot be measured directly. This point is determined through the distances to two already measured points.

Command: **arcintersect**

First arc centre: *(select first point)*

First radius: **2.34** *(Enter distance)*

Second arc centre: *(select second point)*

Second radius: **1.65** *(Enter distance)*

Pick side: *(pick a point on the right or left side)*

Plot intersection as [polYline/Point/Lines] <Lines>: *(select object type)*

Depending on the object mode, only one point is inserted or the connections are drawn as line or polyline.

The horizontal distances always refer to the current UCS. The Z coordinate of the point of intersection is determined by the mean value of the Z values of the two base points.

Manual measurement – Arc intersection (transparent)

	Command:	`ARCINTERSECT2
	Ribbon tab	As-Built TS Design

This command allows to determine a point coordinate by the geometrical principle of arc intersection.

Therefore from this point two distances to two known points have to measured by lasermeter or tape. The command calculates the 2D position of this new point.
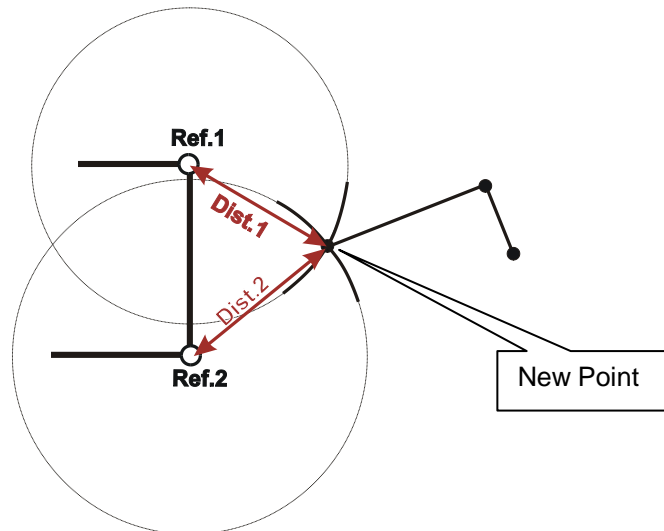
„Transparent" means, that this command can be called inside any other AutoCAD or As-Built command. For instance if you want to draw a line and want to determine one or several breakpoints by arc intersection, the "´ARCINTERSECT2" command is called inside the "LINE" command.

The following command line print may show you how it works:

Command: **line**

Specify first point:

Specify next point or [Undo**]: '_TCADARCINTERSECT2** >> Arc intersection

(call of the transparent arc intersection command)

>>>> Specify first point: (Click first reference point)

>>>> Specify second point: (Click second reference point)

>> Select a side: (click point near to the expected position)

>>>> Horizontal distance from first point: (distance to the first reference point)

>>>> Horizontal distance from second point: (distance to the second reference point)

Resuming LINE command.   (Arc intersection finished. Back to the line command)

Specify next point or [Undo]: 267.0204,149.4127,0.0000   (calculated point is drawn)

Specify next point or [Undo]:   (line command waits for the next coordinate)
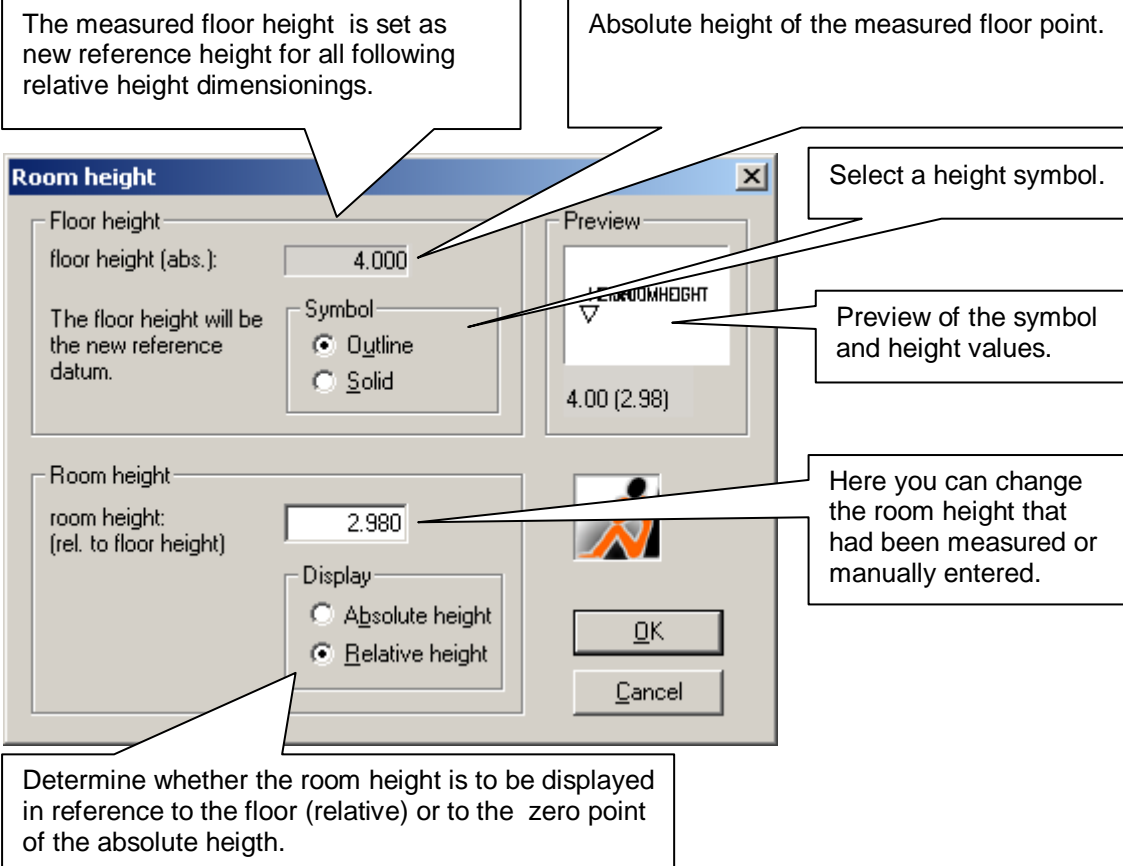
Height dimension - Room height

	Command:	ROOMHEIGHT
	Ribbon tab	As-Built TS Design

The command 'roomheight' 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   *(Measure one point on the floor)*

Point on the ceiling or enter [Value]: <Value>   *(Measure 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 dimensionings.

Absolute height of the measured floor point.

Select a height symbol.

Room height

Floor height
floor height (abs.):   4.000

The floor height will be the new reference datum.

Symbol
○ Outline
○ Solid

Preview

⌐⌐⌐ROOMHEIGHT
▽

4.00 (2.98)

Preview of the symbol and height values.

Room height
room height: 2.980
(rel. to floor height)

Display
○ Absolute height
⊙ Relative height

Here you can change the room height that had been measured or manually entered.

OK

Cancel

Determine whether the room height is to be displayed in reference to the floor (relative) or to the zero point of the absolute heigth.

OK confirms the entries. The height symbol is inserted at the position of the floor point.

The dimensioning block is created on the layer specified in the ➔**As-Built Settings** under the key 'Layerheights'. If this value is kept empty, the current layer will be used. By default this value is set to 'HEIGHTS'.

Height dimension tools - Absolute height

	Command:	ABSHEIGHT
2.41	Ribbon tab	As-Built TS Design

The command "absheight" works as the command "relheight" and is used for measuring and dimensioning absolute heights.

You start with following request:

Command: **absheight**

Measure height point: *(Measure point)*

Following dialogue appears:

Select a height symbol.

This settings shows that you are working with the command 'Absolute height'. Changing to 'Relative height' corresponds to the command 'Relative height'.

You can enter a prefix which will appear in front the height value. (optional)

Preview of the symbol and height values.

Insert height

Symbol
- ○ Outline   ○ Surface
- ○ Solid   ○ Soffit
- ● General height

Display
- ● Absolute Height
- ○ Relative Height

Prefix

Preview

○HEIGHT

6.98

absolute height of measured point: 6.980
relative height of measured point: 2.980
current datum: 4.000

☐ Repeat insert

☐ height of the measured point is the new datum

OK
Cancel

Activate this field if you want to run several height dimensionings with the same settings. You will only be asked to measure/enter the respective height point/value.

If a height point has been entered, it also serves as insertion point for the dimensioning block.

The dimensioning block is created on the layer specified in the ➔**As-Built Settings** under the key 'Layerheights'. If this value is kept empty, the current layer will be used. By default this value is set to 'HEIGHTS'.

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.

Height dimension tools - Relative height

[2.41] ⤳	Command:	RELHEIGHT
	Ribbon tab	As-Built TS Design

The command 'relheight' works as the command 'absheight' and is used for measuring and dimensioning relative heights (according to the current reference height).

You start with following request:

Command: **relheight**

Measure height point or enter [Value]:   *(Measure point or option 'V' for manual entry)*

Following dialogue appears:

Select a height symbol.

This settings shows that you are working with the command 'Relative height'. Changing to 'Absolute heights' corresponds to the command 'Absolute height'.

Preview of the symbol and height values.

You can enter a prefix which will appear in front the height value. (optional).

Activate this field if you want to run several height dimensionings with the same settings. You will only be asked to measure/enter the respective height point/value.

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 created on the layer specified in the ➔**As-Built Settings** under the key 'Layerheights'. If this value is kept empty, the current layer will be used. By default this value is set to 'HEIGHTS'

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.

Height dimension tools - Convert relative to absolute height

	Command:	RELTOABS
	Ribbon tab	As-Built TS Design

The command 'Reltoabs' is used to convert relative heights to absolute heights. The command works only for relative heights that have been created with As-Built for AutoCAD commands. These are:

- Height dimension tools - Room height (Roomheight)

- Height dimension tools - Absolute height (Absheight)

- Height dimension tools - Relative height (Relheight)

- Annotation - Window dimension annotation (Dimwindow)

- Annotation - Create rotated arch plan (Rotarch)

- ...

Height dimension tools - Convert absolute to relative heights

	Command:	ABSTOREL
	Ribbon tab	As-Built TS Design

The command "abstorel" is used for converting absolute heights to relative heights. This command only works for absolute heights that have been created with As-Built for AutoCAD commands. These are for example:

- Height dimension tools - Room height (Roomheight)

- Height dimension tools - Absolute height (Absheight)

- Height dimension tools - Relative height (Relheight)

- Annotation - Window dimension annotation (Dimwindow)

- Annotation - Create rotated arch plan (Rotarch)

- ...

Height dimension tools - Set relative reference datum

▽	Command:	REFDATUM
	Ribbon tab	As-Built TS Design

With the command "refdatum" you can set the reference datum for the relative height dimensioning. The datum can be adopted from a point and be entered by hand.

Command: **refdatum**

The reference datum is 0.000000

Measure point or enter [Value]: *(Measure point for new reference datum or option 'V'*

for entry by hand)

The reference datum has been set to 2.500000

All following relative height dimensioning refer to this new reference datum.

Height dimension tools - Subsequent editing -Set reference height of height dimension

	Command:	REFBLOCK
	Ribbon tab	As-Built TS Design

With "refblock" you can change the reference height of selected As-Built for AutoCAD height blocks in the current drawing.

Command: **refblock**

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.

Height dimension tools - Subsequent editing - Show properties of height dimension

	Command:	HBLOCKLIST
	Ribbon tab	As-Built TS Design

With the command "hblocklist" you can display the current properties for a As-Built for AutoCAD height block.

Command: **hblocklist**

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)*

Height dimension tools - Subsequent editing - Set absolute reference datum

	Command:	ABSDATUM
	Ribbon tab	As-Built TS Design

This command is used for changing the absolute height reference of the drawing. You can find it in the As-Built for AutoCAD menu under 'Height dimension tools' → 'Subsequent editing'.

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 TachyCAD height dimensions that are already existing in the drawing will be adjusted to the new height reference. Following As-Built for AutoCAD 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:

111.81m – 0.7m = **<u>111.11m</u>**
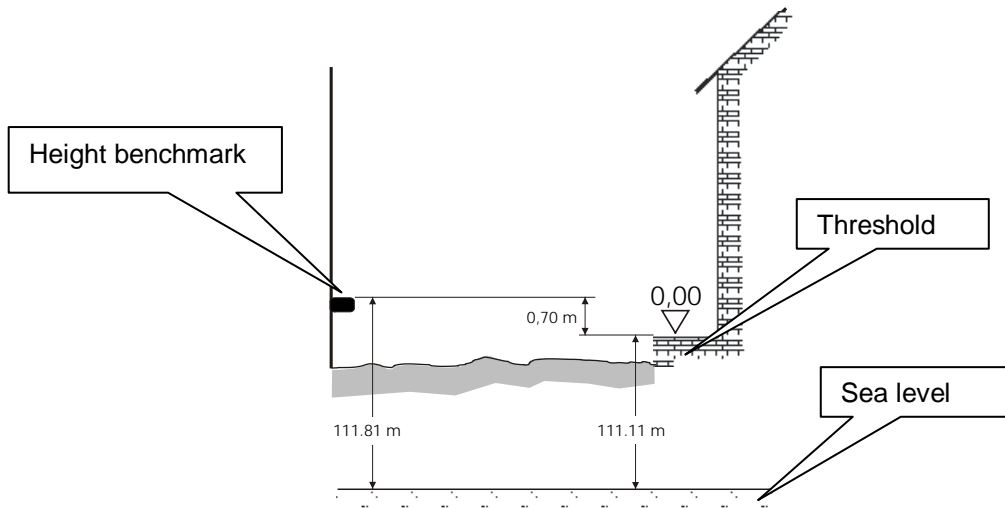
Therefore, the drawing needs to be shifted 111.11m.

When running the command 'ABSDATUM' the following dialogue opens.
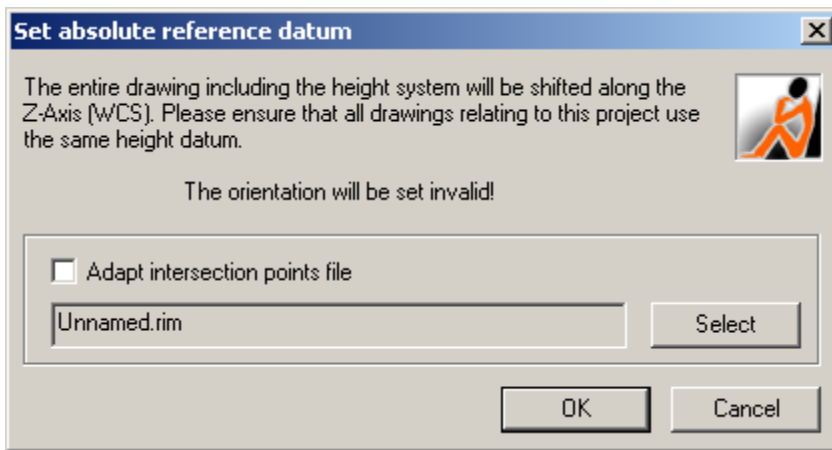
Besides important advice it provides the possibility to select the intersection points file for adaptation. This file with the extension 'rim' is usually used for the 'arc intersection' command. Close the dialogue with '**OK**'. Further query follows in the command line:

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 *.rim*

have been adjusted)

Current orientation deleted.   *(Note: Orientation data have been deleted,*

new orientation necessary!)

Total objects shifted: 39
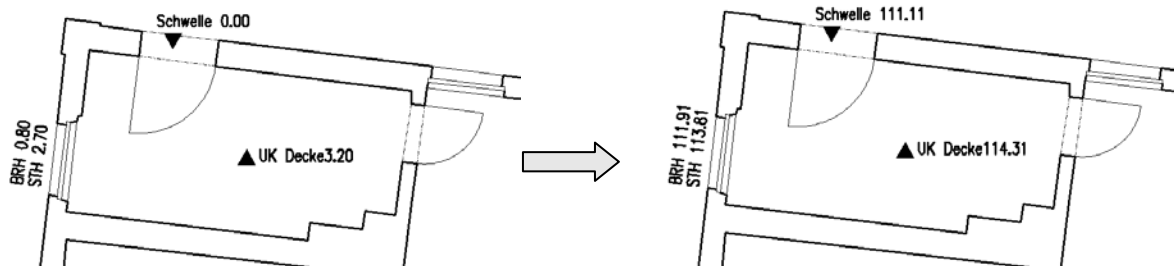
Important advice!

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 (→**Edit drawing** – Flatten). 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.

Annotation - Create rotated arch plan

	Command:	ROTARCH
	Ribbon tab	As-Built TS Design

The command "rotarch" 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: Next to the summit of the arc the polyline must have a vertex.

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 "rotarch" 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 3D polyline [Last]:

Now following dialogue appears:
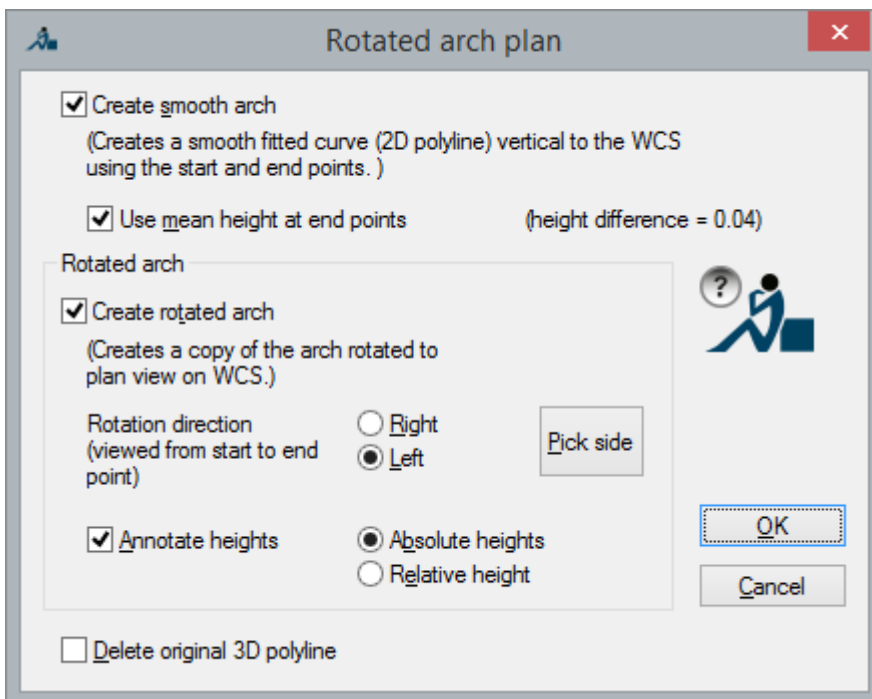
Normally, the lowest height (for arcs pointing upwards) or the highest height (for arcs pointing downwards) are used for the rotation plane. The option "Use mean height at end points" middles the heights of arc ends, which allows to ignore a height difference that exists due to building tolerances, but 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 for AutoCAD. 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 the apex of the arc or close to it. If this is not possible, measure the arc as symmetrical as possible.
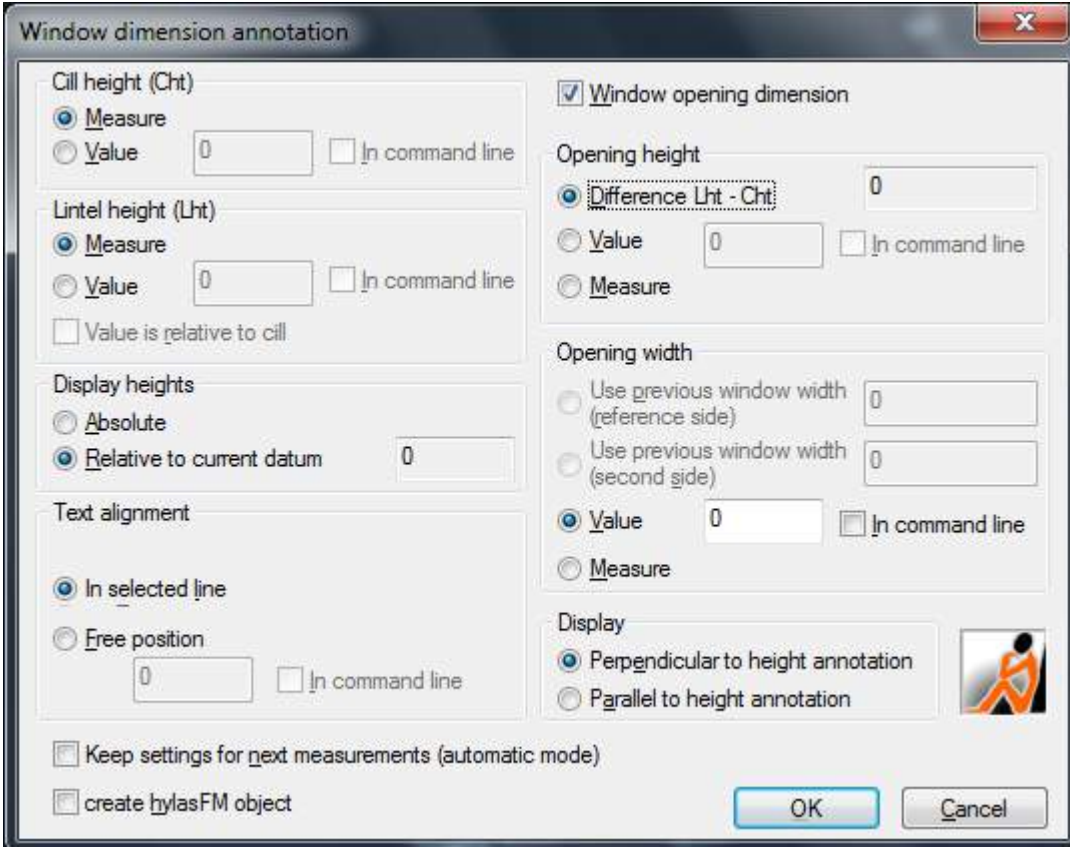
The adjusted arch is created on the same layer as the measured 3D polyline. The rotated arc is set to the layer the name of which is set in the As-Built for AutoCAD ➔**Settings** under the key "Layer – ArchPlanLayer". If this value is kept empty, the current layer will be used. The key "ArchDimensionLayer" is used for the height dimensioning of rotated arch.

Annotation - Window dimension annotation

	Command:	_TCADDIMWINDOW
	Ribbon tab	As-Built TS Design

The command "dimwindow" is used to dimension a window. The necessary values for parapet and lintel height as well as – optionally - the clear 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 measured or inserted manually.

- **'Measure'** means that you will be asked for a point the Z coordinate of which represents the height.

  Measure point on the cill   (*Measure point for lower edge of window parapet*)
  Measure point on the lintel or [UNdo]:   (*Measure 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.

- '**Measure'** 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 measure these points with a tachymeter 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]:

  (Measure point for lower edge of window)

  Measure second point for opening height or [UNdo]:

  (Measure 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:

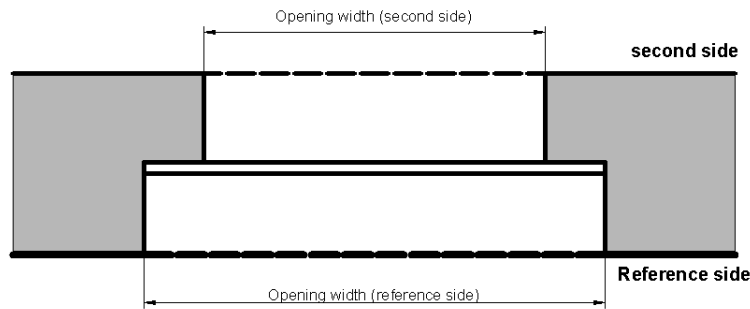 Enter value for clear height or [Undo]: **2.3**   *(insert measured value)*

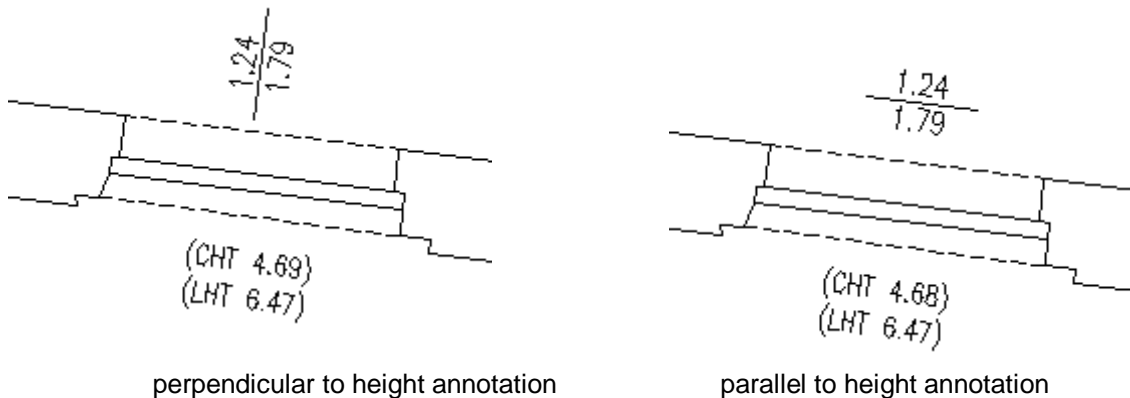 Enter value for clear width or [Undo]: **0.9**   *(insert measured value)*

- **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 will be 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: **dimwindow**

Measure point on the cill or [UNdo]:   *(Measure a point on the window parapet)*

Measure point on the lintel or [UNdo]:   *(Measure 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 for AutoCAD command ➔**As-Built Settings**.

Create feature data object

This option can only be applied, when you use the feature data management of As-Built for AutoCAD. If the used structure definition has a predefined class "Window", a window feature data object will automatically be generated additionally. The attributes "Height" and "Width" will automatically be filled with the opening measurements.

Annotation - staircase dimension annotation

	Command:	TCADDIMSTAIR
	Ribbon tab	As-Built TS Design

The "TCADDIMSTAIR" command inserts a block into the drawing. It has the name "StairDimBlock" and serves as staircase dimension.

Three values are displayed as follows:

| Number of steps | Rise in cm | Going in cm |

16x18.2/25.7

After starting the command the following dialogue box opens up:

Staircase dimension annotation

Number of steps
16   □ In command line

Should the command be automatically called following the command → "**Measure Staircase**" then the number of steps has been already determined.

Rise of a step
○ Select 2 points
⦿ Value   0.18   □ In command line

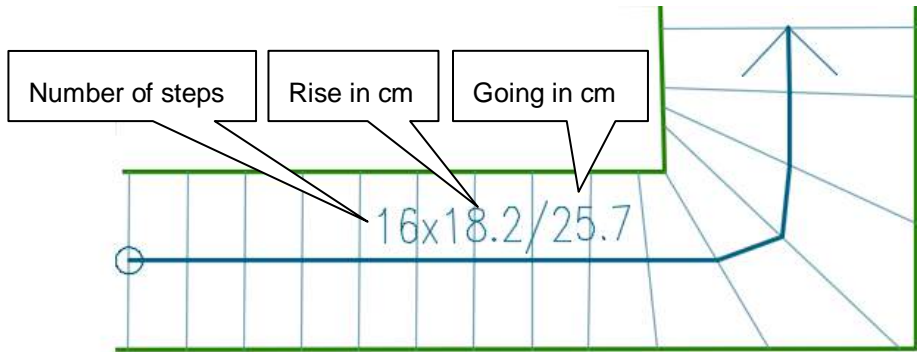Height of a step.
Either by clicking on/observing two points (difference in Z values = riser value)
or by entering the value

Going
◉ Use previous stair going
⦿ Value   0.25   □ In command line

The going of a step. The option button "Use previous stair going" is only selectable when the command has been automatically called following the command → "**Measure Staircase**"

Text alignment
○ Automatic
⦿ In selected line
○ Angle   45   □ In command line

Line or angle for aligning the dimension text

□ Keep settings for next measurements (automatic mode)

Activate automatic mode (for a description see below)

OK   Cancel

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 dimensioing 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 "Building – Stairs – Dimension unit" entry.

Annotation - Beam dimension annotation

	Command:	DIMBEAM
	Registerkarte:	As-Built TS Design

The command 'dimbeam' allows you to dimension the rectangular cross-section of beams.

The cross-section values can either be specified by a number or by the distance of two points.

Reference point for beam cross-section

Insertion point for beam dim. text

0.14 / 0.22

Second cross-section value

First cross-section value
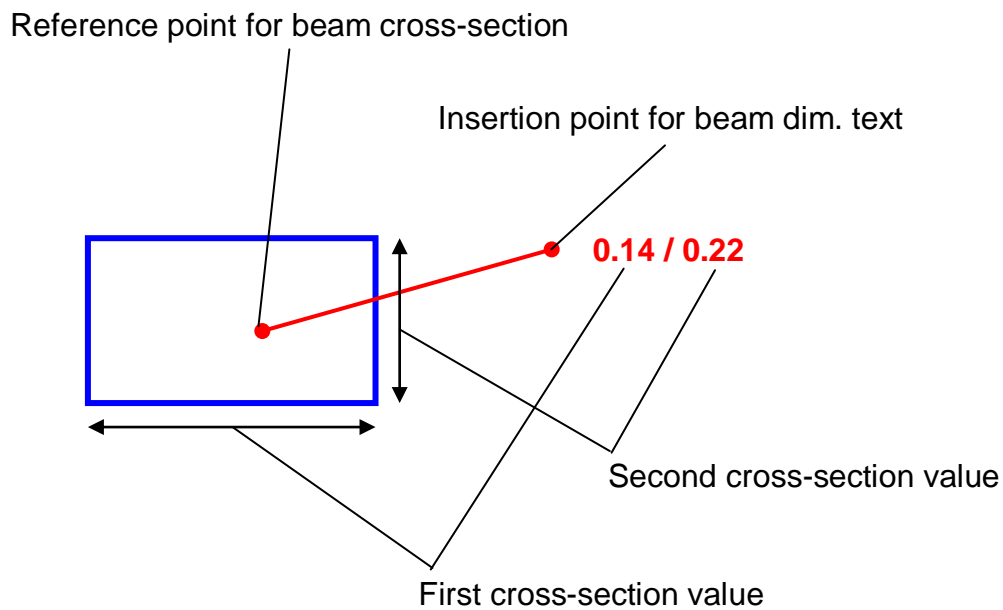
The default setting puts the beam dimensioning on the BEAMDIMENSIONING layer. If you want to change the default setting, you have to use the command →**As-Built Settings**.

Annotation - Height variance annotation

⊼141	Command:	DIMVAR
	Ribbon tab	As-Built TS Design

The command "dimvar" is used for measuring and dimensioning a height difference between two points (opening height). These points can be in any position to each other. After the command has been started, following request appears:

Command: **dimvar**

Measure first point for height variance: *(Measure first point)*

Measure second point for height variance: *(Measure second point)*

After the measurement of the desired height points, following dialogue appears:

Select the symbol that should stand before the dimensioning text

Insert height variance

Symbol
○ No symbol
○ Symbol 1
● Symbol 2

+Dimension

prefix | dimension | suffix
LHT | 2.50 |
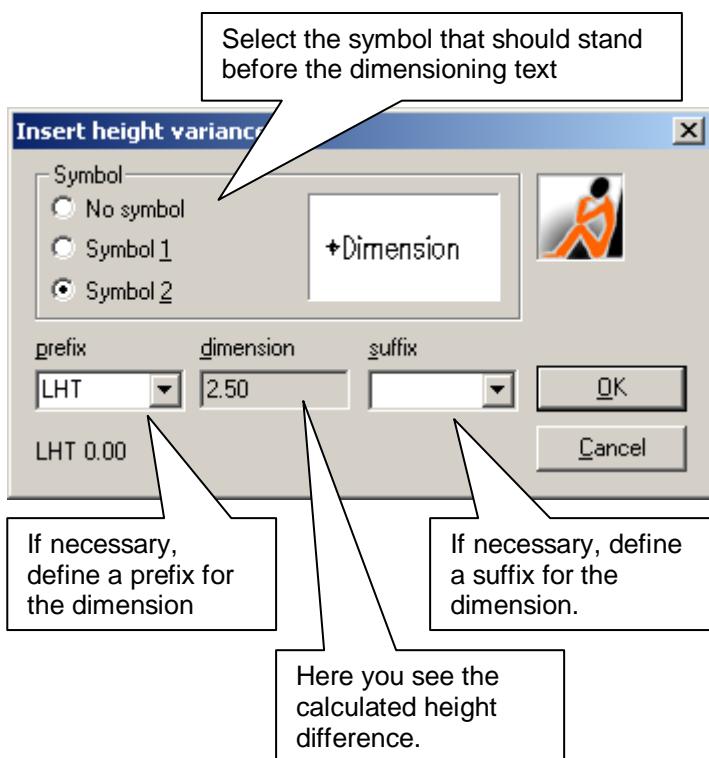
LHT 0.00

[OK]
[Cancel]

If necessary, define a prefix for the dimension

If necessary, define a suffix for the dimension.

Here you see the calculated height difference.

After you have made all settings, confirm with OK. Back in the drawing, following text stands in the command line:

Insertion point of block: *(Select a position for the dimensioning text)*

Enter rotation angle or [Two points] <0.000>:

(select an angle for the alignment of the text or click on the respective point)

In comparison to other dimensioning commands, this command is only used for height differences of two points. The current height reference (absolute or relative) is neither used nor changed.

The command is kept very general in order to allow you to adapt it to your individual needs. Under the As-Built for AutoCAD command ➔ **As-Built Settings** you find the key values:

- Under the key 'LayerOpeningHeight' you can change the name of the layer on which the dimensioning blocks of the drawing are recorded. If you keep the value empty, the current layer is used

- The key values "BASICDIM1" - "BASICDIM3" determine the name of the symbols as they should be displayed in the dialogue. (in the example 'Without symbol', 'Symbol 1' and 'Symbol 2'.

- The block templates for symbols and keys can be found – as all As-Built for AutoCAD blocks – in the As-Built for AutoCAD installation directory under DWGBASE under the filename BASICDIM1.DWG, BASICDIM2.DWG and BASICDIM3.DWG. There you can also adapt their geometry.

Annotation - Insert numbered point

	Command:	NUMPOINT
	Ribbon tab	As-Built TS Design

The command 'numpoint' is used to insert a point symbol into the drawing by entering its coordinates and point number. The coordinates need to exist in the World Coordinate System (UCS) i.e. in the same system the orientation has been done.

After entering the command, following dialogue appears:

Insert the coordinates of the numbered point here.

Insert the number of the point here.

Alternatively, you can indicate the position of the point in the drawing.

Activate this field if you want to create more than one numbered point.
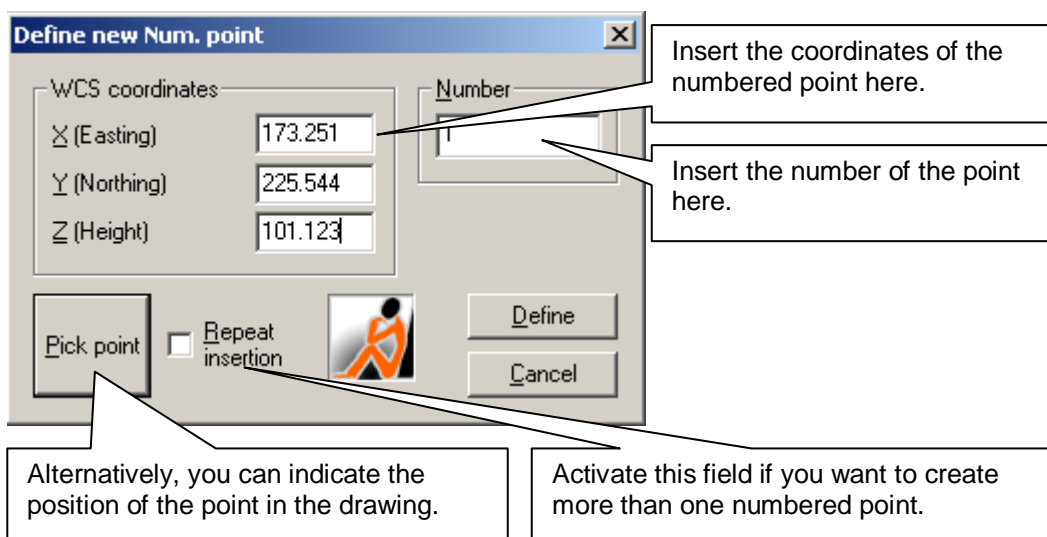
With **Define** a block of the type **NUMPOINT** will be inserted at the indicated position on the current layer. The command also checks if the point number had already been assigned to another point previously. If this is the case, a note will appear. You will then be asked to insert a different point number. The programme suggests a number raised by one.

Hint: Numbered points can be used for several purposes. Here are some examples:

- Offline measured points: You have measured your points with your instrument offline (internal data recording, without As-Built for AutoCAD). Now you want to import these points with a number into your AutoCAD drawing. Use the command ➔**Edit drawing - Import points from file** to import them as numbered points.

- Points to be staked out: If you want to transfer points onto the site with the command ➔**Measurement methods** – Stake out, it may sometimes prove difficult to select every point directly from the drawing. Just number the points to be staked out and insert the numbers in the stake out dialogue.

- In classical surveying certain points have to be marked sometimes (e.g. height points or border points). However, you do not always want to use these points as control points, but you still want to emphasise them. For this you can use numbered points instead.

- In the documentation of archaeological excavation numbered points are well suited to mark the finds.

Annotation - Insert north arrow

	Command:	NORTHARROW
	Ribbon tab	As-Built TS Design

With the command 'northarrow' you can insert a north arrow symbol into the drawing.

> Command: **northarrow**
>
> Insertion point of block: *(Select point)*
>
> Enter rotation angle or [Two points] <0.000>: *(Select second point for arrow direction)*

Alternatively, you can enter a value for the rotation angle of the arrow. With the option 'T' for 'Two point selection' you can align the arrow at two existing elements in the drawing.

This is the standard shape of the north arrow symbol:
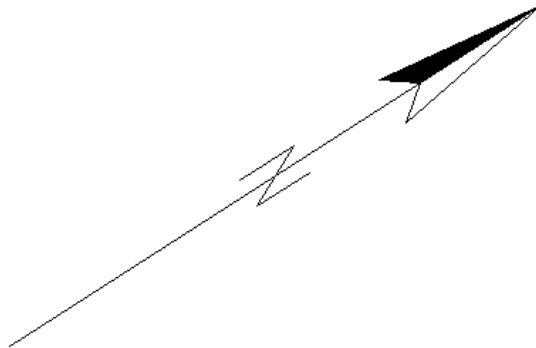
To adapt the shape of the symbol to your individual needs, change the block template 'NORTHARROW.DWG' in the directory \DWGBASE in your As-Built for AutoCAD installation.

Annotation – Insert coordinate frame
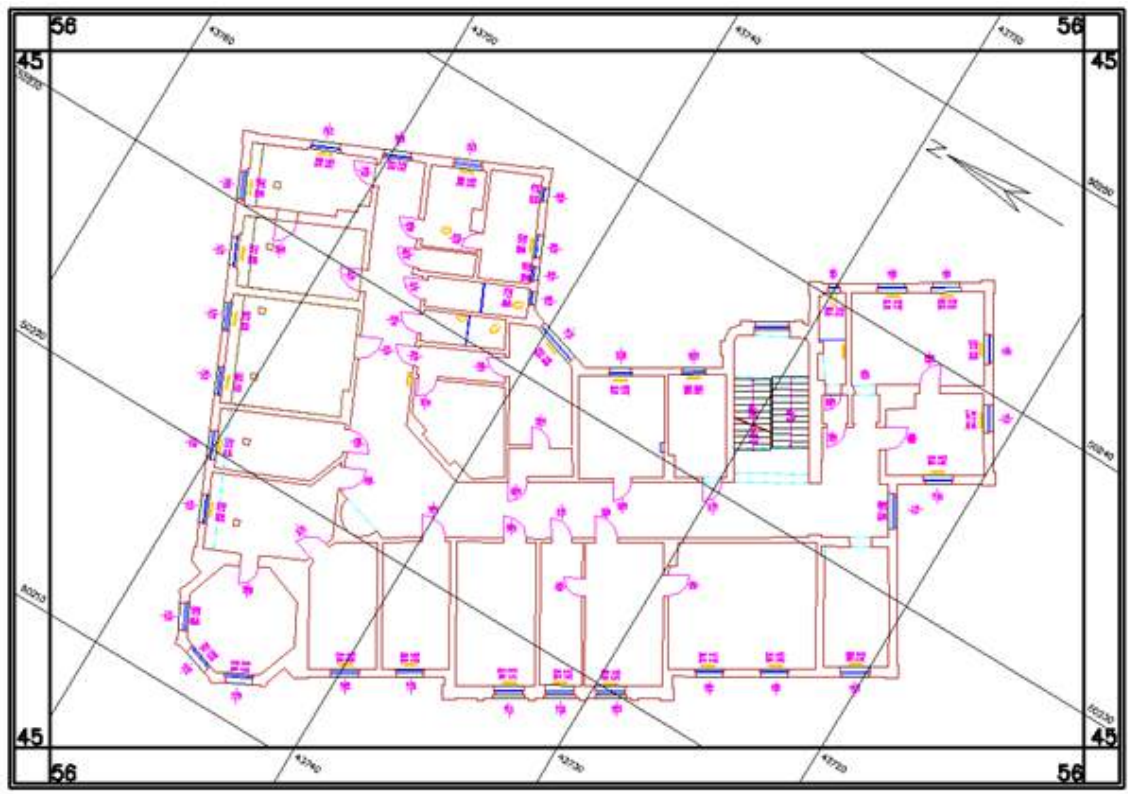
	Command:	COOGRID
	Ribbon tab	As-Built TS Design

With the "COOGRID" command you may insert a coordinate frame with coordinate dimensioning and grid for any perpendicular section into the drawing.

An example:

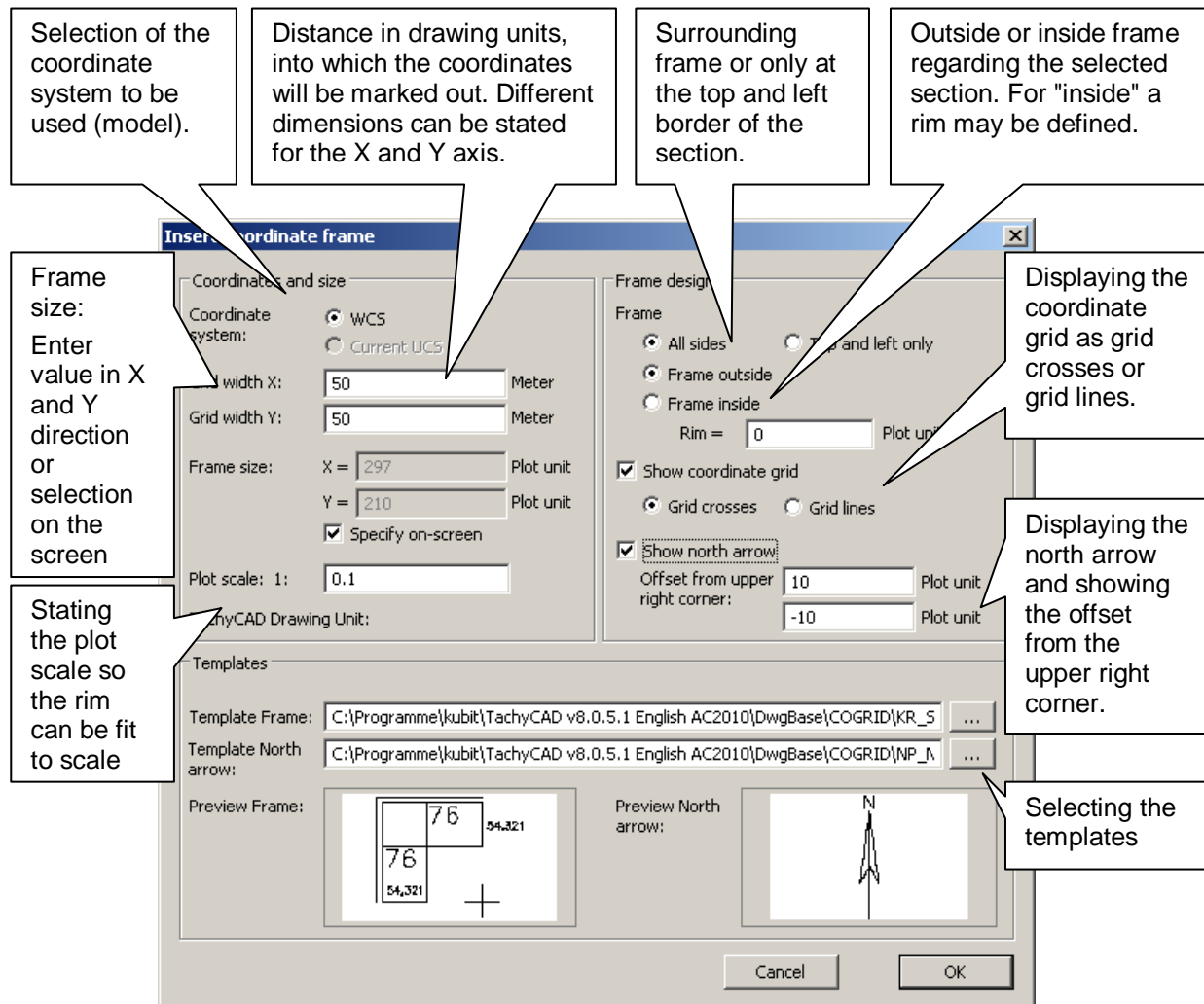The coordinate frame may be featured variably by individually designed templates. But a number of ready to go templates will be provided.

The frame may be inserted into the drawing within the model or the paper area. The coordinate dimensioning is always based on the model are coordinates though. The WCS or the UCS may serve as basis. The top view has to be set for the underlying coordinate system.

Command run

After calling up the command a window will open where all essential settings can be set. If the layout area is active, you will have to select the relevant view first.

Selection of the coordinate system to be used (model).

Distance in drawing units, into which the coordinates will be marked out. Different dimensions can be stated for the X and Y axis.

Surrounding frame or only at the top and left border of the section.

Outside or inside frame regarding the selected section. For "inside" a rim may be defined.

Frame size:

Enter value in X and Y direction or selection on the screen

Stating the plot scale so the rim can be fit to scale

Displaying the coordinate grid as grid crosses or grid lines.

Displaying the north arrow and showing the offset from the upper right corner.

Selecting the templates

Settings

Coordinate system: Either the WCS or the current UCS (if existing) can be used as basis for the dimensioning of the coordinates. In order to be able to insert the coordinate frame, the top view of the according coordinate system (view parallel to the XY plane of the relevant UCS or WCS) has to be selected within the drawing view. If the coordinate system is twisted regarding its XY plane, the entry cannot be selected.

Grid width X and Y: Distance in drawing units in which the coordinates will be marked out. You may state different units for the X axis and the Y axis.

Frame size: You may state a definite dimension for constant frame sizes, so only the lower left corner of the wanted section has to be stated. You will more often use the option to draw a perpendicular section on the screen in order to specify the frame size. In order to be able to do, so activate the "Specify on-screen" option.

Plot scale: The selection of the correct plot scale is important, so the frame layout, like width and font size, is always the same within the plot. The plot scale to be stated

(ratio plot unit in mm to drawing unit) should match the plot scale within the AutoCAD dialog "Plot".

Example model area: A drawing (drawing unit meter) is to be plotted in the map scale 1 : 100. The plot scale then has to be set to 1 : 0.1. Reason: The plot units in AutoCAD are always in millimeters (for metric paper formats). Therefore the wanted map scale 1 : 100 has to be multiplied with the ratio of drawing to plot unit (1 meter / 1.000 millimeters = 0.001) in order to get the matching plot scale or 1 : 0.1

Example paper area: Here it is easier. Since the plan scale is controlled via the view window, you will have, in general, to set plot scale 1 : 1, unless the paper format is to be reduced or enlarged. In that case you will have to adapt the plot scale as well.

Frame design:

- "All sides" The frame will be drawn circular.

- "Top and left only": The frame will only be drawn on the top left rim of the selected section.

- "Frame outside": If this option is selected, the frame will be drawn directly outside of the selected section.

- "Frame inside ": The frame will be drawn within the selected section.

- When selecting "Rim", you can state an offset in order to separate the frame from a drawing frame for example.
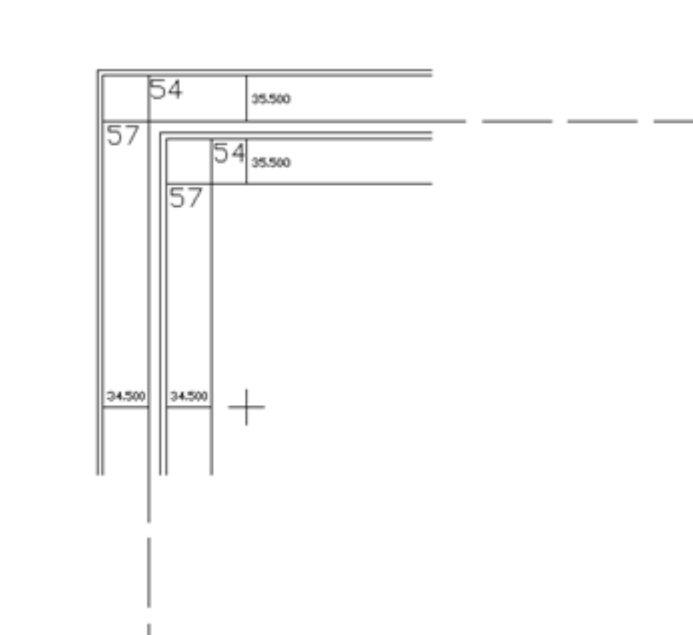
Show coordinate grid:

- Grid crosses: The elements (lines, circles, points, etc.) from the template will be drawn at the grid points.

- Grid lines: A grid with continuous lines will be drawn. The template has to contain lines as cross elements for this.

Show north arrow: As block the north arrow will be inserted into the selected template. The X and Y measure for the insertion position of the north arrow refers to the top left corner of the selected section.

Templates: Here you select the templates for the frame and the north arrow. A small selection of templates will be provided. You will find those within the As-Built for AutoCAD program directory in the sub folder \DWGBASE\COGRID. It is also possible to create your own templates.

Further command run

After you have adjusted all wanted settings, close the dialog with OK. Next you will be prompted to select the section.

For the model area you will always have to select a perpendicular section by selecting two opposite points. For the layout area, the section may match the dimensions of a view (no further selection) or you may select a section that is different from the view.

After you have selected the section the coordinate frame will be drawn. The frame will be inserted as block reference. In order to change the frame afterwards, you will have to use the block editor or use the AutoCAD command "Explode".

Within the paper area the view window is locked by the command execution, so it is impossible to change the view, because this could lead to invalidity of the coordinate frame. You may unlock the window manually (Click window and call up context menu with the right mouse key. Select "No" at "Display locked")

Hint: If the view is twisted, the coordinates will change to the right or left when aligning in north direction. That means, the northing (Y-values) will marked out to the top and the easting will be marked out to the left.

Information regarding the templates:

Next to the provided templates and their individual adaptation like changing the line color and width, font size, etc. you may create your own templates.

The templates for the coordinate frames are AutoCAD drawings, which contain the basic elements of a frame, like frame line, coordinate labeling, crosses or grids.

According drawing objects have been assigned to these elements. Via this assignment the individual drawing elements will be used when creating a frame.

The following elements will be differentiated according to the following figure (grid size = 10):
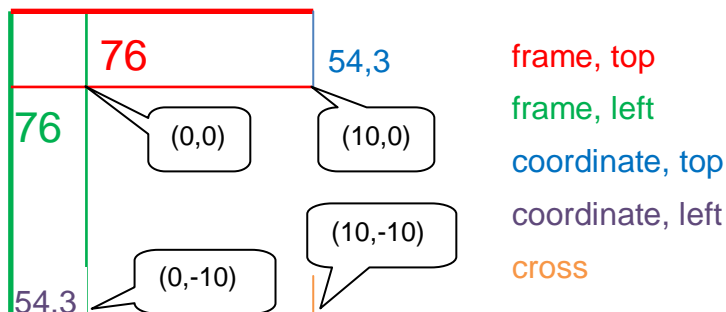
The template drawing shows the top left corner of the coordinate frame. The dimensions of the frame (width, font size) refer to plot units (mm). The origin 0,0 has to be defined at the inner corner. For further elements a square grid of the size x is valid. That means, the top coordinate is at x,0, the left coordinate is at 0,x and the

cross at -x,-x. In order to determine the grid measure by the program, at least one object has to be defined as cross element.

Only lines (no polylines) and text objects are allowed as drawing objects for frame and coordinate elements. You may use circles or points for the crosses though. For the "Frame" element you may use text as leading coordinate digit, so it does not have to be named for every coordinate labeling.

The drawing objects of the template have to be assigned to the individual frame elements separately. You will do this with extended data (xData). In order to do so enter the command "COOGRIDEDITTEMPLATE" into the command line. Select the drawing objects belonging to the elements within the opening window. In order to show the assignment on the screen use the "Show" buttons.

The formatting of the coordinate labeling will be done via statement of the position of the location to be shown. That means, the text "76" puts out the 7th and 6th place before the comma of the coordinate (matches the first two digits for Gauß-Krüger).

The text for the coordinate labeling should generally be aligned centered.

The center or pivotal of the north arrow has to be inserted at 0,0 within a template drawing for the north arrow. The size has to be stated in plot units.

Annotation – 3D distance dimensioning

	Command:	TC3DDIM
	Ribbon tab	As-Built TS Design

With this command you may calculate the distance of two points in space and dimension them. The 3D dimensioning will thereby always be aligned according to the current view, so that you will always be able to read the distance dimension in an optimal way.

First point for dimensioning:

Second point for dimensioning:

The 3D dimensioning is supposed for the fast calculation of distances and the presentation of spatial data.
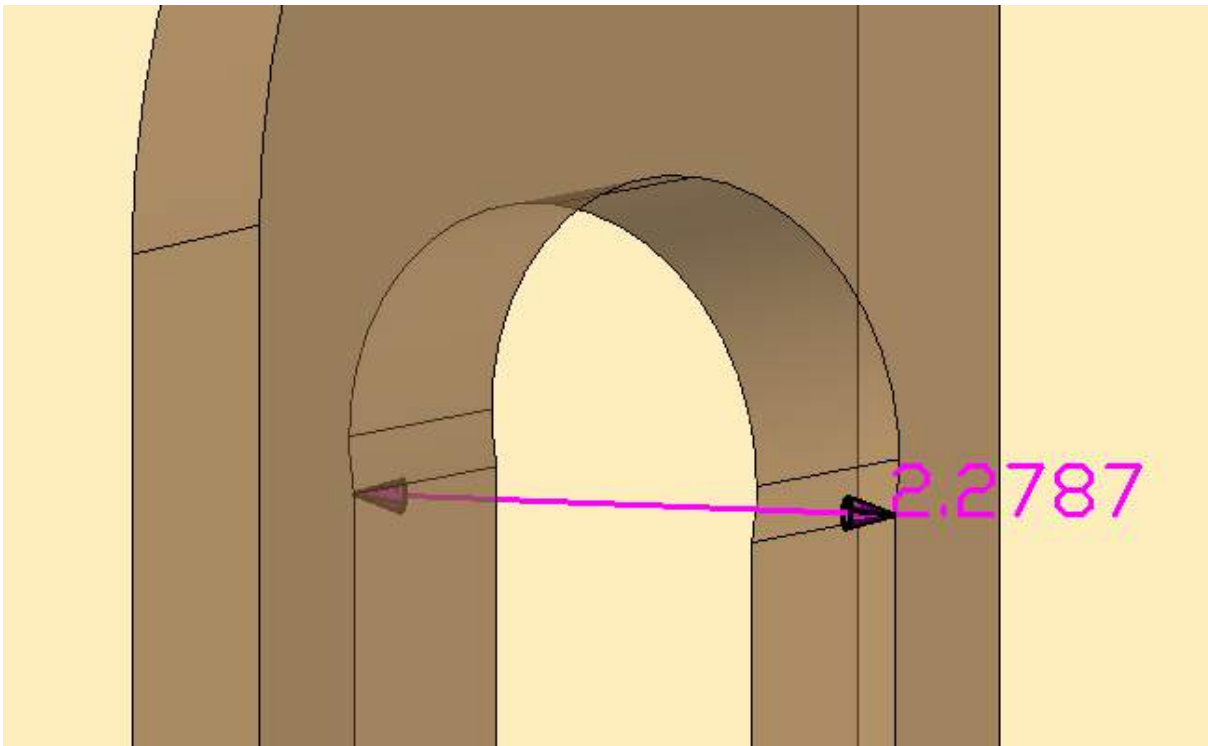
The current AutoCAD text style (text font) will always be used for the display.

The dimensioning will be inserted on the "3DDIMENSIONS" layer by default. You may adapt the layer via the "settings" command, heading "Layer" under "Layer3DDimensios".

Cylinders – Draw cylinder

	Command	KCCREATE
	Ribbon tab	As-Built TS Design

The KCCREATE command creates a new kubit cylinder. You determine the cylinder centerline by specifying two points. Also specify the cylinder radius. Upon calling up this command, following message appears.

> Command: **kccreate**
>
> Select start point of cylinder: *(choose or measure start point of cylinder)*
>
> Select end point of cylinder: (*choose or measure end point of cylinder*)
>
> Diameter: <22.677784>:   *(click point or enter value for diameter)*

You can either enter the radius manually or click on it in the drawing. After the selection of the second cylinder point, a line from that point will appear. The length of this line defines the cylinder radius.

Since the centerline of the cylinder is not directly visible and thus cannot be measured with a total station, this command is unsuitable for measuring cylinders. It is, however, better suitable for constructing and completing the 3D model.

Via the AutoCAD characteristics window you may call up and change the characteristics of the cylinder at any time.

also see: ➔ Cylinders - Measure cylinder

The generated cylinder is, by default, not affixed in its axis.
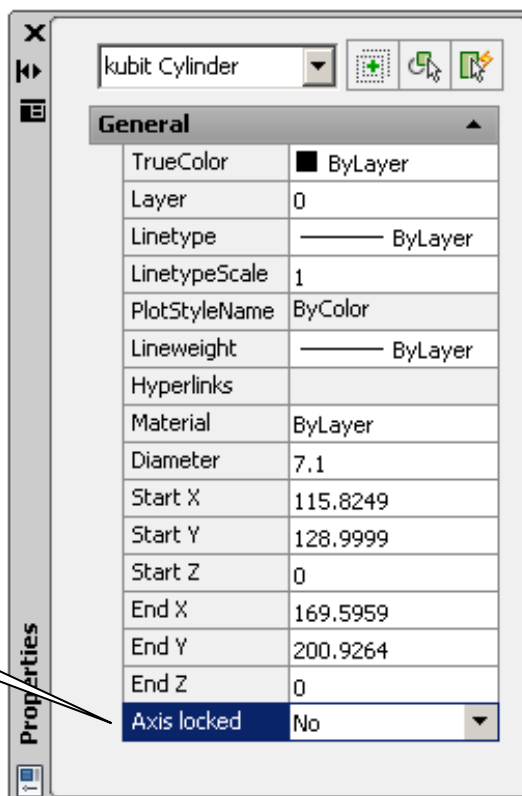
Convert Cylinder into solids: To create an AutoCAD solid from a kubit cylinder, just use the AutoCAD command „EXPLODE"
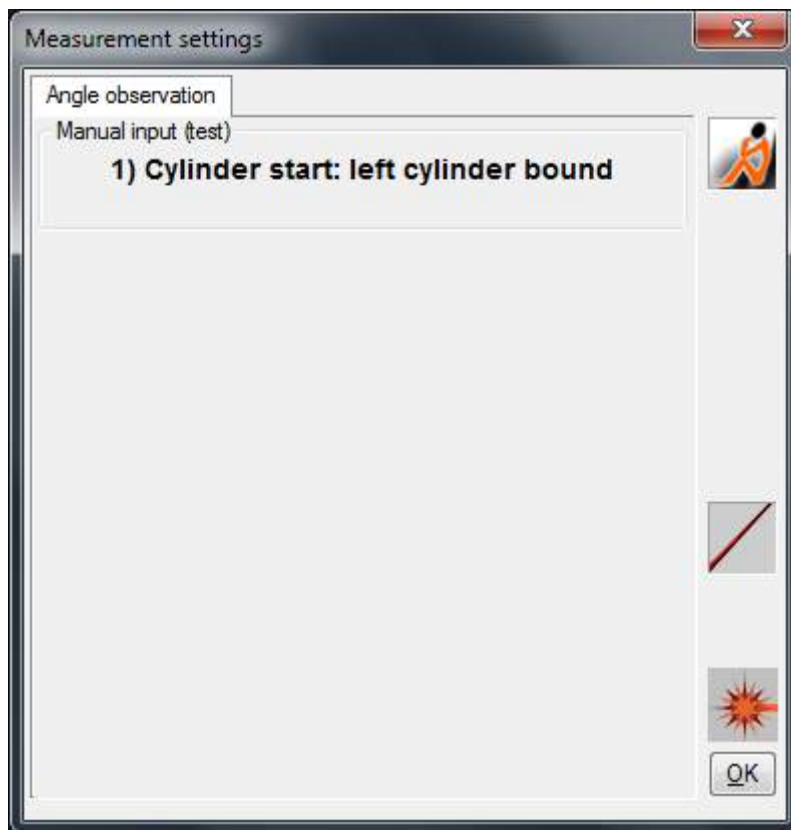
Cylinders - Measure cylinder

	Command	MEASCYLINDER
	Ribbon tab	As-Built TS Design

The command "MEASCYLINDER" allows you to create a new Kubit cylinder by measuring six defined points. You have to measure four tangent points and two arbitrary points on the cylinder surface. With these elements the cylinder is well-defined. The drawing below shows the measuring points and their sequence as seen from the instrument.
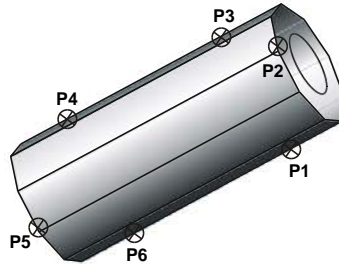
The measuring points are prompted one after the other. The large As-Built for AutoCAD measurement box appears and a short point description is given:

Command sequence:

Three points on first end of the cylinder:

1.: Pipe start: left cylinder bound: *(measure tangent on one side → P1)*

2.: start point on cylinder: *(arbitrary surface point on this cylinder end → P2)*

3.: Pipe start: right cylinder bound: *(measure tangent on other side → P3)*

Three points on other end of the cylinder:

4.: Pipe end: right cylinder bound: *(measure tangent on one side → P4)*

5.: endpoint on cylinder: *(arbitrary surface point on this cylinder end → P5)*

6.: Pipe end: left cylinder bound: *(measure tangent on other side → P6)*

Angle observations are carried out to the tangent points P1, P3, P4, P6. You aim at a point that lies on the "cylinder edge", seen from your station. When you measure the tangent points, only the direction to the target point is important while the distance is not. If you initiate the measurements for these points from your laptop, no distance measurement is carried out at all and only the directions are scanned. If you initiate a full measurement from the instrument, it might happen that your instrument has problems to carry out a distance measurement for tangent points since the laser beam impinges from a very steep angle or only half impinges onto the cylinder surface. In this case you can simply put your hand in front of the laser beam since the distance measurement value is not processed anyway.

A normal full measurement is carried out to the points P2 and P5; i.e. distance and directional measurements. With the points P2 and P5 you determine the provisional cylinder length. The cylinder length can later be changed easily – either with the command → **Cylinders - Adjust cylinder length** or by pulling the "grips" at the cylinder ends.

After calling up the command, a line from the current station becomes visible that is to represent your target line during the measurement.

In contrast to the command →**Cylinders – Draw cylinder,** the cylinder is determined only by elements that you can see from your station. You neither need to enter the cylinder centerline nor radius. These values result from the measured tangent and surface points.

Convert Cylinder into solids: To create an AutoCAD solid from a kubit cylinder, just use the AutoCAD command „EXPLODE"

Cylinders - Adjust cylinder length

	Command	ADJCYLINDERLENGTH
	Ribbon tab	As-Built TS Design

The command " ADJCYLINDERLENGTH" allows you to change the length of a kubit cylinder. Measure a point that lies perpendicular under the cylinder end. If you project this point into the XY plane and drop a perpendicular onto the also projected centerline of the cylinder, you obtain the new cylinder length.
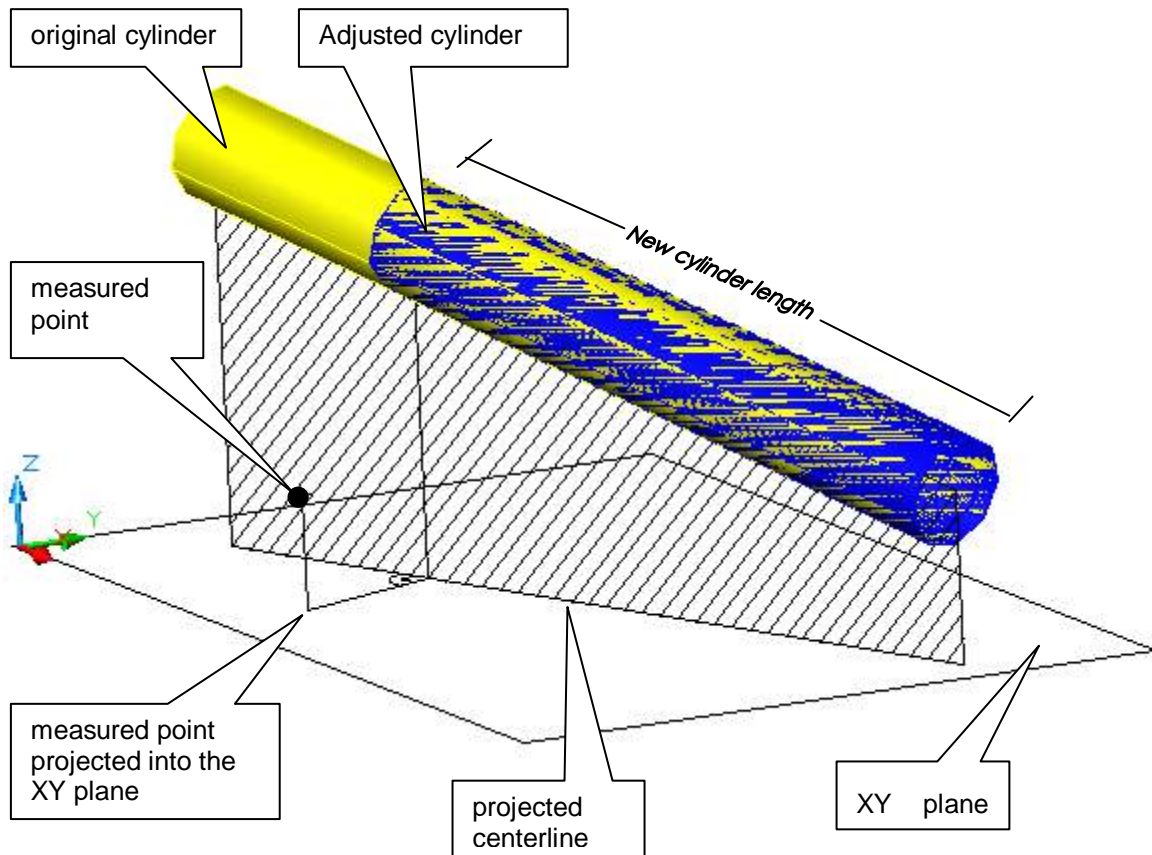
Command sequence:

Command: **adjcylinderlength**

Select cylinder to be adjusted in length: *(click cylinder on the end to be modified.)*

Measure point perpendicular to cylinder end: (*measure point*)

In some cases the cylinder end might not be visible from the instrument station. An assistant could then get into a position under the cylinder end and show the point that lies exactly under the cylinder end on the floor. By doing so you can get a previously measured cylinder section to the right length.

The measuring point can, of course, also lie above the cylinder instead of under the cylinder.

Cylinders - Adjust cylinder radius

Command	ADJCYLINDERRADIUS
Ribbon tab	As-Built TS Design

The command "ADJPIPERADIUS" allows you to alter the cross section of a Kubit cylinder. After you have selected the cylinder, the current radius will be displayed and you will be asked to enter a new value.

Command: **adjcylinderradius**

Select cylinder of which the radius is to be altered: *(click the cylinder to be modified)*

Current radius = 1.985634

New radius or [Measuring point/Perimeter] <1.000000>:

(enter new value for radius or choose option)

option "Measuring point":

The new radius is determined by a point measurement (or point click). The measured point lies on the surface of the modified cylinder.

New radius or [Measuring point/Perimeter] <1.000000>: m   *(Enter "m" for this option)*

Point on surface or [Radius/Perimeter]:   *(measure point)*

option "Perimeter":

With this option you can enter a value for the perimeter instead of the radius. Often the perimeter can be determined easily by means of a measure tape.

New radius or [Measuring point/Perimeter] <1.000000>: p   *(Enter "p" for this option)*

Perimeter or [Measuring point/Radius]:   *(Enter perimeter)*

Cylinders - Split cylinder

	Command	CYLINDERSPLIT
	Ribbon tab	As-Built TS Design

This command allows you to split one Kubit cylinder into several cylinder sections.

Command: **cylindersplit**

Select cylinder to be split: *(click on cylinder)*

Measure point on cylinder or [Finish]<Finish>: *(click/measure section point)*

Measure point on cylinder or [Finish]<Finish>: (*Enter or more points*)

...

Depending on the number of section points, several cylinders are created. These will keep their original properties.

Cylinders - Create coaxial cylinder

	Command	COAXIALCYLINDER
	Module/Ext.	building surveying

The command "COAXIALCYLINDER" is used to create a cylinder that is coaxial to an existing cylinder.

There are several ways of measuring a cylinder. You can measure start and end point directly. Or you measure a cylinder in such a way that the start point is taken over from the reference cylinder. The endpoint can also be defined by entering a cylinder length and the cylinder direction.

The following command sequence is an example of a cylinder that connects to the reference cylinder directly and the cylinder end is calculated by length. When selecting the cylinder, take care that you click the cylinder on the side at which the new cylinder is to be connected.

Command: **coaxialcylinder**

Select cylinder: *(click on reference cylinder on the end where the new cylinder should be placed)*

Measure start point for new cylinder or[from Reference cylinder/Undo] <Reference cylinder>: **r**

       (with option r the new cylinder starts on the end point of the reference cylinder)

Measure end point for new cylinder or [Value input/Undo] <Value input>: **v**

(with option v the length of the new cylinder is determined by a value instead of measuring points)

Pipe length [Measuring point/Undo] <Measuring point>: **20**   *(value for cylinder length)*

Select side or[Undo]:   *(click point on one side of the reference cylinder)*

The newly created cylinder holds the same properties as the reference cylinder.

Cylinders - Create parallel cylinder

	Command	PARALLELCYLINDER
	Ribbon tab	As-Built TS Design

The command "PARALLELCYLINDER" is used to create a cylinder that is parallel to an existing cylinder.

To create the cylinder you are asked to enter four points. The first three points at the cylinder start are used to determine the cylinder radius including the cylinder start. The last point defines the cylinder end. All point measurements on the cylinder need to be carried out as line measurement.
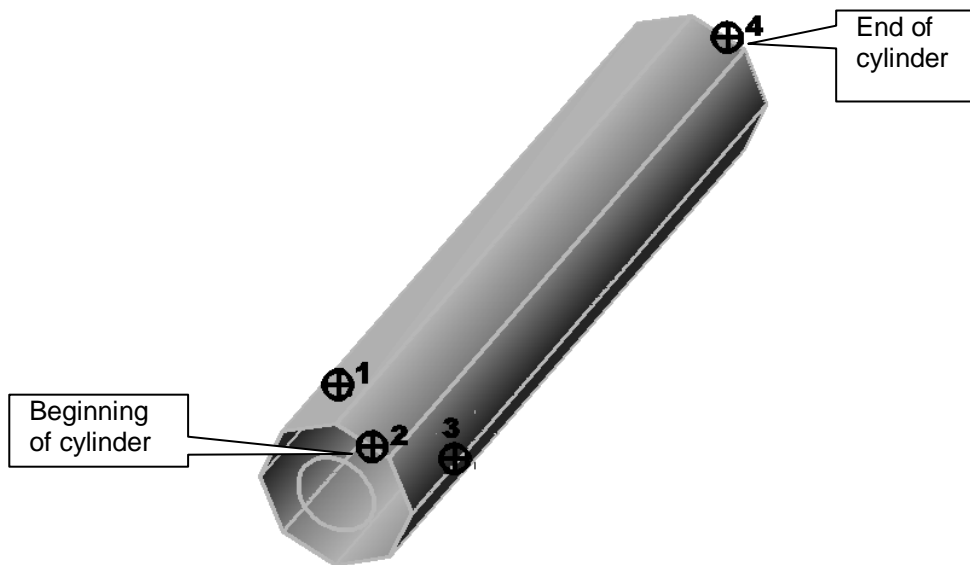
Command:**parallelcylinder**

|	
Select cylinder:	*(select reference cylinder)*
Select point close to left cylinder bound at cylinder start:	*(measure point 1)*
Point in the middle of cylinder at cylinder start or [Undo]:	*(measure point 2)*
Point close to right cylinder bound at cylinder start or [Undo]:	*(measure point 3)*
Point at cylinder end or [Undo]:	*(measure point 4)*

Then the deviation of the measured cylinder from the constructed cylinder is displayed in the command line:

The measured end point deviates from the cylinder wall with 0.063914 AutoCAD-units.

This deviation arises when the measured cylinders are not absolutely parallel. Should the deviation be more than a quarter of the radius, an error message will appear.

Cylinders - Intersect cylinders

	Command	KCINTERSECT
	Ribbon tab	As-Built TS Design

The command "KCINTERSECT" allows you to intersect two Kubit cylinders. There are centerline intersections and cylinder intersections. In the centerline intersection both cylinders are shortened or lengthened until the centerlines meet. In the cylinder intersection, however, both cylinders are shortened or lengthened until the cylinder walls meet.

Initial situation   *Centerline intersection*   *Pipe intersection*
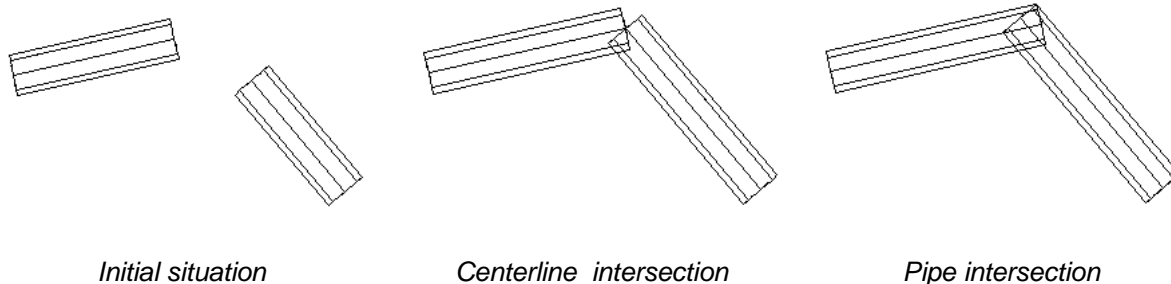
If the selected cylinders lie in a warped position to each other so that the cylinder centerlines would not intersect, the best possible approximation is calculated.

The command to create a centerline section is called up as follows:

Command: **KCINTERSECT**

Center line intersection

Select first cylinder or [Pipe intersection]:   *(click first cylinder on the end to be connected)*

Select second cylinder: *(click second cylinder on the end to be connected)*

If you want to create a cylinder intersection, you can choose this option in the first command line:

Command: **KCINTERSECT**

Center line intersection

Select first cylinder or [Pipe intersection]: **p**   *(option for cylinder intersection)*

Select first cylinder or [Center line intersection]:   *(click first cylinder on the end to be connected)*

Select second cylinder: *(click second cylinder on the end to be connected)*

Cylinders - Connect cylinders (by elbow)

	Command	KCCONNECTELBOW
	Ribbon tab	As-Built TS Design

The command "KCCONNECTELBOW" allows you to connect two Kubit cylinders through a bent cylinder section. The cylinders can lie in a warped position to each other since the connection piece is adjusted through a (spatial) curve. If both cylinders have different radiuses, concentric intermediate pieces will be used. The radius of the cylinder connection is determined by the mean radius of both cylinders.

The resulting bent cylinder section is an AutoCAD solid whose geometry cannot be changed later. Thus, it can also not be edited with other cylinder commands (e.g. "Adjust cylinder length").

Upon calling up the command, following prompts appear in the command line:

Command: **KCCONNECTELBOW**

Select first cylinder: *(click on first cylinder on the end to be connected.)*

Select second cylinder: *(click on second cylinder on the end to be connected.)*

second cylinder

first cylinder

resulting connection piece
→ **AutoCAD solid**
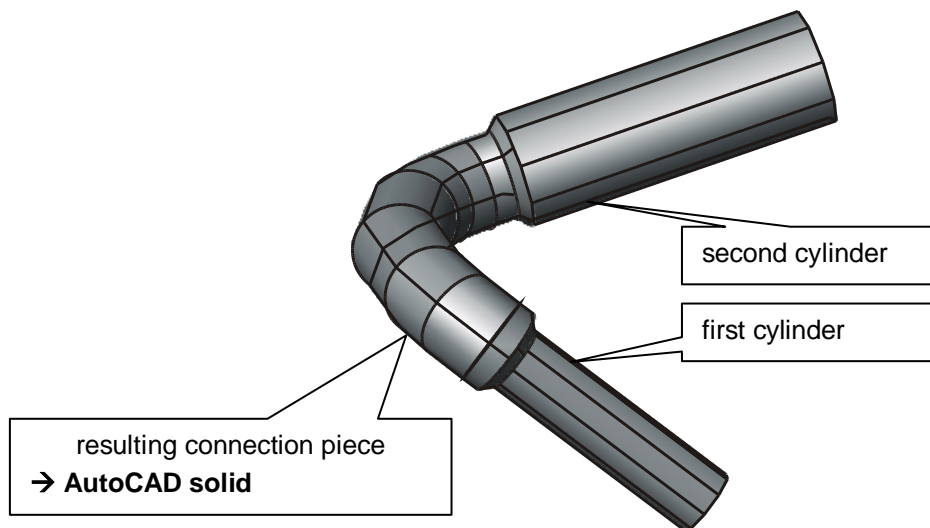
If the selected cylinders are on different layers, a warning message appears. The connection piece is stored on the layer of the first cylinder.

Not all cylinders can be connected by such a connection piece. In case both cylinders are in a very unfavorable position to each other (proportion of cylinder bending to cylinder radius), the "bent cylinder" cannot be calculated. In this case the connection curve is displayed as a 3D polyline and following error message appears in the command line:

Unable to create cylinder connection - angle too small.

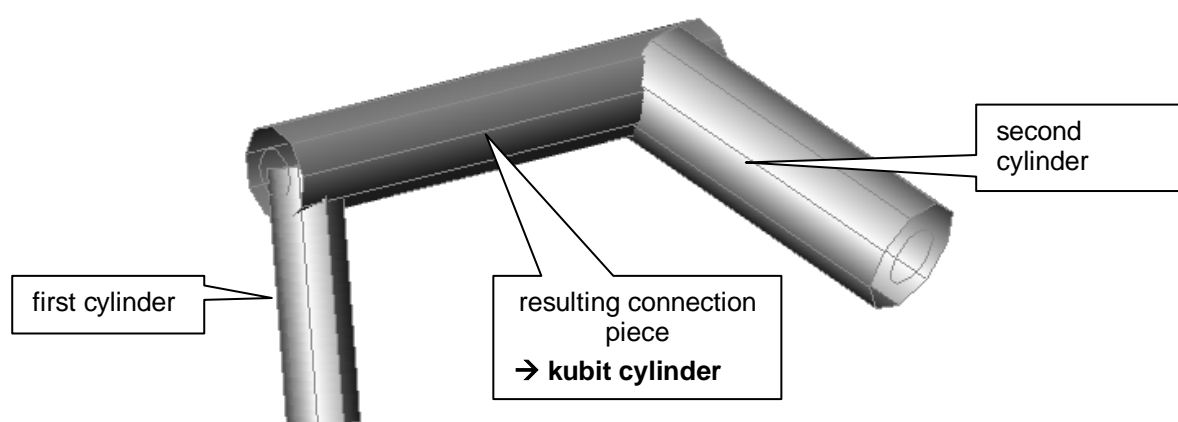## Cylinders - Connect cylinders (by cylinder)

	Command:	KCCONNECTCYLINDER
	Ribbon tab	As-Built TS Design

The command "KCCONNECTCYLINDER" allows you to connect two existing Kubit cylinders by another Kubit cylinder. The two cylinders can lie in any arbitrary position to each other. This command simply interposes another cylinder with mean radius from one cylinder end to the other. In contrast to the command "Connect cylinders through elbow", the interposed cylinder is straight. Here it is a Kubit cylinder and not an AutoCAD solid.

> Command: **kcconnectcylinder**
>
> Select first cylinder: *(click on first cylinder on the end to be connected.)*
>
> Select second cylinder: *(click on second cylinder on the end to be connected.)*

Always click the cylinders on the end to be connected. The advantage of this command is that it allows you to edit the resulting interposed cylinder with other As-Built for AutoCAD cylinder commands later.

A typical area of application of this command is the connection of two pipes that are almost aligned. You can, for example, measure a not visible tee connection with this command:

Measure one section on both sides of the continuous cylinder (a) with →**Measure cylinder**. Then connect these cylinder sections with an intermediate piece using → **"Connect cylinders (by cylinder)"** (b) Now measure a section of the third cylinder (c) and connect it to the main cylinder (d) with " → **Center cylinder**".
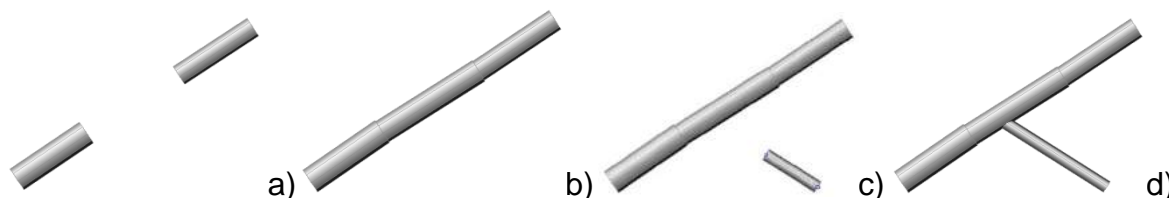
The last command is only possible since the intermediate piece is a kubit cylinder. This command could not be used with a connection created by "Connect cylinders (with elbow)" since that command creates an AutoCAD solid.

Cylinders - Connect cylinders (by torus)

	Command:	KCCONNECTTORUS
	Ribbon tab	As-Built TS Design

This command allows you to connect two existing kubit cylinders by a torus (circular bent part of the cylinder). To true a torus between both cylinders, the cylinder length is automatically adjusted. The torus receives the averaged radius of both cylinder radiuses. In contrast to the command "Connect cylinders through elbow", no concentric intermediate pieces are created. The torus is an AutoCAD solid, whose geometry cannot be changed later. Therefore, it can also not be edited with other As-Built for AutoCAD cylinder commands (e.g. "Adjust cylinder length").

When calling up the command, following prompts appear in the command line and you are asked to select the cylinders. Click the cylinders on the ends to be connected:
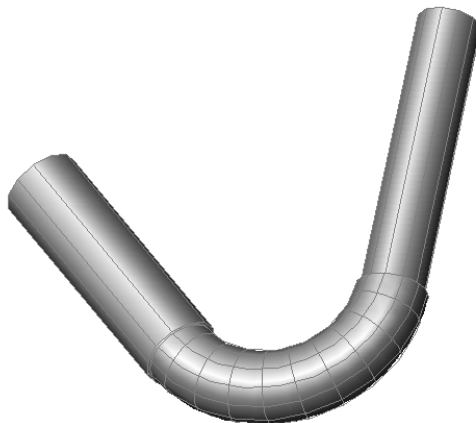
Command: **kcconnecttorus**

Select first cylinder: *(click on first cylinder on the end to be connected.)*

Select second cylinder: *(click on second cylinder on the end to be connected.)*

If the selected cylinders are on different layers, a warning message appears. The connection piece is put on the layer of the first cylinder.

Not all cylinders can be connected through such a connection piece. In case both cylinders are in a very unfavorable position to each other (proportion of cylinder bending to cylinder radius), the connecting part cannot be calculated. In this case the connection curve is displayed as a 3D polyline and following error message appears in the command line:

Extrusion creation failed.

Modeling Operation Error:
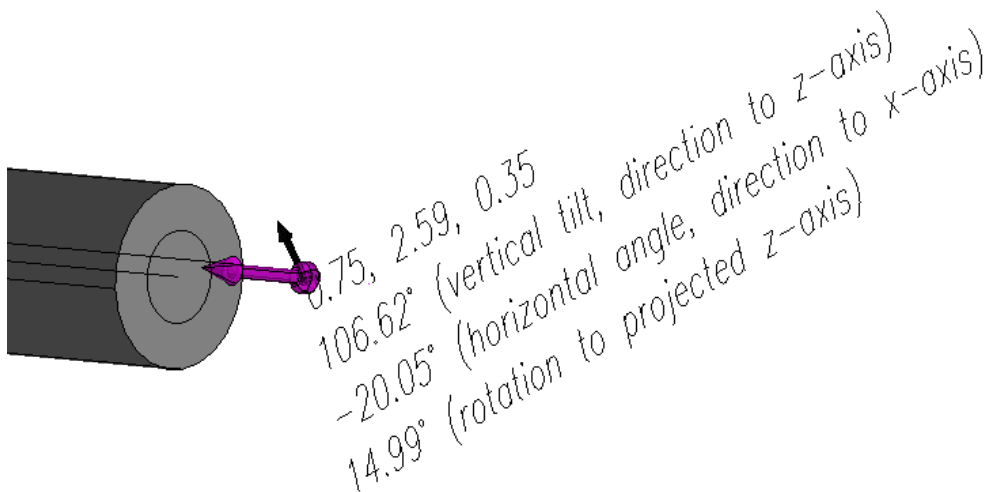
    Wire Intersects Axis.

Unable to create torus.

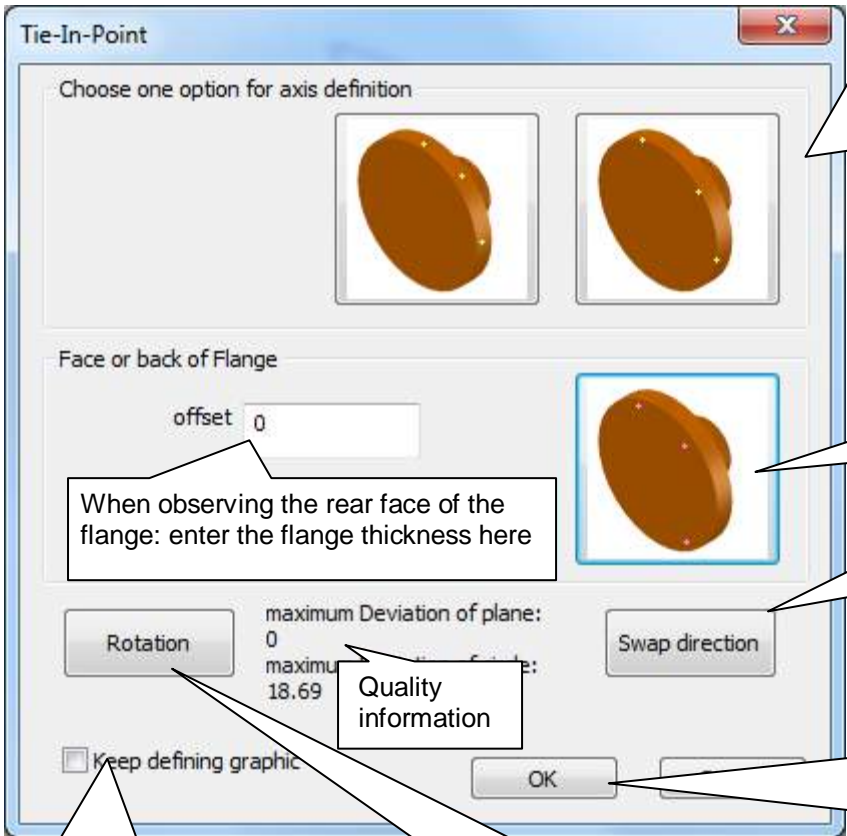**Caution!** This command does NOT work under AutoCAD LT.

Cylinders – Determine tie-in points

	Command:	TCTIEINPOINT
	Ribbon tab	As-Built TS Design

This command is for determining the position of a tie-in point of a pipe flange. 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:

0.75, 2.59, 0.35
106.62° (vertical tilt, direction to z-axis)
–20.05° (horizontal angle, direction to x-axis)
14.99° (rotation to projected z-axis)

Starting the command the displays following dialogue box:

Methods for determining the centreline
• Points on the cylindrical external face of the flange plus points on the face
• Points on the edge of flange

Determine the plane (front face of flange)

Direction of the centreline will be reversed

Plot the results:

Choose one option for axis definition

Face or back of Flange

offset 0

When observing the rear face of the flange: enter the flange thickness here

maximum Deviation of plane:
0
maximu
18.69

Rotation

Swap direction

Quality information

Keep defining graphic

OK

Retain the construction graphic

Determine the alignment of the bolt holes.

The command allows for several procedures. Depending on the visibility/shadows of the pipe select a suitable possibility. The following variants are possible for determining the tie-in point data. It is first assumed that the front face of the flange, or at least its front edge, is visible:

The cylindrical external face of the flange is easily recognisable

At least three well spread points should be measured 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 observe 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.

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 observed that lay well spread out **directly on the edge**. These points are used both for determining the plane and also for determining the lateral surface. This method is especially unsecure because observations on edges can cause imprecision.

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 observed than above mentioned minimum number (and the number shown on the button icons).

Often from the position of the instrument the front face of the flange 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 of the flange. Then it is necessary to additionally give the "shift", the flange thickness. To illustrate the point, all icons on the buttons will be rotated for a shift value that is not equal to zero:
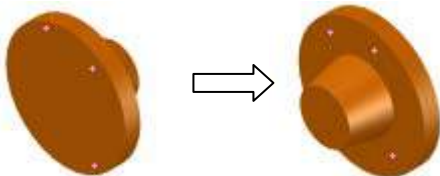
For *better accuracy* observations on the rim of the flange should be avoided. Use instead observations on the rear face together with the additional shift value. Keep a watch on the quality information. Observe more points than the required minimum of three.

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**.

A temporary graphic is inserted into the drawing whilst the data is being determined. The picked points are displayed as polygons and the "Shift" as a blue disc.

The graphic, except for the arrow, disappears when [OK] is clicked, It is possible to keep the graphic by activating the option "**Keep defining graphic**".

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.
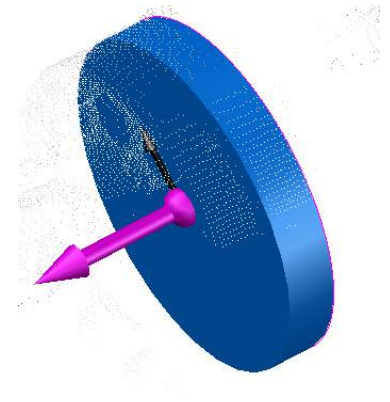
Cylinders - Connect cylinder with plane/hexahedron

	Command	CONCYLINDERPLANE
	Ribbon tab	As-Built TS Design

The command "CONCYLINDERPLANE" allows you to connect a cylinder with a plane by a bent intermediate piece. The bend is calculated in such a way that the cylinder will connect perpendicular to the plane.
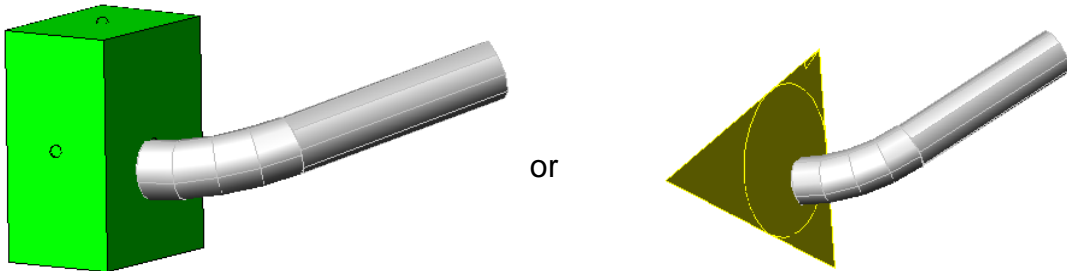
Either a construction plane or the side of a hexahedron can be selected as planes.

Command: **concylinderplane**

Select cylinder: *(click on cylinder to be connected)*

Select construction plane or hexahedron area: (*click on plane to be connected with the cylinder*)

or

Always click the cylinder on the end to be connected. Sides of hexahedrons are selected by clicking the "grip" in the shape of a hemisphere.

The resulting bent cylinder section is an AutoCAD solid whose geometry cannot be changed later. Therefore, it can also not be edited with other cylinder commands (e.g. extending cylinder by the grips).

Should cylinder and plane not intersect (parallelism) or should the cylinder already break through the plane, an error message appears:

Unable to connect the cylinder with the construction plane/hexahedron-area since

the start point of the cylinder already lies on the plane.

 or

There is no intersection point between the center line and the selected plane.

The dimension of the selected plane is assumed to be infinite. Thus, the cylinder connection might also be generated outside the graphic representation of the plane (triangles).

This command is typically used for muzzles and wall breaks of pipes.

The point of break through of the bent cylinder connection through the plane results from the calculation of a minimal bend between cylinder end and plane, as the drawing below explains. The bend is calculated in such a way that lines a_1 and a_2 have the same length.
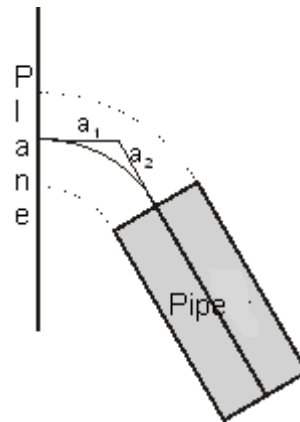
Cylinders - Center cylinder end

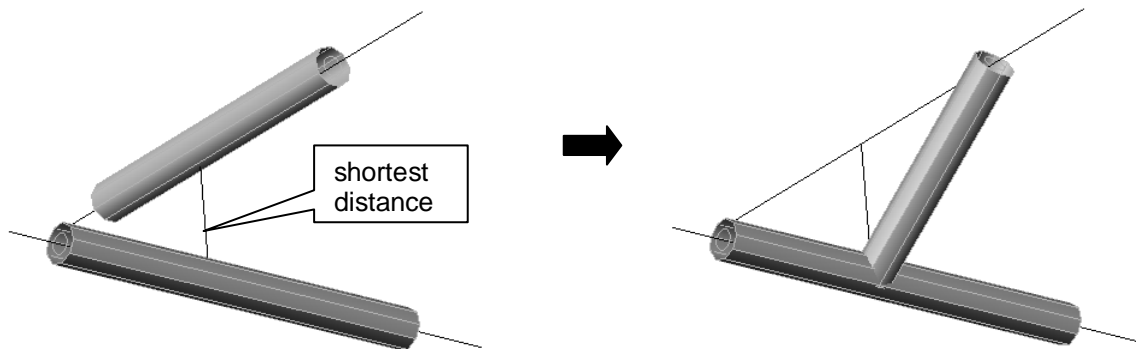|	Command:	KCCENTER
	Ribbon tab	As-Built TS Design

This command is used to move the cylinder end onto the centerline of another cylinder in such a way that it is attached in center.

> Command: **kccenter**
>
> Select reference cylinder: *(click on the cylinder that will be centered on.)*
>
> Select cylinder to be centered: *(click on the cylinder to be centered.)*

The shortest distance between both cylinders is calculated. The start point of the cylinder to be centered will then be moved to the point where the straight line of the shortest distance intersects the centerline of the reference cylinder.

This command can be used to create clean pipe connections. Due to the limited accuracy of measurement, the previously measured pipe sections will be in a slightly warped position. "KCCENTER" eliminates the warp and the cylinders will lie in one plane afterwards. This also means that the centerline measured previously will be changed in its position. Thus, it will not accurately represent the measured geometry. Therefore, this command should only be used to level out slight eccentricities. Extreme cases, as in the drawing above, should be avoided.

The difference to the command → **Cylinders - Bend cylinder** is that a tee connection is created here. "Bend cylinder" creates a „cylinder intersection".

Cylinders - Bend cylinder

	Command	KCBREAK
	Ribbon tab	As-Built TS Design

The command "KCBREAK" is used to bend a cylinder onto the centerline of another cylinder in such a way that it breaks through that cylinder in centre.

> Command: **kcbreak**
>
> Select reference cylinder: *(click on the cylinder that will be bent on)*
>
> Select cylinder to be bent: *(click on the cylinder to be bent.)*

The shortest distance between both cylinders is calculated. The cylinder is then bent onto the point where the straight line of the shortest distance intersects the centerline of the reference line. The bent cylinder consists of two cylinder sections.
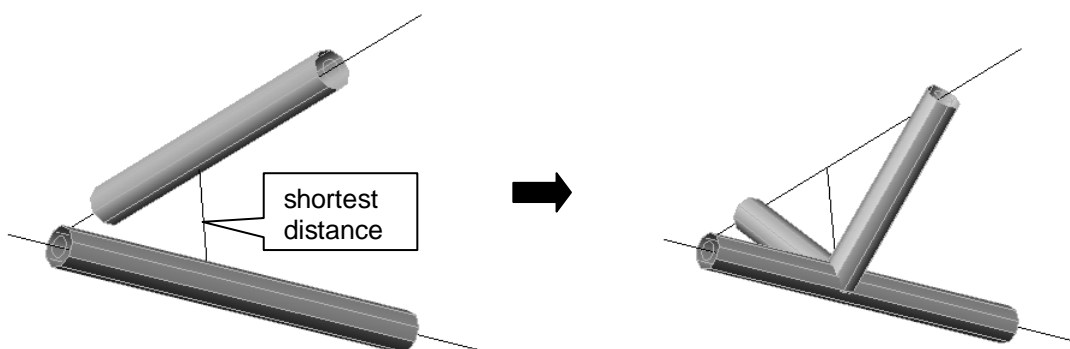
This command can be used to create neat pipe connections. Due to the limited accuracy of measurement, the previously measured cylinder sections will be in a slightly warped position to each other. This command eliminates the warp and the cylinders will lie in one plane afterwards. This also means that the centerline measured previously will be changed in its position. Thus, it will not represent the measured geometry exactly then. Therefore, this command should only be used to level out slight eccentricities. Extreme cases, as in the drawing above, should be avoided.

The difference to the command →**Center cylinder** is that command creates a tee connection, whereas "KCBREAK" creates a cylinder intersection.

Cylinders - Generate AutoCAD 3D solid

	Command	KCTOSOLID
	Ribbon tab	As-Built TS Design

The KCTOSOLID command allows you to convert kubit cylinders into AutoCAD solids.

Command: **KCTOSOLID**

Select cylinders:

Select objects: *(click on cylinder(s) to be converted)*

kubit cylinders are only construction aids. They are custom-designed objects with which you can only work in FARO software. If you want to forward the project to third parties, you have to convert the objects into a format that AutoCAD knows. Therefore, the last step of editing measured cylinders is to convert these into AutoCAD solids. The geometry of these cannot be changed. Thus, this last step should only be taken when the geometry is correct.

You can recognize the conversion when the cylinders do not appear "hollow" any longer but are solid items.

Hint: You can also use the AutoCAD command "explode" (origin) to convert cylinders into solids.

Caution! This command does NOT work under AutoCAD LT.

Cylinders - Generate cylinder axis

	Command:	KCTOAXIS
	Ribbon tab	As-Built TS Design

The "KCTOAXIS" command allows you to convert kubit cylinders into their centerlines and radiuses. These are then displayed as a single line and a circle arranged in centre. Optionally, you can delete the kubit cylinders during this conversion.

> Command: **KCTOAXIS**
>
> Create cylinder center lines and radiuses:
>
> Select cylinders:
>
> Select objects: *(click on cylinder(s) to be converted)*
>
> n Cylinders selected.
>
> Delete or retain cylinders? [Delete/Retain] <Retain>: *(choose option or [Enter] for Retain)*
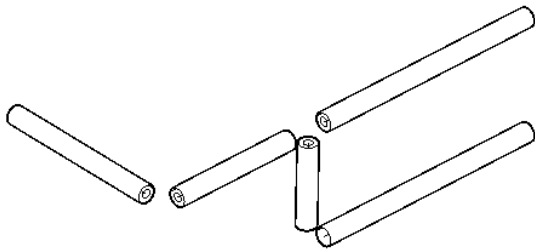>
> n Cylinders converted.

The centerlines and radiuses are created on the layer of the converted cylinder.

Cylinder – Create cylinder string

	Command	KCPIPELINE
	Ribbon tab	As-Built TS Design

With this command you may generate a 3D polyline from individual cylinders, which are part of one cylinder string. The polyline represents the axis of the cylinder string.

After calling up the command, you will be prompted to select a start cylinder. You should make sure, that all cylinders to be looked at, are visible on the screen. If this is not the case, position your drawing with pan and zoom so all cylinders are clearly visible on the screen. If it is already the case, you will select the start cylinder by clicking on it. The calculation of the string will start for both directions from this cylinder.

Command: **kcpipeline**
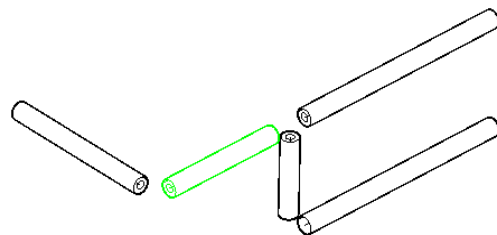
Number of cylinders: 5   *(Number of cylinders visible on the screen).*

Select first cylinder or [Options/Finish]:   *(Click on a cylinder).*
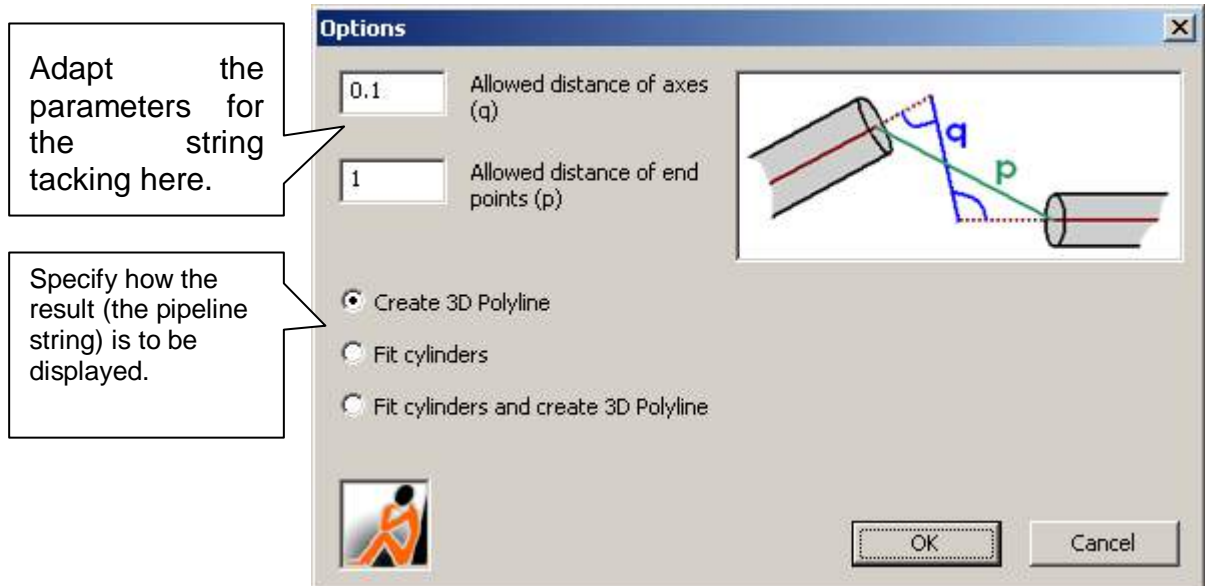
The cylinders belonging to the string will be marked in green. If the expected continuations are not found (as shown in the adjacent figure) you should check the settings for the string tracking.

Select first cylinder or [Options/Finish] <Options> **O**

(Select "O" or press Enter for "Options").

The following settings window will open:

The following table describes the meaning of the parameters for the string tracking:

allowed distance of axes (q)	States how big the distance of the axis lines of two cylinders may be, for them to be still recognized as adjacent.
allowed distance of end points (p)	States how far apart the axis end points of two cylinders may be for them to be still recognized as adjacent.

Adapt the parameters, so the expected string will be found. The parameters will be stored as long as AutoCAD is running.
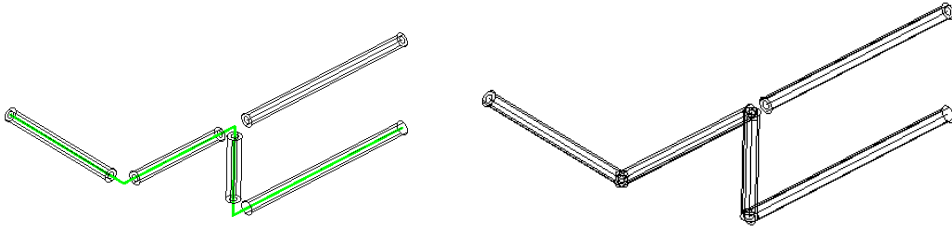
The figure below shows two found cylinder strings:
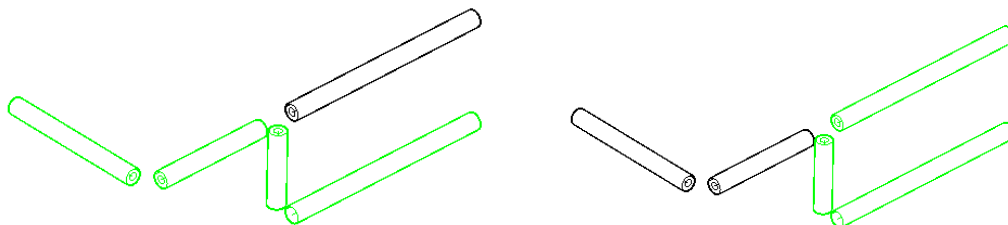
Select first cylinder or [Options/Finish] <Options> **F** *("F" for "Finish").*

Finished.

According the set option the following result will now be generated:

- **Create 3D polyline**: A 3D polyline will be generated, which represents the middle axis of the cylinder string. The cylinders remain unchanged.

- **Fit cylinder**: The cylinders will be adapted. Therefore the individual axes will be changed so the cylinders abut on each other.

- **Fit cylinder and create 3D polyline**: A 3D polyline will be generated, which represents the middle axis of the cylinder string. The cylinders will be adapted.

Cylinders - Insert block on cylinder

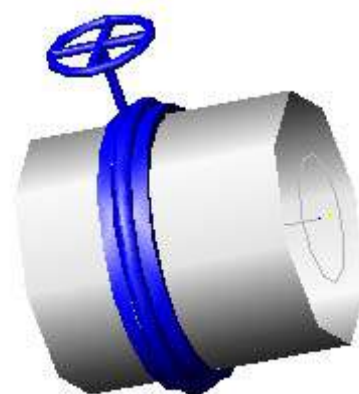|	Command	CYLINDERINSBLOCK
	Ribbon tab	As-Built TS Design

The command "CYLINDERINSBLOCK" allows you to position and insert a pre-defined block on a kubit cylinder. The cylinder centerline is placed and oriented by giving or measuring two to three points. The orientation is carried out via an insertion point and by giving the X and Y axes.

Following **example** illustrates the command sequence:

A three-dimensional body is to be placed on a cylinder. The body needs to be defined with a block before.
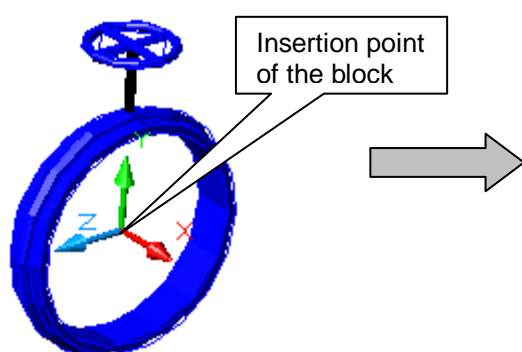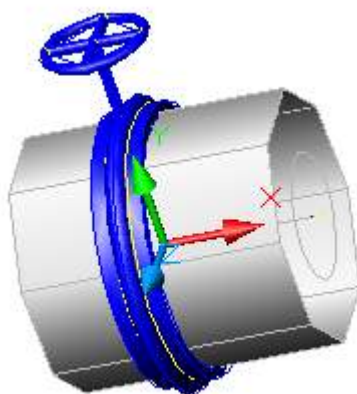

Block definition with position of the axes   *Situation: Pipe with block*

When defining the block, you have to take care of the correct position of the coordinate axes. These will be prompted when the block is inserted on the cylinder. The X axis always runs along the cylinder centerline and the Y axis and Z axis point perpendicularly out of the cylinder.


Upon calling up the command, a dialogue for the selection of a block template opens. Select the desired block. If the block is not yet defined in the drawing, you can enter a drawing file as block template.

You will then be asked to select the cylinder on which the block is to be placed.

Select cylinder: *(click on the cylinder where the block should be inserted)*

Measure insertion point position: *(measure point on cylinder surface to define the insertion point)*

The point will then be plumbed onto the centerline of the cylinder. The plumb point is the insertion point of the block. In a separate block definition file this point is the (coordinate) origin.

Measure a point that - plumbed onto the centerline - defines the insertion point of the block

Plumb point on centerline = insertion point of block
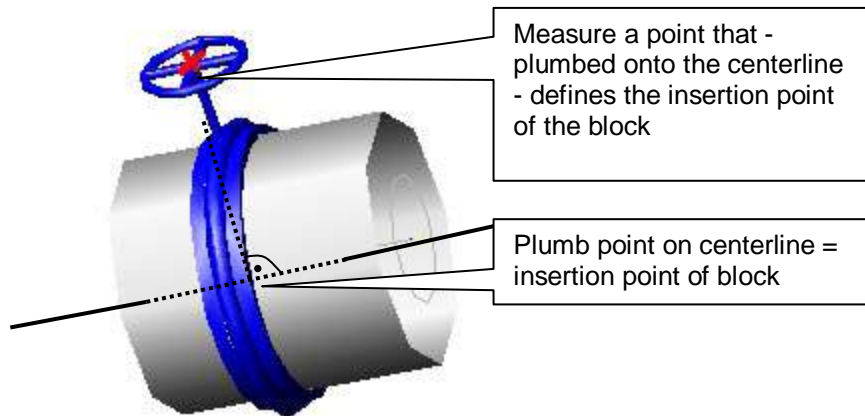
Next, the direction of the y-axis of the block will be prompted. That corresponds to one rotation of the block around the centerline. You will thus be asked for another point. In this case, the previously measured insertion point corresponds to the direction of the Y axis of the block. That's why the option "Insertion point is y-axis" is chosen.

Measure point for y-axis of the block or [Insertion point is y-axis]<Insertion

point>:**I** *(insertion point lies on the Y axis of the block)*

By requesting another point the block can now even be rotated around the Y axis. Thus the X axis of the block is panned off the centerline. However, such a rotation does not make sense in this example since due to the definition of the block the centerline of the cylinder and the X axis of the block are identical. Thus, the option "Center line is x-axis" is chosen.

Measure point for x-axis of the block or [Center line is x-axis]<Center line>:**c**

(center line is identical with X axis of the block)

In this case, the orientation of the block needs to be defined by clicking one side of the cylinder.
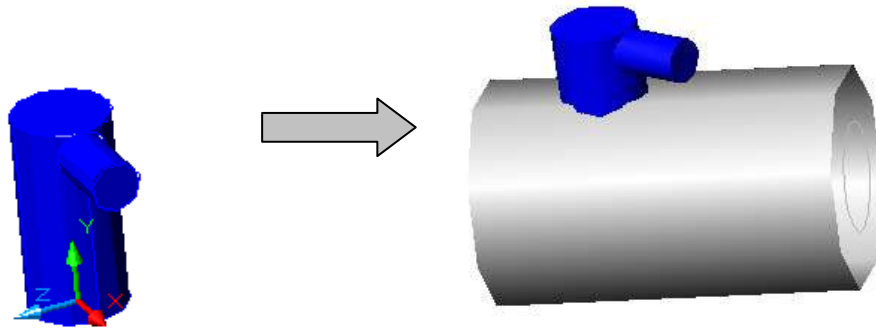
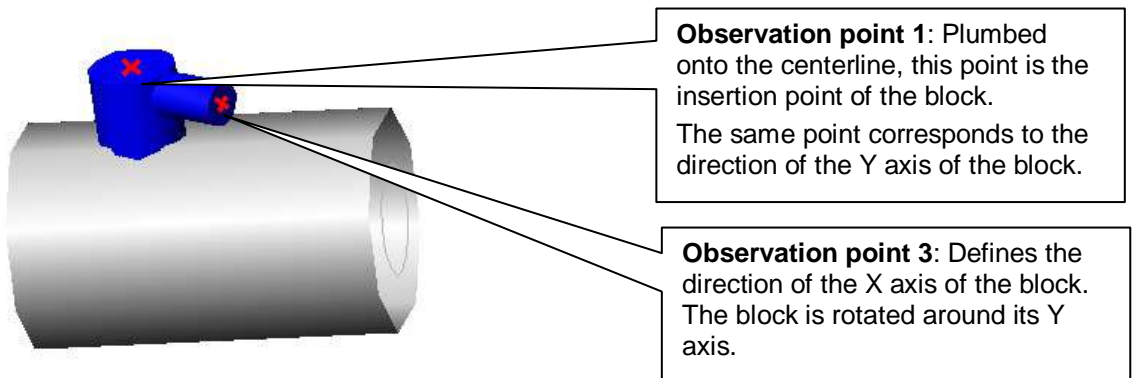Enter side for the orientation of the x-axis: *(click on one end of the cylinder)*

With these entries the block is well-defined in its position.

Another example:

This block template contains an object that is only partially symmetric

Use following observation points and options of the command for the insertion of the block on the cylinder.

Observation point 1: Plumbed onto the centerline, this point is the insertion point of the block.

The same point corresponds to the direction of the Y axis of the block.

Observation point 3: Defines the direction of the X axis of the block. The block is rotated around its Y axis.

Command: **CYLINDERINSBLOCK**

Select cylinder: *(click on the cylinder where the block should be inserted)*

Measure insertion point position: *(measure point on cylinder surface to define the insertion point)*

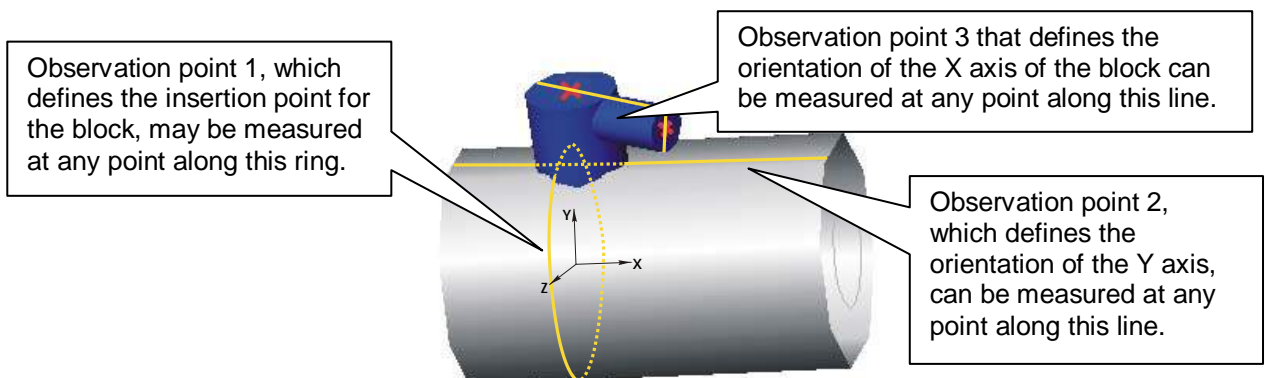Measure point for y-axis of the block or [Insertion point is y-axis] <Insertion

point>: **I** *(insertion point lies on the Y axis of the block)*

Measure point for x-axis of the block or [Center line is x-axis] <Center line>:

(measure point 3 to define the direction of X axis of the block)

The block is thus positioned.

In some cases the individual observation points may be hard to measure due to bad visibility. Therefore, here are some more examples of measuring possibilities.

Observation point 1, which defines the insertion point for the block, may be measured at any point along this ring.

Observation point 3 that defines the orientation of the X axis of the block can be measured at any point along this line.

Observation point 2, which defines the orientation of the Y axis, can be measured at any point along this line.

Remark: The examples for block templates depicted here can be found in the subdirectory \DWGBASE of your As-Built for AutoCAD installation under the entries "sleeve.dwg" (first example) and "nozzle.dwg" (second example). The examples are adjusted for cylinders with a radius of 0.25 m.

Hint: You can use the AutoCAD command "ALIGN" to insert a block on the cylinder. Here, the block is positioned and aligned at three arbitrary points in the room. For some geometry "ALIGN" is more suitable than the command described here.

Caution! This command does NOT work under AutoCAD LT.

Cylinders - Insert cylinder reducer

	Command:	KCREDUCE
	Ribbon tab	As-Built TS Design

With the command "KCREDUCE" you can connect two kubit cylinders by a cylinder reducer. In order to do so both cylinders have to be **coaxial**.

The cylinder reducer is an AutoCAD solid the geometry of which cannot be edited subsequently. For the same reason it can not be edited with other cylinder commands (e.g. "Adjust cylinder length").

When running the command you are first prompted for the selection of the cylinders. Click the cylinders on those ends to be connected:

command: **kcreduce**

Select first cylinder: *(Click first cylinder on the end on which to be connected)*

Select second cylinder: *(Click second cylinder on the end on which to be connected)*

If both selected cylinders lie on two different layers you will be alerted automatically and the cylinder reducer will be stored on the same layer as the first cylinder.

Caution! This command does NOT work under AutoCAD LT.

Hexahedron - Measure hexahedron

	Command	MEASHEX
	Ribbon tab	As-Built TS Design

The command "MEASHEX" allows you to measure a hexahedron. For this, at least five points need to be entered: three points on one side and two on a neighboring side. A **cuboid**, i.e. an orthogonal hexahedron is trued into these points:
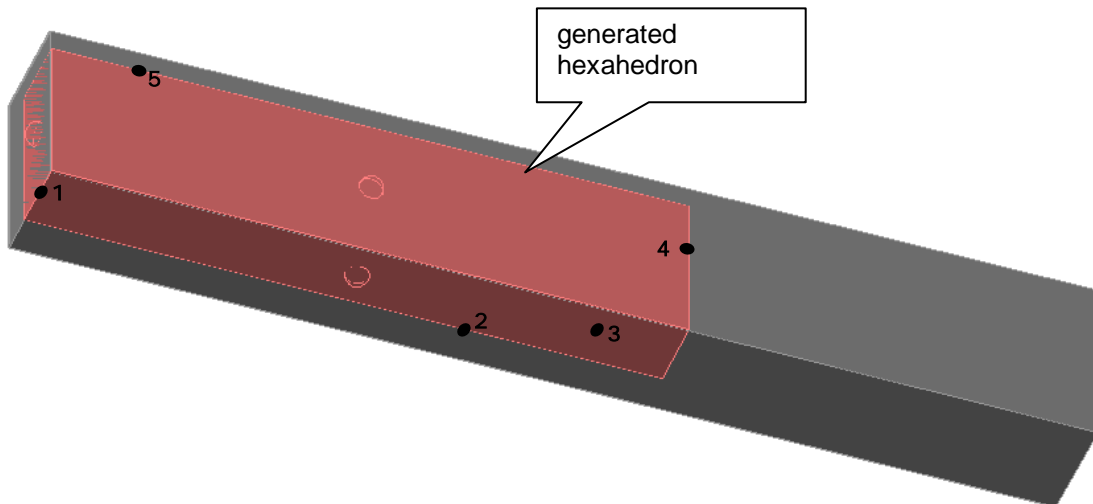
Command: **meashex**

Measure an hexahedron using two neighboring areas.

1. areas 1st point:	*(measure or click point 1)*
1. areas 2nd point:	*(measure or click point 2)*
1. areas 3rd point:	*(measure or click point 3)*
2. areas 1st point:	*(measure or click point 4)*
2. areas 2nd point:	*(measure or click point 5)*

The first three points define one side of the hexahedron. Another side is attached perpendicular to the first side. This second side is determined by the two last points. The remaining sides of the hexahedron can be derived from that since they are all perpendicular or parallel to the two measured sides. The dimensions (height, width, depth) of the hexahedron can also be derived from the measuring points. These are only preliminary dimensions. They will be adapted afterwards to work out the real shape of the object.

The sides on which the measured points lie have the colour of the active layer. Since the other sides only have temporary character, they are highlighted in red. These temporary sides can be adjusted later. They will then lose their temporary status and change into the respective layer colour. There are different ways of adjusting the sides:

- You can drag the side to the true position by using the hemispheric grips and your mouse pointer or by measuring with a total station.

- By using one of the "Stretch hexahedron" commands, the length of a hexahedron can be adjusted in one direction.

- The command ➔**Hexahedron - Edit hexahedron** allows you to align hexahedron sides to planes.

**General information about kubit hexahedrons**

A hexahedron is composed of six sides. I.e. a hexahedron does not have to have regular shapes (like a cuboid), but the position of the sides in space can be chosen arbitrarily. Therefore any irregular shapes can be composed of hexahedrons.

However, the creation of a hexahedron always starts with a regular shape – a cuboid. The individual sides are adjusted later by aligning them with planes and other hexahedrons.

There are several ways of creating a hexahedron. You can either freely position it in the room or align it perpendicular or parallel to the user coordinate system (UCS).

A As-Built for AutoCAD-hexahedron has following properties:

- Every corner and side feature a "grip". The sides are characterized by two intertwining circles ("ball") in the middle of which the grips are positioned.

- You can easily change the length of a hexahedron by "dragging" the side grips with the mouse into the respective direction. Dragging the corners results in the displacement of the whole body in the room. The angle relations of the sides, however, can only be altered with the command ➔**Hexahedron - Edit hexahedron**.

- As with all As-Built for AutoCAD commands, the manipulation can also be carried out by measurement. Click the grip with the mouse and define the new position with the total station.

- The sides carry additional information. A side has the color of the current layer if it has already been defined by measurement. Undefined sides are highlighted in red. Thus you can keep track when some parameters of the hexahedron only become visible from a different station later.
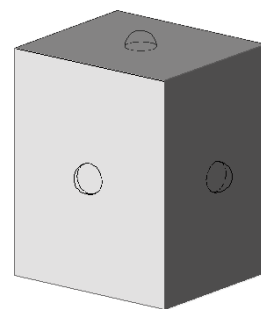
Further commands allow you to alter the length of a hexahedron, to copy the dimensions of a hexahedron or to subtract a hexahedron from a solid.

Hexahedron - Measure hexahedron parallel to UCS

	Command	MEASHEXPAR
	Ribbon tab	As-Built TS Design

The command "MEASHEXPAR" allows you to measure a cuboid hexahedron that is aligned **parallel to the current UCS**. That means that all vertical edges of the hexahedron run parallel to the Z axis of the UCS and that the top and base areas are parallel to the XY plane of the UCS.

In order to define a hexahedron, three points are required. The first three points define the direction and length of the hexahedron in the XY plane. The third point defines the thickness and height of the hexahedron.
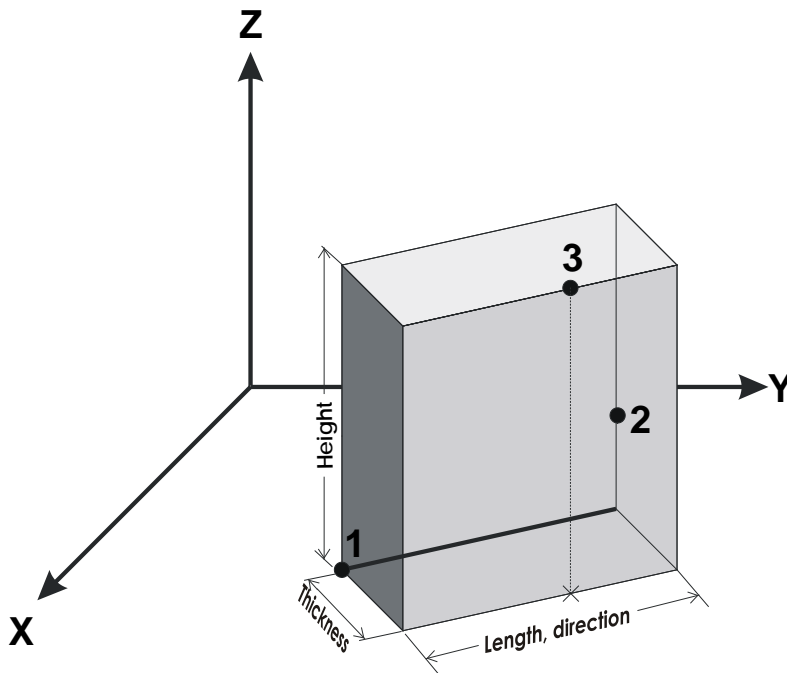
Command: **meashexpar**

Measure an hexahedron parallel to to UCS-XY-Plane using two neighboring areas.

1st area (perpendicular to UCS-XY-Plane) 1st point: *(measure or click point 1)*

1st area (perpendicular to UCS-XY-Plane) 2nd point: *(measure or click point 2)*

2nd area point: *(measure or click point 3)*

If the third point does not lie lengthwise between the first two points, the hexahedron length will be altered again and be dragged to that point.

The command is suited to create bodies that have a vertical orientation, e.g. columns. Under this condition you can save measuring points and thus time. Instead of five, you only have to measure three points.

Hexahedron - Measure hexahedron perpendicular to UCS

	Command	MEASHEXPERP
	Ribbon tab	As-Built TS Design

The command "MEASHEXPERP" allows you to measure a cuboid hexahedron that has **two perpendicular sides to the UCS's XY plane**. I.e. the cuboid lies tilted on the XY plane in one direction.

Four points are required to define the hexahedron. The first two points define the orientation of a plane perpendicular to the UCS's XY plane. Two further points are measured on a neighboring plane of the cuboid. This second plane of the cuboid is defined by the measured points three and four and by the fact that it is perpendicular to the first plane of the cuboid. The dimensions of the cuboid result from the position of the measured points.
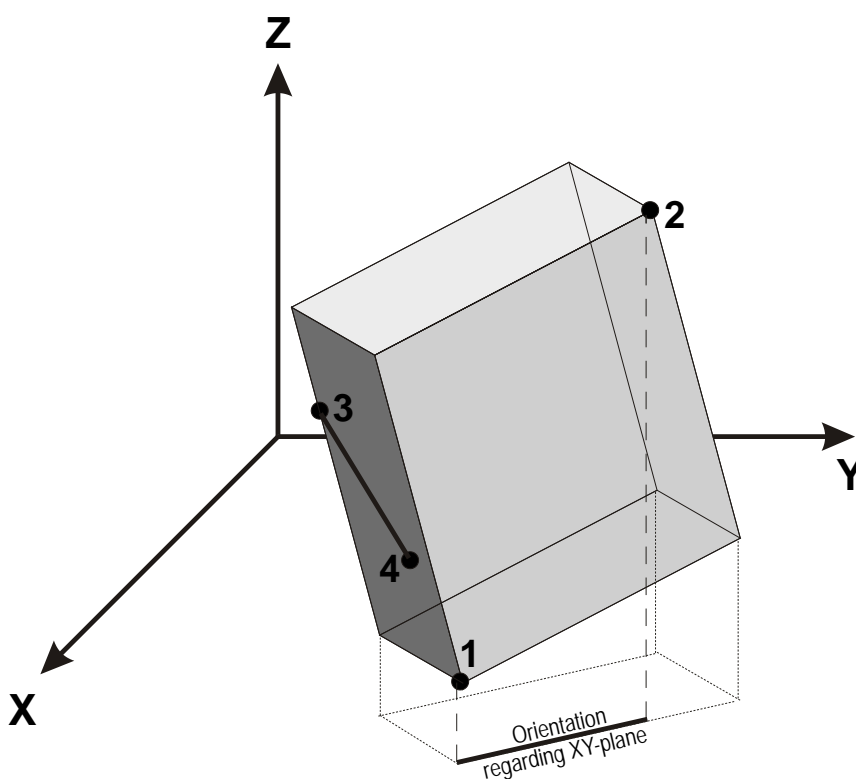
Command: **meashexperp**

Measure an hexahedron perpendicular to UCS-XY-Plane using two neighboring areas.

1st area (perpendicular to UCS-XY-Plane) 1st point:	*(measure or click point 1)*
1st area (perpendicular to UCS-XY-Plane) 2nd point:	*(measure or click point 2)*
2nd area 1st point:	*(measure or click point 3)*
2nd area 2nd point:	*(measure or click point 4)*

This command is suitable for measuring objects that have two vertical sides but lie tilted in the room. Typical examples are roof beams or cross beams.

Hexahedron - Stretch hexahedron by value

	Command	STRETCHHEXVAL
	Ribbon tab	As-Built TS Design

The command "STRETCHHEXVAL" changes the length of an existing hexahedron by a certain value.
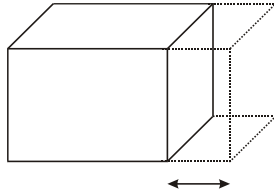
You have to enter the side to be changed and the value of extension:

Command: **stretchhexval**

Select hexahedron area to be moved: *(click on the side surface of hexaedron)*

Stretch by which value: (*Enter value or click/measure 2 points n*)

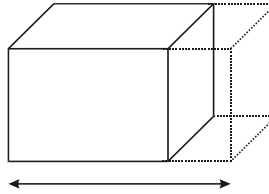You can enter the value either by entering a number or by measuring two points.

If you want to change the length of a hexahedron without entering a value, use the AutoCAD-"grips". Small blue boxes will appear on the edges and sides of the hexahedron. You can grab the hexahedron at the grips on the sides with the mouse and drag it to the desired length.

If you, however, only know the total length of the hexahedron, use the command ➔**Hexahedron - Stretch hexahedron to length**.

Hexahedron - Stretch hexahedron to length

	Command	STRETCHHEXLENGTH
	Ribbon tab	As-Built TS Design

The command "STRETCHHEXLENGTH" allows you to stretch the length of an existing hexahedron to a certain total length.

You have to define the side to be stretched and a value for the total length:

Command: **stretchhexlength**

Select hexahedron area to be moved: *(click on the side surface of hexaedron)*

Stretch to which total length?: **2.13**   (*Enter value or click/measure 2 points*)

You can enter the total length either as a number or by measuring two points.

If you want to change the value of a hexahedron without entering a value, use the "grips".To do so mark the hexahedron. Small blue squares appear on the corners and the side faces. You can simply use the mouse to click on the grips and drag the hexahedron to the desired length.

If you, however, want to stretch the hexahedron by a certain value, use the command ➔**Hexahedron - Stretch hexahedron by value**

Hexahedron - Edit hexahedron

	Command	EDITHEX
	Ribbon tab	As-Built TS Design

The command "EDITHEX" allows you to align the side surfaces of the hexahedron with other planes in the space. Thus, only this command creates real hexahedrons from the object that always has the shape of a cuboid after its generation.

Hexahedron before editing

Adjustment plane

Hexahedron after editing
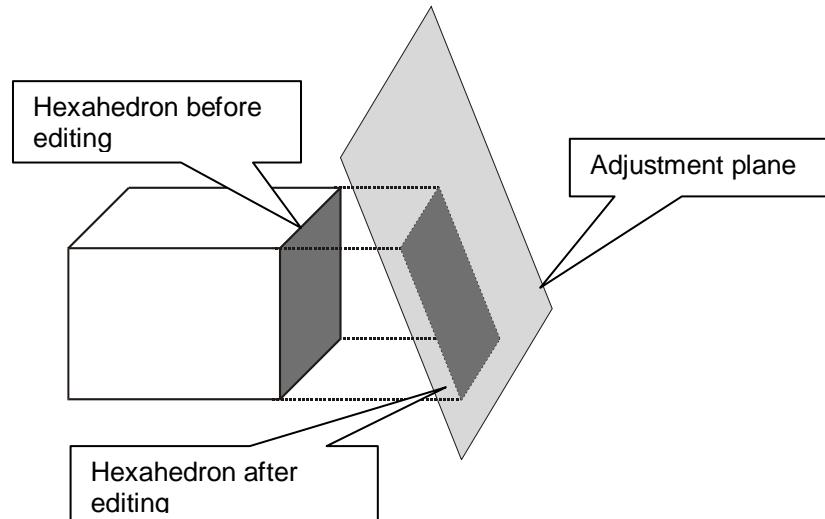
Command sequence:

Command: **edithex**

Select hexahedron area to adjust to an area: *(click on the side surface to be adjusted)*

Select plane or other hexahedron area: *(click on the adjustment plane)*

This command allows you to adjust the temporary cuboid shape of the hexahedron to the real object geometry. This is done separately for each side surface by stretching it to an already defined plane (adjustment plane). There are two ways of defining the adjustment plane:

- Measure it as construction plane. Use the commands ➔**Design – Draw plane** or **Design – Fit plane**. The plane is determined by several points in the 3D space.

- You have or create a hexahedron of which one side represents the adjustment plane.

By taking several steps you are thus quite free to shape a hexahedron. After a side surface has been adjusted, its status automatically changes from "unprocessed" to "processed" and the colour changes from red to the colour of the layer.

The adjustment plane has an indefinite extension. Therefore, it does not matter whether the represented part of the plane (triangle for construction plane or square for hexahedron side) would really intersect with the hexahedron or not.

Not all adjustment planes are suitable for every hexahedron surface. In some cases the configuration might not make sense due to an unfavorable position of the planes. Then following error message appears:

Unable to run operation since it leads to an inconsistent hexahedron.

For example, this is the case when the adjustment plane is perpendicular to the hexahedron surface.

It is also not possible to align two neighboring hexahedron surfaces with the same plane. Two side surfaces would then combine to one surface and the hexahedron would no longer have six sides.

Hexahedron - Copy dimensions

	Command	COPYHEXDIM
	Ribbon tab	As-Built TS Design

The command "COPYHEXDIM" is used to transfer the dimensions of one hexahedron to another one. Enter one to three sides of the "model" hexahedron and the respective sides of the hexahedron to be adjusted.

Command: **copyhexdim**

Select areas of the source and target hexahedron

Select hexahedron areas: *(click first cuboid surface of source hexaedron)*

Select hexahedron areas: *(click second cuboid surface of source hexaedron)*

Select hexahedron areas: *(click third cuboid surface of source hexaedron)*

Select hexahedron areas: *(click first cuboid surface of target hexaedron)*

Select areas on target hexahedron: *(click second cuboid surface of target hexaedron)*

Select areas on target hexahedron: *(click third cuboid surface of target hexaedron)*

If you only want to copy one dimension, there is a shorter command sequence:

Select areas of the source and target hexahedron

Select hexahedron areas: *(click cuboid surface of source hexaedron)*

Select hexahedron areas: *(click cuboid surface of target hexaedron)*

The dimension of the source hexahedron that is to be copied results from the distance of the selected cuboid surface to its opposite surface. Therefore, this command only works when the selected hexahedron side is parallel to its opposite surface. Otherwise an error message appears.

selected side on source hexahedron

selected side on target hexahedron

1

2

Source hexahedron

Target hexahedron

dimension to be copied
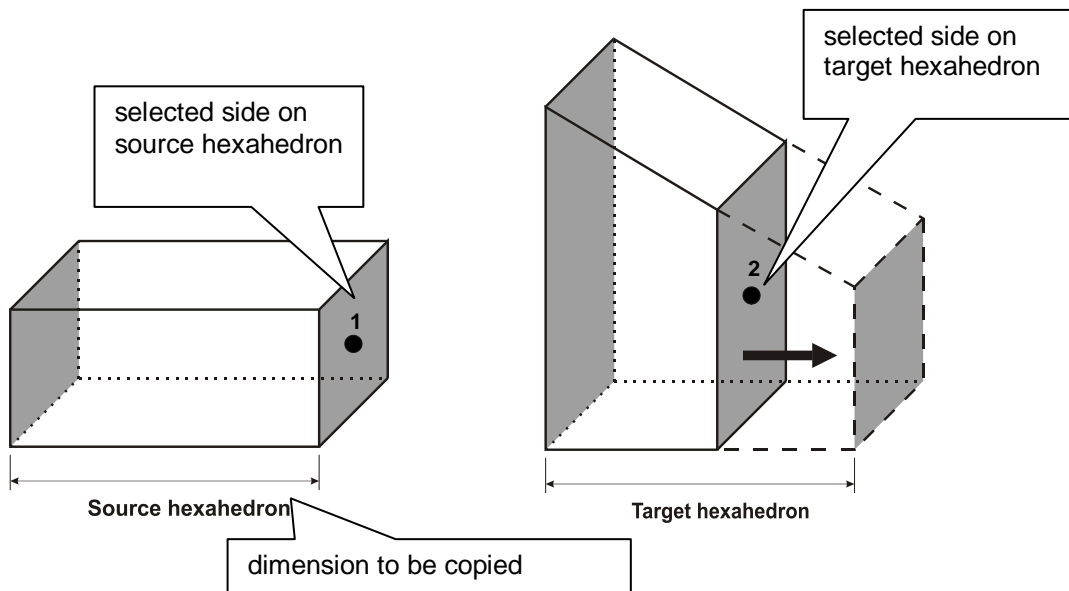
Target and source hexahedron, however, do not have to be parallel. You are also free to decide which source surface is to correspond to which target surface.

Hexahedron - Set area markers

	Command	HEXMARK
	Ribbon tab	As-Built TS Design

The sides of a hexahedron carry additional information. If the side of a hexahedron has already been provided with additional information, it has the colour of the current layer. Undefined sides with temporary character are highlighted in red. Thus, you can keep track if some parameters of the hexahedron are only visible from a different station later.

If a side is edited with an editing command for the first time, it will automatically change from status "temporary" to status "defined".

This command allows you to mark a side as "measured" without altering it. After calling up the command, just click on the respective side:

Command: **HEXMARK**

Select hexahedron area to be shifted: *(click on the side surface of hexahedron)*

Vice versa, a side that has already been defined by As-Built for AutoCAD, can also be assigned the status "temporary".

Hexahedron - Subtract hexahedron from solid

	Command	SUBHEXSOLID
	Ribbon tab	As-Built TS Design

The command "SUBHEXSOLID" is used to subtract a As-Built for AutoCAD hexahedron from an AutoCAD solid.

> Command: **subhexsolid**
>
> Select solid:
>
> Select hexahedron:

The section in which hexahedron and solid intersect is cut out of the solid, and the hexahedron is deleted. In order to apply this command, hexahedron and AutoCAD solid need to intersect at least in part.

This command is especially suitable to include specially shaped openings, recesses or breakthroughs in AutoCAD solids. You can, for example, measure a window opening as hexahedron and then subtract it from the wall solid. .

However, this command is not suitable to subtract two hexahedrons from each other. The resulting body could have more than six sides and thus would no longer be a hexahedron.

Caution! This command does NOT work unter AutoCAD LT.

Hexahedron - Convert hexahedron into solid

	Command	CONVHEXSOLID
	Ribbon tab	As-Built TS Design

The command "HEXSOLID " is used to convert a As-Built for AutoCAD hexahedron into an AutoCAD solid.

Command: **convhexsolid**

Select objects: *(click on hexahedron to be converted, confirm with Enter)*

The As-Built for AutoCAD hexahedrons are only construction aids. They are custom-designed elements with which you can only work in As-Built for AutoCAD. If you want to forward the project to third parties, you have to convert the elements into a format that AutoCAD knows. Thus, the last step when editing the measured hexahedrons is converting them into AutoCAD solids. Since the geometry of these solids cannot be changed later, this step should only be taken when the geometry is correct.

You can recognize the conversion by the missing grips on the sides of the solids.

Hint: You can also use the AutoCAD command "explode" (ORIGIN) to convert hexahedrons into solids.

Caution! This command does NOT work under AutoCAD LT.

Hexahedron - Create prism

	Command	PRISM
	Ribbon tab	As-Built TS Design

The command "PRISM" allows you to create a prism. The prism is an AutoCAD solid that is defined by an arbitrarily shaped base area and a height.

This command works similar to the AutoCAD command "EXTRUSION": You have to specify a closed line that defines the shape of the base and top area. You then have to enter the height of the body by selecting a point on both the base and top area.

Command: **prism**

Select objects for prism cross section. (circles, lines, splines, ellipses, arches):

(click all objects for girthed area of the prism)

Select objects: 8 found

Point on base area: *(click point on base surface of prism)*

Point on upper area: *(click point on top surface of prism)*

Prisms are especially suitable to model walls, columns and other irregularly shaped or bent objects with a vertical extension.

Following AutoCAD line objects can be used to define the shape of the base area:

- Line
- Circle
- Spline
- Ellipsis
- Arch

The line can also lie in a 3D spaceor be composed of different line objects. In order to define the base area, the line is projected into the XY plane of the current UCS. However, one condition is that the line is a closed line that does not intersect itself:
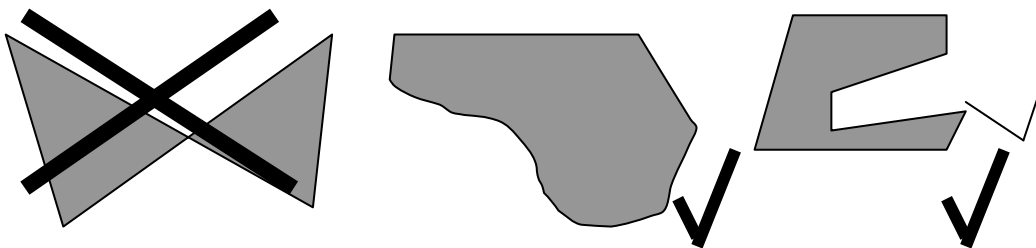
The command does not work with AutoCAD polylines and 3D polylines. In this case you should rather use the AutoCAD command "EXTRUSION".

Caution! This command does NOT work unter AutoCAD LT.

Intermeshing – Intermeshing of points

	Command	MESHPOINTS
	Ribbon tab	As-Built TS Design

A net of triangular surfaces will be placed through a group of AutoCAD points by this command. An AutoCAD poly surface mesh (PNET) will be built.

This command is suitable for modelling irregular surfaces like arches, natural stone walls or even the ground level.

command: **meshpoints**

Select points for meshing: *(choose the AutoCAD points to be intermeshed)*

113 found.

Select objects: *(Confirm selection by pressing "Return")*

Define initial triangle, point 1 <automatic>: *(specify first point of the initial triangle)*

Point 2 [Undo]: *(specify second point of the initial triangle)*

Point 3 [Undo]: *(specify third point of the initial triangle)*

Enter maximal length of edges or <automatic>: *(enter value or two points*
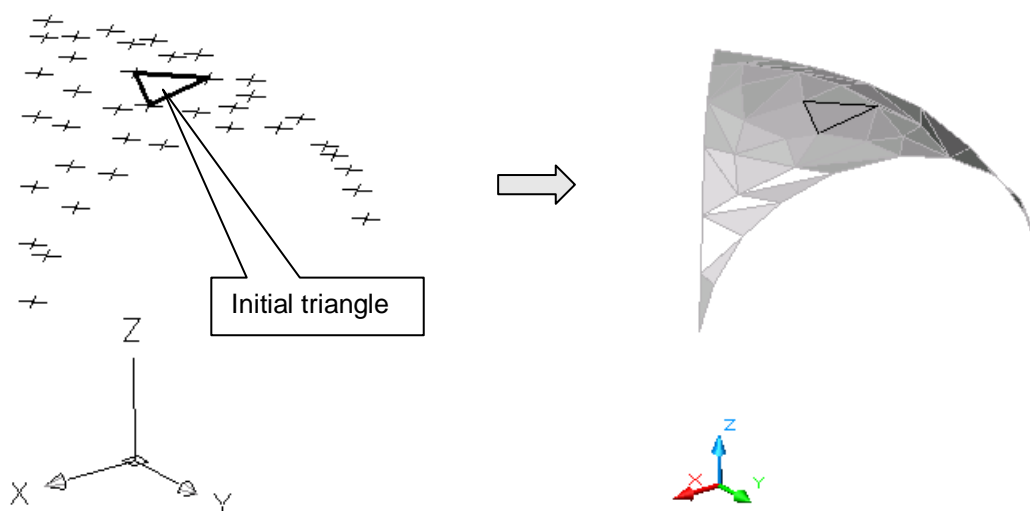
for maximum edge length of triangle meshes)

Point 113 of 113

Should the selected points be deleted? [Yes/No] <No>: *(enter "Y"(yes) or "N"(no))*

A special algorithm tries to find a triangle configuration, which shows the surface in the best possible way, assisted by the statement of the **initial triangle**. For the initial triangle just click three neighboring points on the surface which shall build a triangle within the finished net. The statement of the initial triangle is optional. As-Built for AutoCAD starts at any place to intermesh to points when you select the "automatic" option.

The intermesh result will also be affected by the statement of the **maximum edge length of the triangles**. This statement is optional as well. You may either enter a value or click two points of which the spatial distance will be used. It is advisable to state two neighboring intermeshing points, which have – regarding the complete network – a fairly big distance to each other.

You should observe the following advices in order to get a satisfying intermeshing result:

- The more the surface is curved, the higher the density of the measured AutoCAD points has to be.

- The density of the AutoCAD points should be as evenly as possible. The result of the intermeshing will be degraded by "holes" within the point grid or double measured points.

- The curve of the surface has to be continuous. Breaking edges cannot be defined. So, if the surface bends around a "sharp edge", every side has to be intermeshed separately.

- Be sure to measure enough points within the boundary area of the surface. If you measured too less points, the boundary of the poly surface mesh will look frayed.

Intermeshing – Create intersection lines

	Command	PNETCONTOURLINE
	Ribbon tab	As-Built TS Design

When using this command, an existing poly surface mesh (PNET) will be cut with a construction plane and generate one or several intersection lines.

At first you have to define a construction plane which shall serve as cutting plane for the net.

Befehl: **pnetcontourline**

Select section plane: *(Select* construction plane *which shall serve as cutting layer)*

Number of intersections <1> or [Undo]: **10** *(Number of profile lines to be generated)*
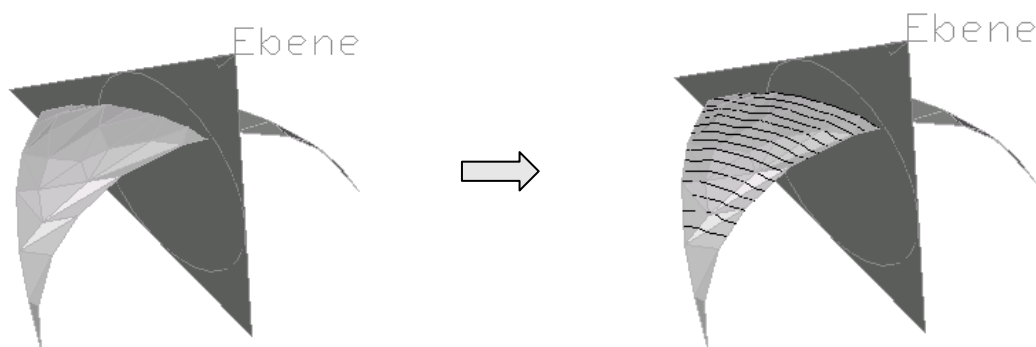
Distance between intersections <1.000000> or [Undo]: **0.05**

(Interval between the profile lines)

Point on that side on which the sections are intended [Undo]:

(Click on a point of that side of the plane, where the profile lines are to be generated)

Please select polyface mesh(es):or [Undo/Finish]:

(Select one or several poly surface meshs, which are to be cut)

The emerging profile lines will be inserted into the current layer of the drawing as 2D polylines.

With this command you will be able to establish cuts oriented in any direction, which are situated on complicatedly formed surfaces. Please use this command together with the As-Built for AutoCAD command ➜**Intermeshing – Intermeshing of points**.

This command is also suitable to generate contour lines. To shape the ground level you also use the ➜**Intermeshing – Intermeshing of points** command.

Hint: If you want to generate just one cutting line of a net with a plane, you may alternatively use the ➜**Design – Intersect plane – Intersection with Lines, Planes, 3D-Solids, PNets** command.

Intermeshing – Convert mesh into points

	Command	PNETTOPOINTS
	Ribbon tab	As-Built TS Design

With this command you can convert an AutoCAD poly surface mesh (PNET) into AutoCAD points. For every node of the mesh an AutoCAD point will be inserted.

By using the command ➔**Intermeshing – Intermeshing of points**, you can create poly surface meshes. This command calculates a triangular intermeshing from a group of AutoCAD points and generates a corresponding poly surface mesh. You have the option to delete the original points. If you have chosen this option and later on you want to execute the intermeshing again, you will first have to regenerate AutoCAD points from the poly surface mesh. The following is the command for that operation.

After you have called up the command the following is prompted in the command line:

Select polyface mesh: *(Click the poly surface mesh to be transformed)*

Confirm your selection by pressing Return. You will now be prompted:

Do you want to delete the selected polyface mesh? [Yes/No] <No>:

Depending if you choose "Yes" or "No", the poly surface mesh will be deleted or maintained. Confirm your selection by pressing Return. Every interpolation point will be changed into an AutoCAD point.

Intermeshing – Edit intermeshing (Pnet)

	Command	PNETEDIT
	Ribbon tab	As-Built TS Design

You use this command to edit an AutoCAD poly surface mesh (PNET), which has been built by the command ➜**Intermeshing – Intermeshing of points**. Sometimes, in case of a bad line-up of points, there will be unwanted effects at the boundary of the intermeshing. The boundary of the mesh seems to be frayed or there are inappropriate triangles. With this command it is possible to improve the intermeshing manually. You may add new points to the intermeshing, delete or add existing edges.

When calling up this command, you will first be prompted to select the poly surface mesh to be edited:

> Befehl: **pnetedit**
> Select polyface mesh: *(select mesh)*

Now the mesh will be loaded into the memory. If it is a very big mesh this may take several minutes. After that you select the option that is to be executed for the mesh.

> Area 649 of 649 areas read.
>
> First edge of the triangel to be closed or [add Point/Delete edge/Smooth boundary/Finish]
> <Delete edge>:

You can choose between "add point", "add edge", delete edge" and "smooth boundary". Be aware that the option "add edge" will always be active when starting the command. Once you have chosen the command and selected the mesh, you will be able to execute several operations on the mesh without having to exit the command. Commands that will lead to a self penetration of the mesh will not be accepted.

The single options are explained in the following.

Add edge

This option enables you to connect two existing edges of the intermeshing with another edge. By doing so a new triangle is created. You have to select two neighboring edges which are to be closed to a triangle.

> First edge of the triangel to be closed or [add Point/Delete edge/Smooth
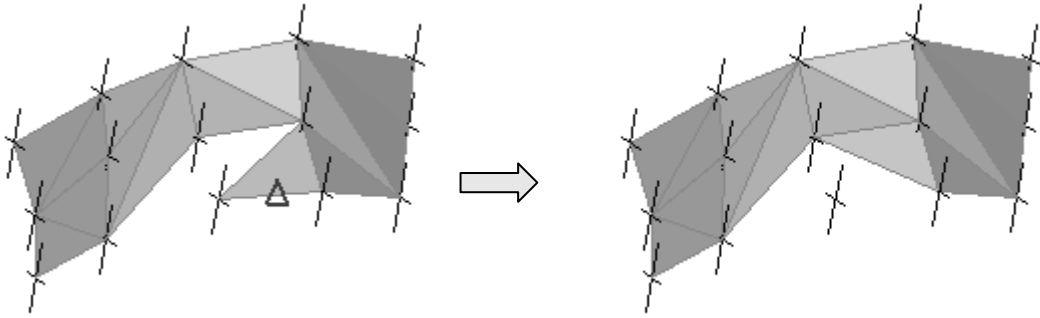> boundary/Finish] <Delete edge>: *(Select first edge)*
> Second edge of the triangle to be closed [Undo]: *(Select second edge)*

When selecting the edges, the snapping function will automatically be on "midpoint". Click the middle of the accordant edges.

In case of this message

> There is another edge between the two defined edges.

no triangle will be generated. Most of the times this happens, when two net points are situated very close to each other (point has been measured twice). Then, there has been generated a very small triangle during the intermeshing, which is situated between the two selected edges. In this case delete one of the two points and intermesh the remaining points again. Then in most cases you will get a better result.

Delete edge

You will delete a boundary edge of the mesh with this operation. The according triangle will be deleted automatically.

Edge to be deleted or [add Point/add Edge/Smooth boundary/Finish] <add Edge>:

(select the border edge to be deleted)
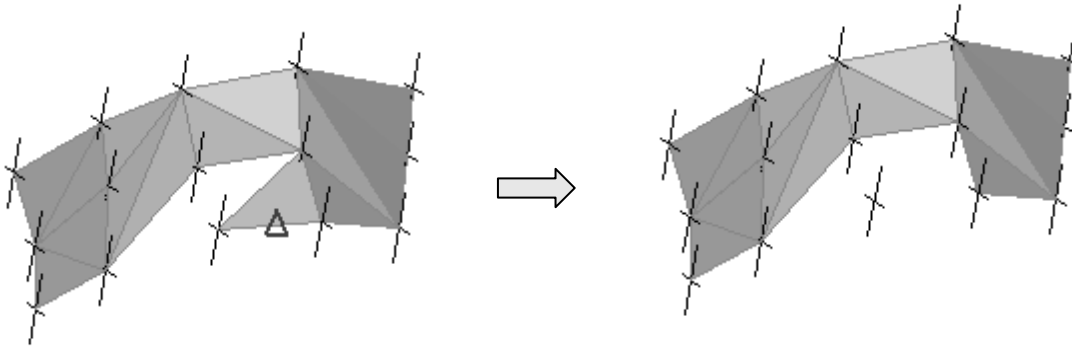
Add point

This option enables you to add a new point to the mesh. By doing so a new triangle will be created. You will have to state the new point as well as the boundary edge of the mesh to which the triangle is to be connected.

New point or [add Edge/Delete edge/Smooth rim/Undo/Finish] <add Edge>:

(Select point to attach)

Select edge on the rim of the mesh for attaching [Undo]:   *(Edge to build a triangle with point)*
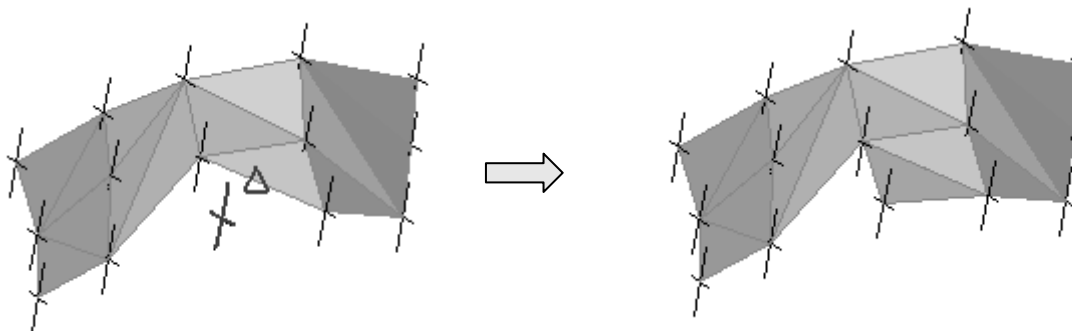
Smooth boundary

This option enables you to fill larger gaps of a net with triangles in one step. Therefore you have to state two mesh points on the boundary of the gap and a third mesh point in the very inner of the gap. Select the third point where you would also start if you were to place the first triangle manually. Usually this will be the "lowest point" of the gap.

Define first point for contraction or [add Point/add Edge/Delete
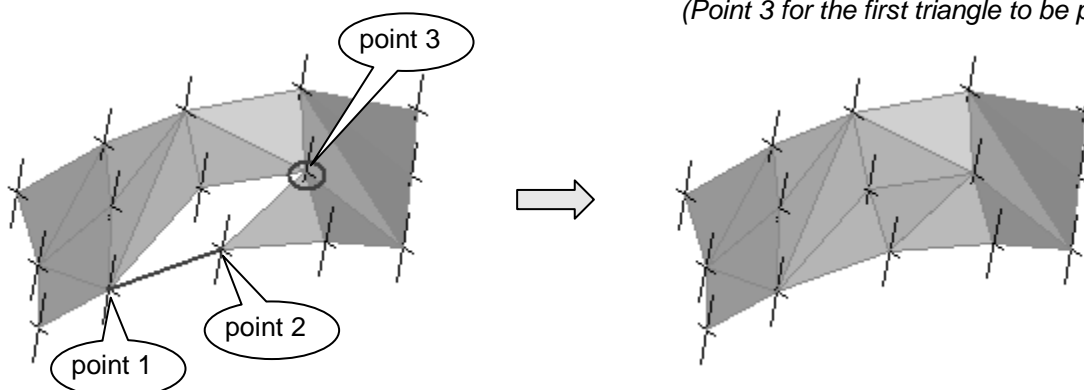
edge/Undo/Finish] <Add edge>:   *(Point 1, to which new triangles are to be attached)*

Define second point for contraction [Undo]:

(Point 2, to which new triangles are to be attached)

Define further point on the rim at the tip of the outlier [Undo]:

(Point 3 for the first triangle to be placed)

If this command should be aborted with the following message

Contracting the selected area results in a self penetration of the mesh.

you may try to fill the area with the "add edge" command. This enables you to find the location where the problem occurred. Often there are two net points situated very close together (point has been measured twice). In that case very small triangles are generated during the intermeshing, which will lead to bad intermeshing results. In those cases delete one of the two points and intermesh the points again. In most of the times you will get a better result then.

Intermeshing – Show area of mesh (PNet)

	Command:	PNETAREA
	Ribbon tab	As-Built TS Design

The command "pnetarea" calculates the area of an existing polyface mesh (PNet) and displays it in the command line.

Command: pnetarea

Select polyface mesh: *(click the PNet)*

Area: 49.869014

Hint: For creating a PNet use the command →**Intermeshing – Intermeshing of points**.

Help – Manual for Total Station Interface

	Command:	TCMANUALBUILDING
	Ribbon tab	As-Built TS Control/Design

The command opens a help-document in PDF format.

To read the help-documents you need to have installed the Adobe Acrobat Reader on your computer.

The manual is divided into two parts: