Leica T-Scan Collect
Reference Manual

- when it has to be **right**

Leica
Geosystems

1 Introduction

1.1 Purchase

Congratulations on the purchase of the Leica Geosystems T-Scan Collect software.

This manual contains instruction for setting up the product and operating it. Read carefully through the User Manual before you use the product.

These are original instructions and part of the software. Keep for future reference and pass on to subsequent holders/users of the hardware or software. This User Manual contains information protected by copyright and subject to change without notice. No part of this User Manual may be reproduced in any form without prior and written consent from Leica Geosystems AG.

Leica Geosystems shall not be responsible for technical or editorial errors or omissions.

© Leica Geosystems AG

1.2 Trademarks

Product Names are trademarks or registered trademarks of their respective owners.

1.3 Validity of this Manual

This manual applies to the Leica Geosystems T-Scan Collect software.

1.4 Feedback

Your feedback is important as we strive to improve the quality of our documentation. We request you to make specific comments as to where you envisage scope for improvement. Please use the following E-mail address to send your suggestions:

support.tracker@hexagonmetrology.com

1.5 Contact

Leica Geosystems AG
Metrology Division
Moenchmattweg 5
5035 Unterentfelden
Switzerland
Phone +41-62-737 67 67
Fax +41-62-723 07 34
www.leica-geosystems.com/metrology

2 Contents

3   Revisions

Rev.	Date	Changes	Author	Released
1.0	04/2001		FP	
3.0	01/2003	Adjustments to Version 3.0	FP	BK
4.0	10/2003	Adjustments to Version 4.0	FP	MPo/AB
4.2	4/2004	Adjustments to Version 4.2	FP	BK
5.0	12/2004	Adjustments to Version 5.0	FP	
5.01	5/2005	Adjustments to Version 5.01	FP	
6.0	4/2006	Adjustments to Version 6.0	FP	SG/AB
7.0	7/2007	Adjustments to Version 7.0	FP	BK/SG
8.0	5/2009	Adjustments to Version 8.0	FP/FS	FS/BK
9.0	8/2010	Adjustments to Version 9.0	FP/FS	
10.11	10/2013	Adjustments to Version 10.11	FP	
10.2	9/2014	Adjustments to Version 10.20	MBN	
10.3	12/2014	Adjustments to Version 10.30	BK/MBN	
10.32	12/2015	Adjustments to Version 10.32	BK/MBN	
10.34	07/2016	Adjustments to Version 10.34	BK	

4 T-Scan Collect - Software – Introduction

4.1 Installation and Program Start

- Installation of T-Scan Collect

  T-Scan Collect has been pre-installed on your system. If you want to install it again, please use the Product USB Stick. Please note that T-Scan Collect is a 64-bit application and will only run on 64-bit operating systems.

- Start of T-Scan Collect

  Use the Windows start menu or the shortcuts on your desktop to start the application.
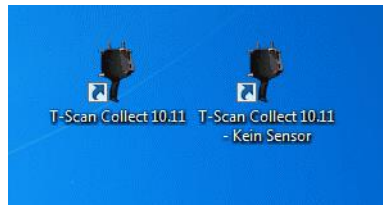
If you only want to use T-Scan Collect for viewing, processing or evaluating data (i.e. you do not want to measure), select the NoSensor shortcut which can be used without digitizing hardware.

4.2 Licensing

The T-Scan Collect software is protected by a CODEMeter dongle. The driver for the CODEMeter dongle is automatically installed with the software. If you would like to upgrade the software, please verify that the dongle is valid for the new software version. If you are unsure, please contact support.tracker@hexagonmetrology.com and request the required license file for upgrading the CODEMeter dongle to the new version. For more information, see chapter 19.1.

5 Working with Viewers

5.1 3D Viewer

After the program start the user interface shows up:
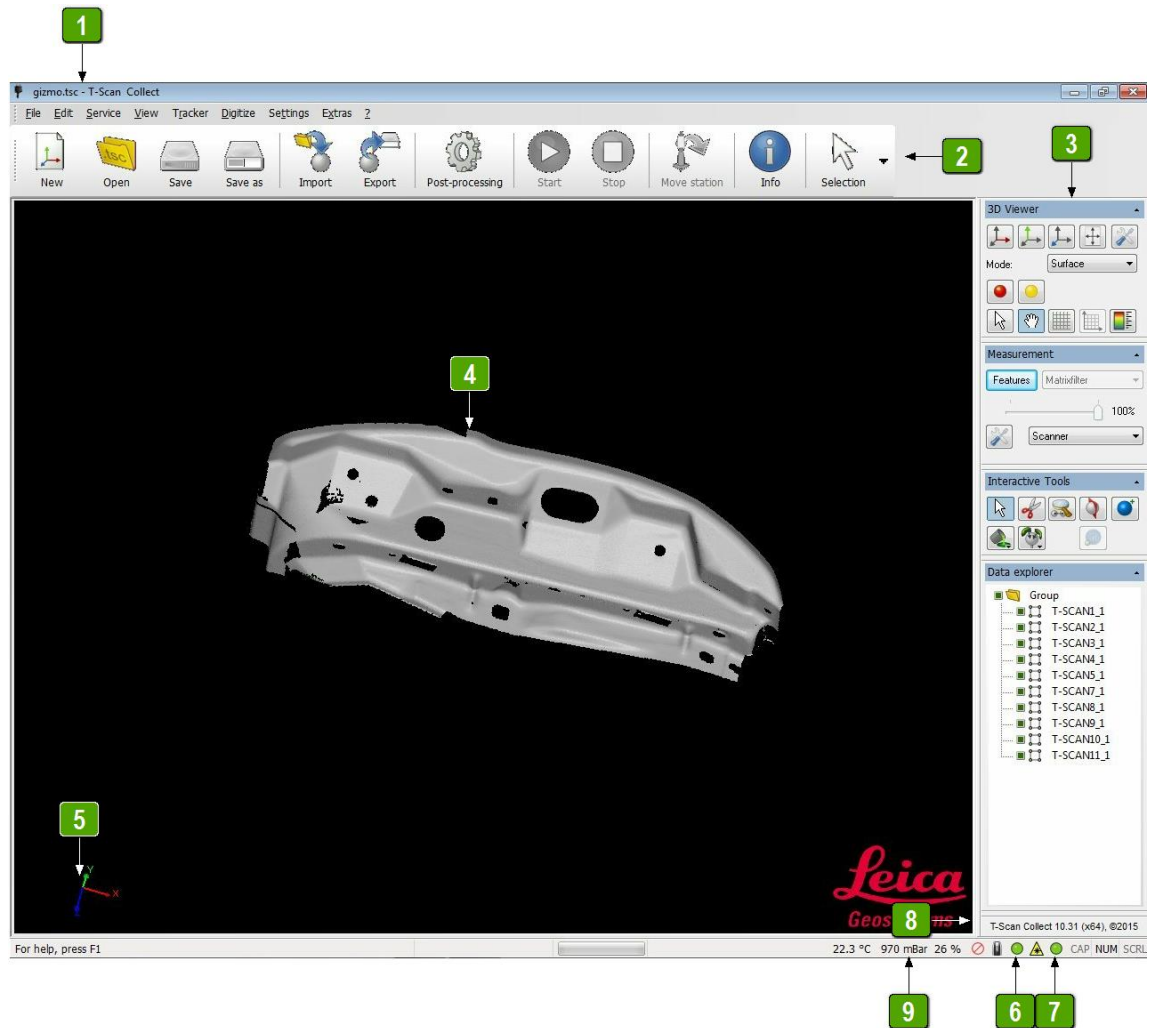
1 Title bar, contains the name of current project

2 Toolbar with icons for frequently used functions

3 Several toolboxes. To save space, you can minimize/restore the toolboxes by clicking on the corresponding button in their upper right corner

4 3D viewer with visualized data

5 Coordinate axes

6, **7** Status indicators for tracking system and controller, see chapter 5.2

8 Version information

9 Information from Weather Station

The area marked with **4** in the above image shows the actual 3D viewer. The 3D Viewer toolbox provides access to frequently used display functions:
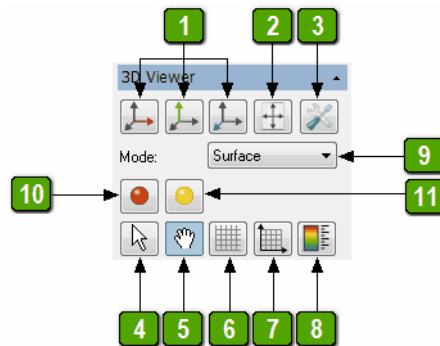
1 Selection of viewing direction along the positive X, Y or Z coordinate axis. If you hold the <Shift> key while clicking, the viewing direction is along the negative coordinate axis.

2 View all: If you use this button the zoom factor and the viewing position are adjusted so that all visible data is shown. You can also achieve this by pressing the middle mouse button while the cursor is over data or by holding down the <Shift> key and clicking with the left mouse button in the 3D viewer.

3 Opens the dialog for *the 3D Viewer Settings* (see chapter 17.3).

4 Selection mode: By left clicking a data set in the 3D viewer, this data set is displayed in the assigned selection color (see chapter 17.3.4) and marked red in the Data Explorer toolbox.

By right-clicking on a data point, the point is marked and its coordinates are displayed (picking).
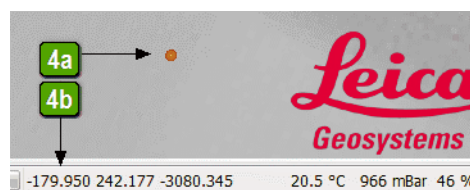
4a: Selected data point
4b: Point coordinates

5 Activation of viewing mode for interactive adjustment of the viewing direction, the position, and the zoom factor.

6 Toggles a grid in the 3D viewer which can be used to quickly estimate distances or sizes. The current distance between two major grid lines is displayed in the lower left corner. This function cannot be used at the same time as **7**.

7 Show coodinate axes: This option is only available if the viewing direction is along a coordinate axis (see **1**). The displayed coordinate axes are labeled. This function cannot be used at the same time as **6**. If you rotate the scene in the 3D viewer, the coordinate axis are automatically hidden.

8 Colorize: This option is available as soon as two or more rasterized data sets are displayed. When activated, overlapping areas of the data sets will be colorized according to the distance of related points; a color lookup table indicates which color represents which distance.
The following toolbox opens:

Here you can enter the maximum distance that should still be colorized. This allows a quick examination of the data quality and thus of the quality of the calibration. If large gaps occur between the rasterized data, a tolerance-based matching (see chapter 11.3.2) or, if necessary, a calibration (see chapter 9) should be performed. If deviations in data sets exceed the entered value, they are displayed in the default color like scans that do not overlap.
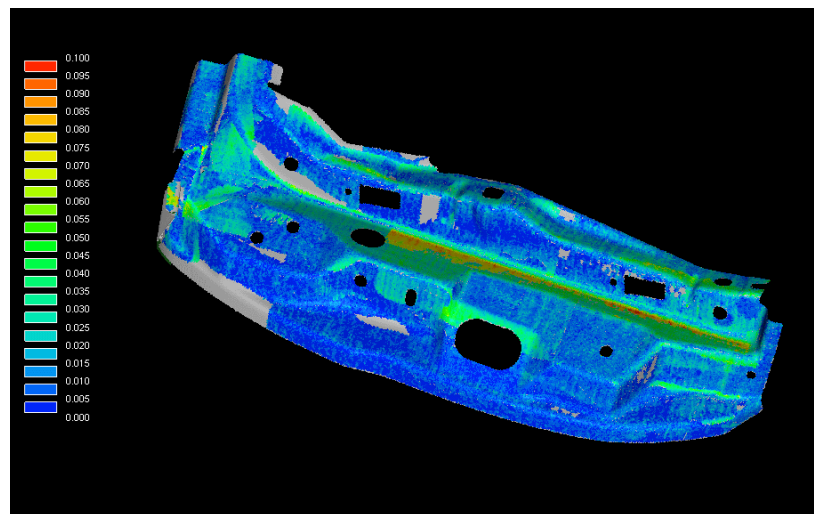
Note

Colorizing large data sets may take a few seconds.

---

9 Viewing mode: The available settings are described in chapter 17.3.3.

10 Show/hide reference points: Toggles visualization of all nominal points created with the Transform to Nominal Points wizard (see chapter 11.1.2).

10 Show/hide tie points: Toggles visualization of all tie points created with the Move Station wizard (see chapter 15.9).

The viewing direction, position and zoom factor in the 3D viewer can be interactively controlled with the mouse. Activate the viewing mode **5** or use <Space Bar> (as long as the cursor is in the 3D viewer). In viewing mode, the following options are available:

Orientation:
Press the left mouse button, keep it pressed and move the cursor. The visualized object area is shown using the dynamic display settings and is rotated according to the mouse movements. When you release the left mouse button, the object is shown using the static displays settings again.

Position:
Press the right mouse button, keep it pressed and move the cursor. The visualized object area is shown using the dynamic display settings and is moved according to the mouse movements. When you release the mouse button, the object is shown using the static display settings again.

Zoom factor:
Press the middle mouse button, keep it pressed and move the cursor up and down. The visualized object area is shown using the dynamic display settings and is zoomed according to the mouse movements. When you release the mouse button, the object is shown using the static display settings again.

You can also use the mouse wheel to change the zoom factor.

Zoom window:
Press the <Shift> key; the mouse cursor changes to an arrow with a rectangle. Press the left mouse button while holding down the <Shift> key and move the cursor. As long as the left mouse button is pressed, you can define a rectangular area. When you release the left mouse button, the selected area is zoomed to fit the 3D viewer.

Defining the center of rotation:
To define the center of rotation, briefly press the <S> key. The cursor changes to an arrow with a dot. With the left mouse button, click a point to define the center of rotation on the object's surface. The object is moved in such a way that the new center of rotation is in the center of the 3D viewer.

> **Note**
>
> *By default, the center of rotation is in the center of gravity of the virtual box, which encloses the whole object. However, you can freely determine the center of rotation as described above. To set the center of rotation back to default, use the tool* **2** *.*

Axis rotation:

Press the <Alt> key and then press the left mouse button. Keep both pressed and move the mouse cursor. A line from the center of rotation to the mouse cursor is drawn to help you control the rotation. The view is rotated according to the mouse movements.

Illumination:

Press the <L> key and then press the left mouse button. Keep both pressed and move the mouse cursor. The scene is illuminated according to the mouse movements.

5.2   Status Indicators

The status indicators are provided in the lower right area and can contain the following symbols:

Scanner: A symbol next to this graphic indicates whether a connection to the scanner has been established.

If this symbol is green, a connection has been successfully established. If it is red, the application has either been started in *NoSensor* mode or there is no connection to the sensor via the controller. In this case, switch off the controller and check all connections to the controller and to the sensor. Then switch the controller back on and wait until the symbol turns green.

Tracking system: A symbol next to this graphic indicates whether a connection to the tracking system has been established.

If this symbol is green, a connection has been successfully established. If it is red, the application has either been started in *NoSensor* mode or there is no connection to the tracker via the controller. In this case, switch off the controller and check all connections to the controller and to the tracking system. Then switch the controller back on and wait until the symbol turns green. The symbol for the tracking system is yellow during measurement.

Probe: This icon show the probe that is currently tracked:

Scanner, left side (side 1)

Scanner, right side (side 4)

Scanner, top side (side 2)

Scanner, rear side (side 3)

T-Probe

 Reflector

 No Probe

 Online Transformation: This symbol indicates that an online transformation is active
(see chapter 11.1.1).

5.3 Additional Coordinate Axes

By selecting *Settings* → *Coordinate axes*, you can display additional coordinate axes at
arbitrary positions within the 3D viewer. The following dialog appears:
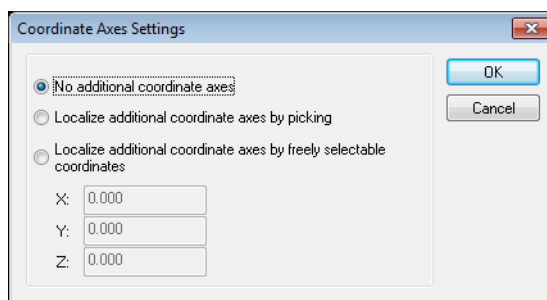
<u>No additional coordinate axes</u>
The additional axes will not be displayed.

<u>Localize additional coordinate axes by picking</u>
The positions of the additional coordinate axes can be defined by picking, as described in
chapter 5.1, item 4a . The additional coordinate axes are positioned at the selected points.

<u>Localize additional coordinate axes by selectable coordinates</u>
The entries for the X, Y and Z coordinates contain the last defined positions of the coordinate
axes, regardless of whether they have been defined by picking or entered manually.

New values take effect as soon as you click *OK*.

5.4 Data Explorer

This toolbox provides tools for grouping, editing and selecting data sets, and for the
customization of display characteristics.
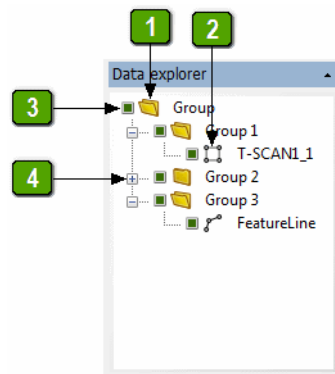
1 Group: The folder symbol represents a group, which may contain data or other groups.

2 Object name, data type and state: The object name is assigned automatically, but can be changed. The symbol depends on the data type and can look like this:

- Non-rasterized scan data
- Rasterized scan data
- Triangle mesh
- Polyline
- Polyline set
- Alignment data
- 3D point
- Unstructured point cloud
- NURBS curve
- Unknown data type

Symbols of selected data sets are highlighted in red, e.g. ⬚ for a selected, rasterized data set. If the data set is shown with customized display options, its symbol is marked with an asterisk: ⬚*.

3 Object visibility: This symbol can look like this:
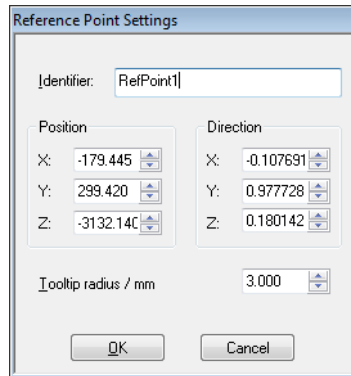
- ⬛ for data: data is visible;   for groups: all group entries are visible
- ☐ for data: data is hidden;   for groups: all group entries are hidden
- ▣ only for groups: some group entries are visible, some are hidden

4 By clicking "+" or "-" you can expand or collapse groups.

When at least one object is selected in the data explorer, you can open a context menu with the following commands by clicking the right mouse button:

Rename	R
Show	H
Select	S
Cut	Ctrl+X
Paste	Ctrl+V
Delete	Del
Import...	
Export...	
Edit...	
Convert to NURBS...	
New group	Ctrl+G
Object(s)	▸
Alternate colors per group	Alt+G
Alternate colors per object	Alt+O
Default color	Alt+D
Shiny material	Alt+F
Environment Mapping...	Alt+M
Properties...	P

Rename   Renames the selected object. You can also achieve this by pressing <F2> or clicking on the selected object.

Show / Hide   Shows or hides object(s). You can also achieve this by clicking the ■, ■ and ☐ symbols.

Select / Deselect   Selects or deselects object(s)

Cut   Removes the selected objects and adds them to an internal clipboard

Paste   Adds objects from the internal clipboard to the selected group

Delete   Deletes the selected objects; this action cannot be undone!

Import   Imports data to the current group

Export   Exports selected data to a file

Edit   Only available for reference points; the following dialog appears:

Identifier:
Displays the name of the reference point and allows renaming it

Position:
Displays the position of the current point
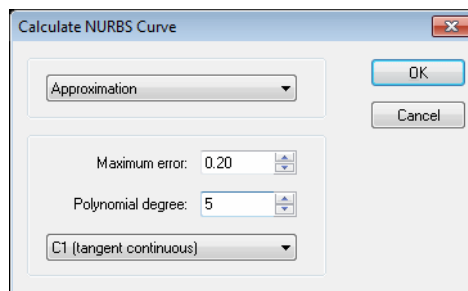
Direction:
If available: Direction from which the reference point was measured; the length of the direction vector is normalized to 1 and the point is shown in green. If no direction information is available, the fields display (0, 0, 0) and the point is shown in blue.

Tooltip radius:
Displays the radius of the T-Probe's tip with which the point was acquired.

Convert to NURBS | Only applicable to lines; the following dialog appears:

Interpolation:
The NURBS curve passes exactly through the measurement points. By adjusting the *Polynomial degree* and choosing the method, you can define how the NURBS curve will be created.

Approximation:
The NURBS curve is approximated to the measurement points and will not deviate by more than the specified maximum error at the control points. By adjusting the *Polynomial degree* and choosing the method, you can define how the NURBS curve will be created.

Three methods are available:

C0 (discontinuous): Neither the tangent nor the curvature of the created NURBS curve will be continuous

C1 (tangent continuous): The tangent of the created NURBS curve will be continuous

C2 (curvature continuous): The curvature of the created NURBS curve will be continuous

Note

Depending on the data and the settings you choose it might happen that the NURBS curve cannot be calculated. In this case, adjust the parameters and recalculate the curve.

New group	Adds a new subgroup to the selected group
Object(s)	Opens the submenu for other object settings
Alternate colors per group	Assigns a different color to each group
Alternate colors per object	Assigns a different color to each object
Default color	Assigns the default color to each object
Shiny material	Assigns a shiny color and material to each object
Environment Mapping	Opens a dialog where you can select an image that is projected onto the objects and helps to identify faults in the surface. For best results select *Smooth shading* in the 3D viewer settings.
Properties	Displays the characteristics of the object

Object(s) submenu:

Color...	Ctrl+C
Material...	
Group	G
Ungroup	
Use customized settings	
Customize...	
Flip inside/outside	

Color	Opens a dialog where you can choose a color for the selected objects
Material	The following dialog appears:
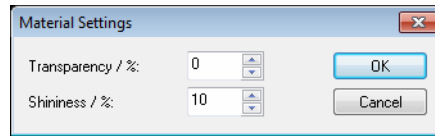
Transparency:

Enter the transparency of the object (in percent). 0% means the object is opaque, 100% means it is translucent.

Shininess:

Enter the shininess of the object (in percent). 0% means the object is not shiny, 100% means the object is very shiny.

Group	Combines all selected data sets (also those in a selected group) into a new group. The name of the new group is the name of the first data set in the group, followed by the number of data sets in the group (if the group contains more than one data set).
Ungroup	Moves all selected data sets (also those in a selected group) to root and then deletes all selected groups.
Use customized settings	The selected objects are either displayed with the global settings for the 3D viewer (see chapter 17.3) or with customized display settings. The symbols of objects with customized display settings are marked with an asterisk, e.g.: .
Customize	Opens a dialog for customizing the display settings for the selected objects (see also chapter 17.3)
Flip inside/outside	Flips the inside and outside of a triangle mesh

Note

Please note that some commands are only available if a single object is selected, while other commands are also available if multiple objects are selected.

You can select multiple objects in different ways:

1. *Click the first object, then move the cursor to the last object. Press and hold the <Shift> key and click the last object. All objects in-between are selected.*

2. *Click the first object, then press and hold the <Ctrl> key and click other objects. Clicking an already selected object deselects it.*

3. *You can combine the two methods described above.*

4. *You can also use the shortcuts <Shift> + <PageUp> / <PageDown> / <Home> / <End > to select multiple objects.*

Please keep in mind that any changes you make are also saved in the project. The next time you open the project, all settings that were in effect the last time you saved the project will be active again.

Shortcuts, cursor keys and drag and drop are easy ways to increase efficiency. When using shortcuts, make sure that the data explorer has the focus.

6 Customizing the User Interface

The T-Scan Collect user interface can be customized. You can define your own shortcuts, menus and toolbars. Choose *View* → *Toolbars* or right-click a toolbar to open the context menu.
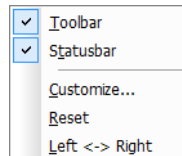
The context menu provides the following functions:

Toolbar
Shows or hides the standard toolbar

Status bar
Shows or hides the status bar

User defined toolbars
If you defined your own toolbars, you can choose which to show or hide

Reset
Resets all shortcuts, menus and toolbars to the default settings of T-Scan Collect. All user-defined changes will be lost. This action cannot be undone!

Left ←→ Right
Swaps the positions of the toolbars

Customize...
Opens a dialog where you can define and change shortcuts, menus and toolbars. As soon as the dialog is open, you can drag and drop buttons onto or off toolbars to move, add or delete buttons.

The dialog contains various tabs:

Commands
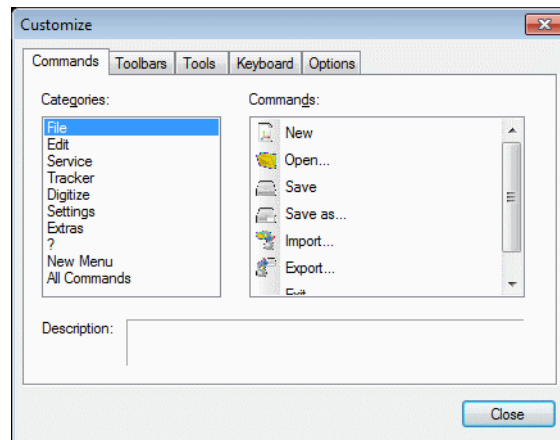
You can add all listed commands to a toolbar. Left-click a command and drag it onto a toolbar while holding the mouse button down. When you release the mouse button, the command is added to the toolbar.

You can also remove a command from a toolbar: Click the icon and drag it off the toolbar while holding the left mouse button down.

Toolbars
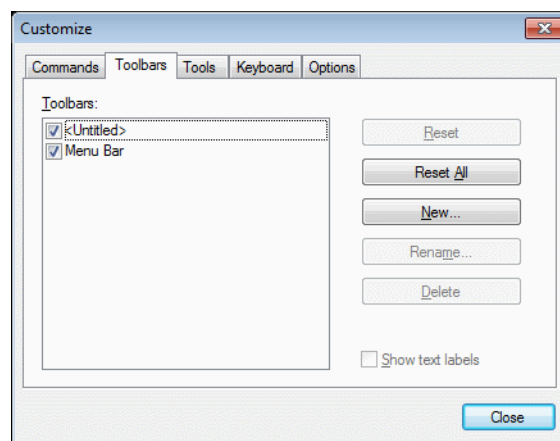
A list shows the available toolbars. By selecting or deselecting the checkboxes in the list, you can show or hide the toolbars.

To create a new toolbar, click *New...*. A dialog box appears where you can enter a name for the new toolbar.

To delete a toolbar, select it and click *Delete*.

Click *Rename…* to rename the selected toolbar. A dialog box appears where you can enter a new name for the toolbar.

You can reset the standard toolbar to the default settings of T-Scan Collect by clicking *Reset*. This action cannot be undone!

To reset all toolbars to the default settings of T-Scan Collect, click *Reset All*; user-defined toolbars will be lost. This action cannot be undone!

You can choose whether to show or hide the text labels of the icons by selecting or deselecting the *Show text labels* checkbox.

Tools
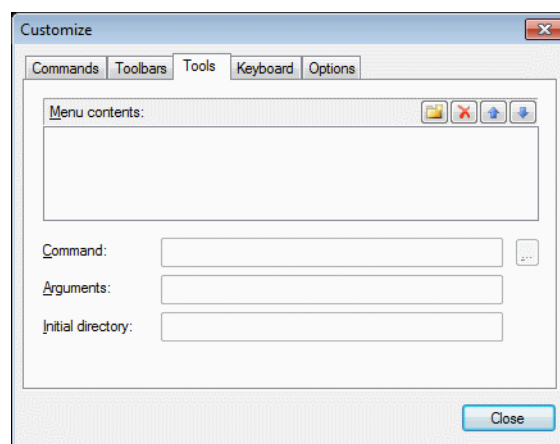
With this tab, you can manage the commands provided in the *Extras* menu.

By pressing the <Ins> key or clicking the *New* icon, you can create a new menu entry. An input line appears where you can enter a name for the menu entry, e. g. "Start COMET Inspect". In the *Command* line, define the application that will be started when the entry is selected; e. g. D:\Program Files\Steinbichler\INSPECTplus 5.23\ INSPECTplus.exe. If you want to pass arguments to the application, enter those in the *Arguments* line. *Initial Directory* defines the directory in which the command will be executed.

By pressing <Del> or clicking the *Delete* icon, you can delete the selected entry.

By pressing <Alt> and <Up> or <Alt> and <Down> or the icons 🔼 and 🔽, you can change the position of the selected entry within the menu.

Keyboard
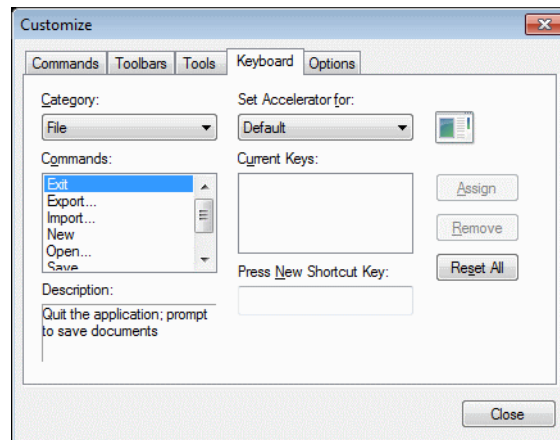
Here you can define shortcuts for all menu commands. In the *Category* list, select the menu that contains the command. The *Commands* list then shows all the commands of this category. Select the command for which you want to change the shortcut. The current shortcut (if available) is displayed in the *Current Keys* box. Select the input line for *Press New Shortcut Key* and press the key combination for the new shortcut. Then click *Assign*.

To remove the shortcut for a command, select the command and click *Remove*.

To reset all shortcuts to the default settings of T-Scan Collect, click *Reset All*. This action cannot be undone!

Options

Select or deselect *Show ScreenTips on toolbars* and *Show Shortcut keys in ScreenTips* according to your preferences. It is recommended to select both options.

The *Large Icons* function is currently not supported.

7 Measurement

7.1 Selection of Measurement Mode

To select a measurement mode, choose *Settings* → *Measurement…* or use the Measurement toolbox:
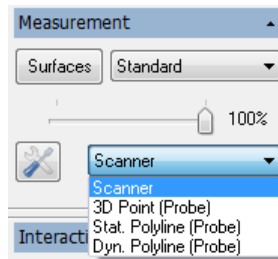
- In *Surface* mode, object surfaces can be digitized. This mode is activated after the program start by default

- In *3D Point* mode, single points will be measured with the T-Probe (Probe)

- In *Static Polyline* mode, lines will be measured with the T-Probe (Probe) where each point of the line has to be measured explicitly

- In *Dynamic Polyline* mode, lines are measured with the T-Probe (Probe) where the points of the line will be measured automatically according to the chosen settings

7.2 Measurement Settings

The settings can be adjusted separately for almost every measurement mode. To do this, choose *Settings* → *Measurement…*. A dialog appears where you can select the tab for the measurement mode you want to adjust. You can also open the dialog from the measurement toolbox by clicking the *Measurement parameter settings* icon.

7.2.1 General
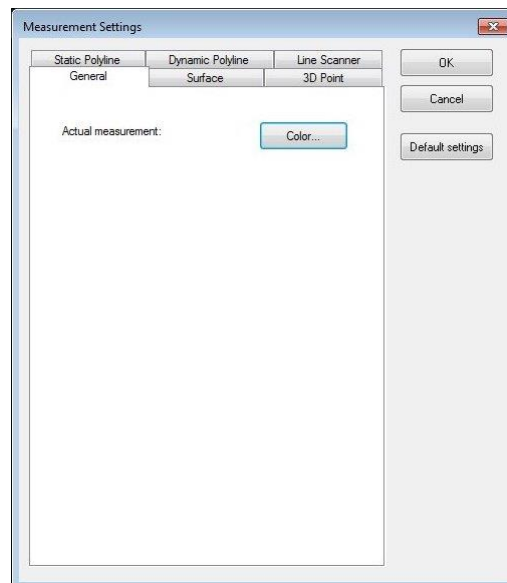
Color

This button opens a dialog where you can specify a color for the last measurement data you acquired.

7.2.2 Line Scanner
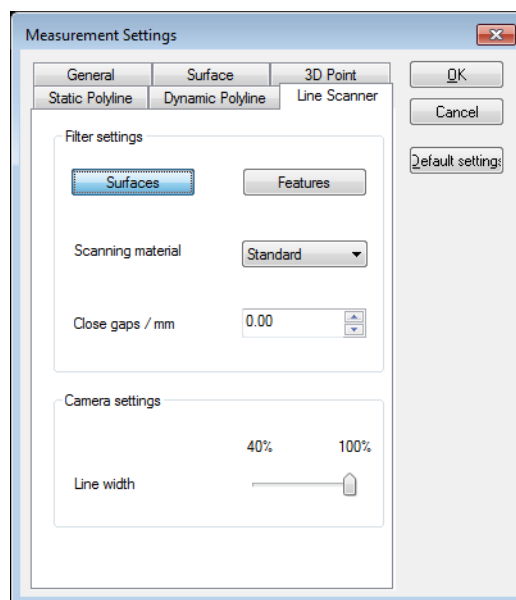
This dialog provides settings for adjusting the line scanner. If inappropriate settings are made here, however, you might not be able to acquire any measurement data. If you would like to restore the default values, click the *Default Settings* button.

Surfaces

Press *Surfaces* if you want to measure surfaces. In this mode, measured data are rasterized immediately after acquisition using the selected settings (see chapter 7.2.3). Besides, the value for *Close gaps* will be used as well as the reflection filter defined in the materials editor (see chapter 17.10).

Optionally you can select *Surfaces* in the *Measurement* toolbox by hitting *Surfaces*/*Features*.

Features

Press *Features* if you want to measure features like e.g. gaps or boundaries. In this mode, measured data are not rasterized, but visualized as a point cloud. *Close gap*s will be deactivated.

Optionally you can select *Features* in the *Measurement* toolbox by hitting *Surfaces*/*Features*.

Material selection

From the list of materials, select the one which resembles your object's properties the most. By default, the following materials are available: *Standard*, *Shiny bright*, *Dark* and *Shiny dark*.

You can add, modify or delete further materials using the *Material Editor* (see chapter 17.10).

Optionally you can select a material in the *Measurement* toolbox.

Close gaps

If gaps appear within a scan line, they can be filled automatically by means of interpolation. Specify a value for the maximum gap size up to which a gap will be closed automatically.

This setting is useful when measuring surfaces like fabrics, felt, foam material or similar, and a complete surface is required.

Line width

With this slider, you can reduce the width of the line to be scanned down to 40% of the maximum width. By choosing a smaller line width, you can increase the line frequency.

Optionally you can modify the line width in the *Measurement* toolbox.

Note

Please note that the visible width of the laser line is not influenced by this setting.

7.2.3 Surface Mode
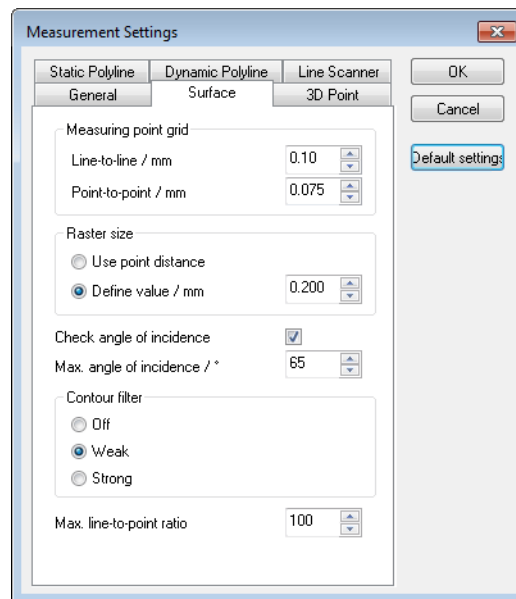
Line to line

This value defines the minimum distance between two scan lines. The minimum distance prevents the acquisition of too dense data if the scanner is moved very slowly or not at all. The software checks if the scanner moved more than the entered Line to line value. If it did not, the current scan line is rejected.

The absolute distance between the scanner positions is calculated, i.e. spatial movement in the scan line direction also increases the distance. Only the distance to the previous scan line is calculated. If the scanner is repeatedly moved over the same point, data is acquired with each new movement.

Point to point

This value determines the average point distance on the scan line. It refers to the center of the scanner's measurement range and changes with increasing or decreasing distance to the object. The value can only be changed in 0.075 mm steps.

Raster size

If the *Use point distance* radio button is selected, the raster width is set according to the point to point distance.

If you want to use a different raster width, select *Define value* and specify the raster width. Bear in mind that a raster width of less than 0.1 mm will dramatically increase the amount of data generated.

A rasterization process is started when

- *the Measurement Settings for the Line Scanner is set to* Surfaces *(see chapter 7.2.2) or*

- *post-processing is started and the scene contains non-rasterized scanner data.*

<u>Check angle of incidence</u>

If this checkbox is selected, a filter is applied to the data. The filter removes points measured with an angle of incidence that exceeds the specified maximum value. How this filter works is described in more detail below.

<u>Maximum angle of incidence</u>

Measured points for which the angle of incidence between the laser beam and the surface exceeds the entered value are rejected. The smaller the value, the less data you will acquire, but the higher the data quality will be.

The angle is initially checked within each scan line. If the Measurement Settings for the Line Scanner is set to *Surfaces* (see chapter 7.2.2), the angle is additionally checked in the direction of motion of the scanner.

The following illustrations demonstrate the principle of the filter. The arrow indicates the direction of motion of the scanner.

Angle treatment when moving in the direction of motion

The angle between the laser plane and the object exceeds the entered threshold: No measurement data will be acquired.
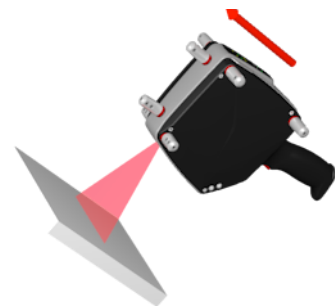
Measuring correctly: The angle between the laser plane and the object is almost perpendicular.

Angle treatment within a scan line

No measurement data is acquired if the angle between the laser beam and the surface exceeds the entered value.
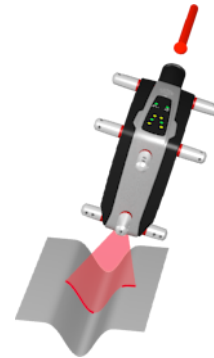
To measure the surface completely, do two scans. Tilt the scanner first in one direction...

...and then in the other direction.

Contour filter

This filter is only applied if the measurement settings for the Line Scanner is set to *Surfaces* (see chapter 7.2.2). It can improve the data quality of the scan data when measuring contoured objects using an inappropriate measurement strategy.
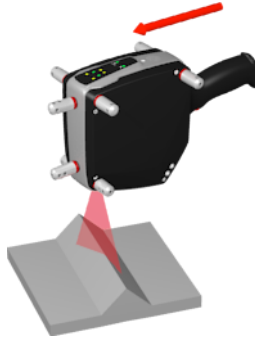
Inappropriate measurement strategy: the scanner is moved too fast and the scan line is almost parallel to the object's contour. The contour filter will remove data.

If the scanner is moved too fast relative to the selected point-to-point distance and the scan line is almost parallel to the object's contour (see illustration), surface elements that do not lie completely on the object surface might be created between neighboring scan lines. The object's contour will then look 'blurred'.

The contour filter analyzes the points of neighboring scan lines and prevents the points from being connected to a continuous surface if an inappropriate measurement strategy is used. Therefore, gaps in the data may occur, providing a visual indication to the user that this area should be measured again using a more suitable strategy.

The filter strength can be adjusted to achieve optimal data quality. If the filter is set to *strong*, more data will be rejected. If the filter is set to *weak*, less data will be rejected; however, data might then be retained that does not actually lie on the object's surface. You can also disable the filter if you want to keep the data in any case.

The measurement strategy described above should generally be avoided. Instead, you should always try to measure perpendicularly to the object's contours (see illustration below). Even if the filter is set to *strong*, the algorithm will detect if an appropriate strategy has been used and will retain all data.
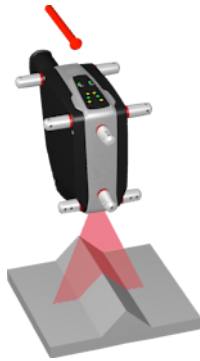
Appropriate measurement strategy: The scanner is moved perpendicularly to the object's contour. The contour filter will retain all data.

Max. line-to-point ratio

If the speed the scanner is moved over a measurement object is too fast relative to the selected point-to-point distance, widely spaced scan lines will be created. When this data is then rasterized, the point-to-point distance used is taken as the basis for rasterization. As a result, the "gaps" between the scan lines may be filled (interpolated) with many points, which excessively increase the data volume without improving the data quality.

Ideally, the scanner is moved fast enough so that the distances between points on a scan line are approximately equal to the distances between neighboring scan lines.

If the ratio of "distance between scan lines" to "point distance within a scan line" exceeds the specified value, these scan lines are not connected to create a continuous surface, but gaps are left. This can help the user to optimize the measurement strategy and avoid unnecessarily large data volumes.

7.2.4 3D Point Mode
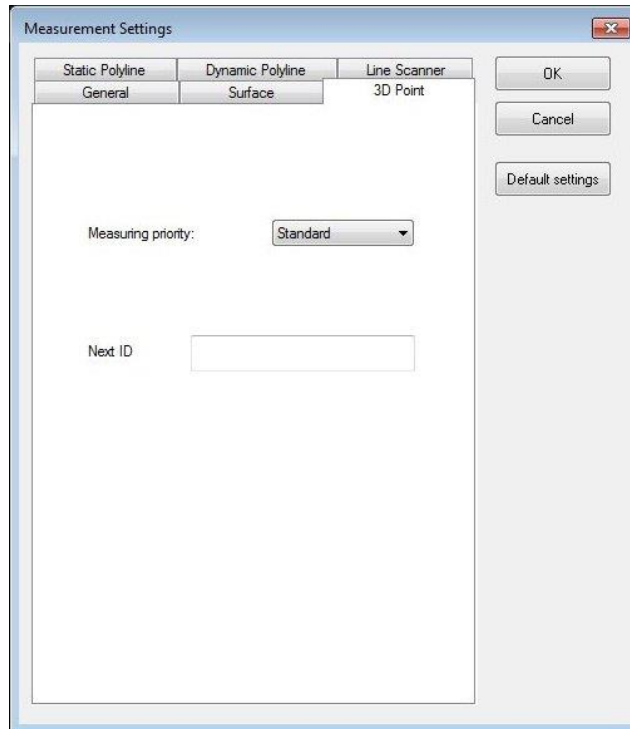
For this measurement mode, data is acquired and averaged internally. The averaged value is output as a measurement point.

Measuring priority
This setting controls the method how a 3D point is measured. By default, the following options are available: *Precise, Standard* and *Fast*.

Next ID
This field is only available in the settings for 3D points and defines the name for the next point to be measured. If the ID does not end with a digit, a sequential number – beginning with "1" – is automatically appended to the name.

7.2.5 Static Polyline Mode
The settings for these two modes are identical.
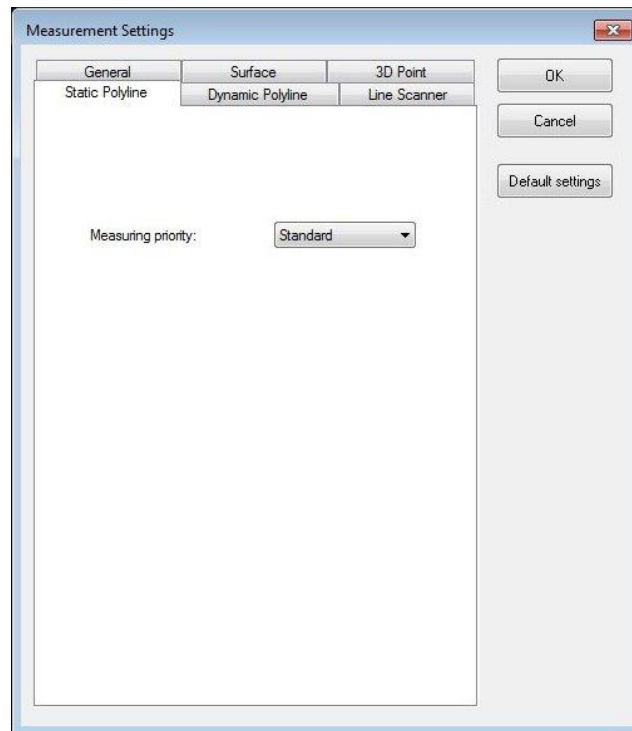
For this measurement mode, data is acquired and averaged internally. The averaged value is output as a measured point.

Measuring priority

This setting controls the method how a 3D point is measured. By default, the following options are available: *Precise*, *Standard* and *Fast*.

7.2.6 Dynamic Polyline Mode
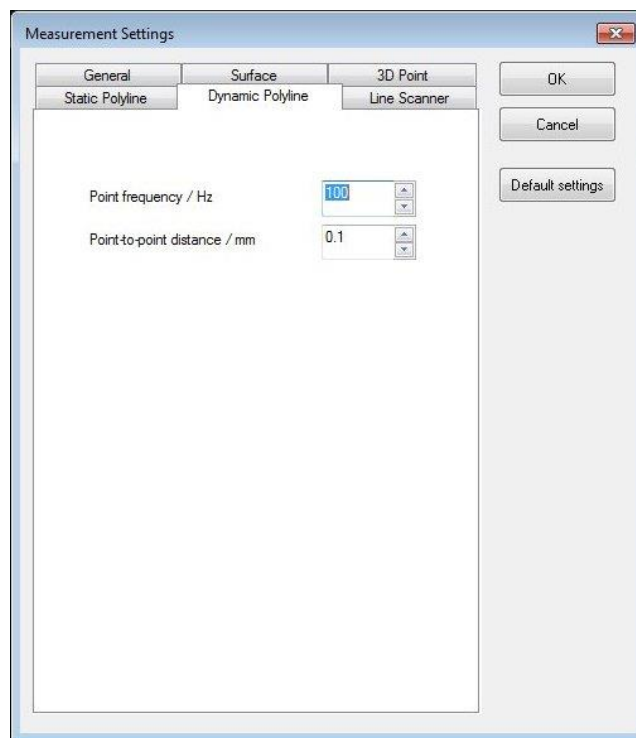
Point frequency
The points will be acquired with the entered frequency.

Point-to-point distance
This value defines the minimum distance between two successive points. The minimum distance prevents a too dense data recording if the T-Probe is not moved or moved very slowly. The software checks if the T-Probe moved more than the *Point-to-point distance* value. If not, the current point will be rejected.

7.3 Starting a Measurement
When you click the *Start* icon in the standard toolbar or choose *Digitize → Start*, the scanner or the T-Probe is activated in the selected measurement mode (Surface, 3D Point, Static Polyline). The *Start* icon is disabled and the *Stop* icon is enabled.

7.3.1 Surface Mode
In *Surface Mode*, 3D points on the object's surface can be measured with the T-SCAN scanner.
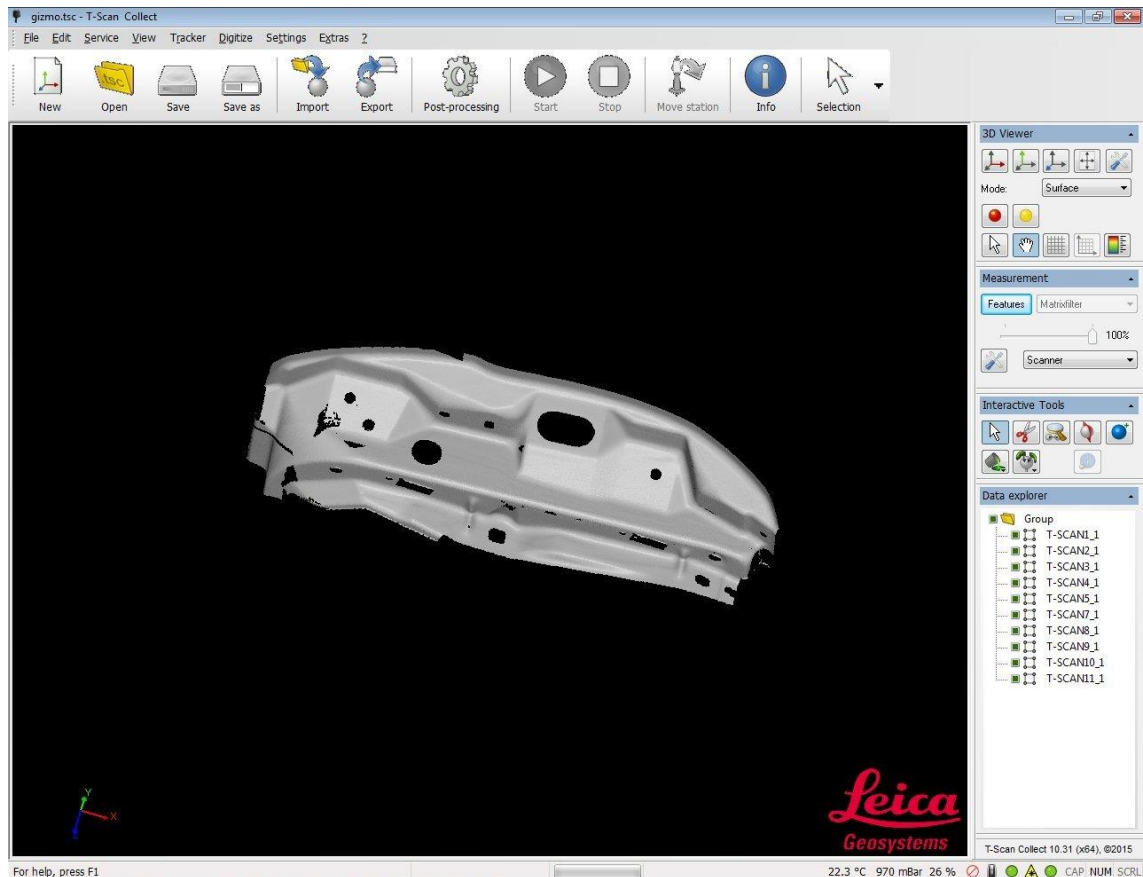
The measurement itself is started as soon as you use the trigger button on the scanner handle. T-Scan Collect automatically becomes the active application – depending on the settings (see chapter 17.15) – if another application was active. You can make as many measurements as required.

During measurement, 3D coordinates are calculated and the newly measured data is shown in the selected color (typically red, see chapter 7.2.1).

If the measurement setting for the line scanner is set to *Surfaces* (see chapter 7.2.2), the data is displayed in the default color immediately after it has been rasterized.

A new entry is created in the data explorer toolbox. If the measurement setting for the line scanner is set to *Features* (see chapter 7.2.2), the entry is named "T-SCAN[sequential number]". The number is incremented with each scan.

If the measurement setting for the line scanner is set to *Surfaces*, a scan might be split up into several parts. This happens if the scanner orientation changes significantly during a scan, thus influencing the rasterization direction. In the data explorer, splitted scans are named "T-SCAN [sequential number]_[sub number]", where [sub number] always starts with 1.

7.3.2 3D Point Mode

In *3D Point Mode*, single 3D points can be measured with the T-Probe. Depending on the selected settings (see chapters 17.13 and 17.14), the T-Probe coordinates window and/or the wireframe of the T-Probe will show up as soon as the measurement starts.

To acquire a measurement point, press the trigger button on the T-Probe or press the <M> key. As soon as the measurement was successfully completed, the measurement point is displayed in green color. You can acquire as many measurement points as are required.
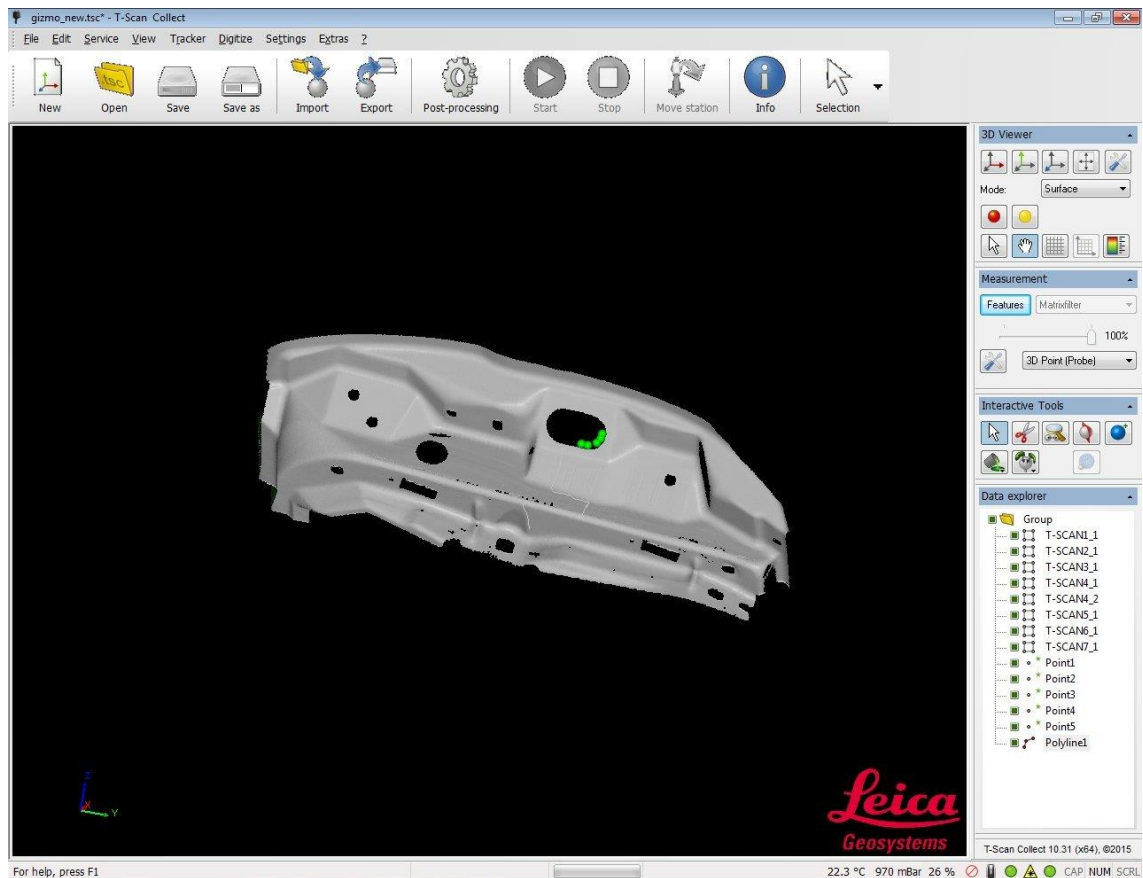
Note

Every measured point is stored in a separate data set and named according to the settings you made (see chapter 7.3.2). If a data set of the same name already exists, another data set is created using the next free number.

By pressing the <D> key, you can delete the points in the reverse order in which they were acquired.

7.3.3 Static Polyline Mode

In *Static Polyline Mode*, single 3D points are measured with the T-Probe and automatically connected to create line segments. This way, you can acquire polylines. Depending on the selected settings (see chapters 17.13 and 17.14), the T-Probe coordinates window and/or the wireframe of the T-Probe will show up as soon as the measurement starts.

To acquire a measurement point, press the trigger button on the T-Probe or press the <M> key. As soon as the measurement was successfully completed, the measured point is visualized

and connected to the previous point by a line. You can adjust the point size and line size (see chapter 17.3) to optimize the visualization. A polyline may contain any number of points.

You can start a new polyline by pressing the <N> key. The last active polyline is closed and displayed in blue.
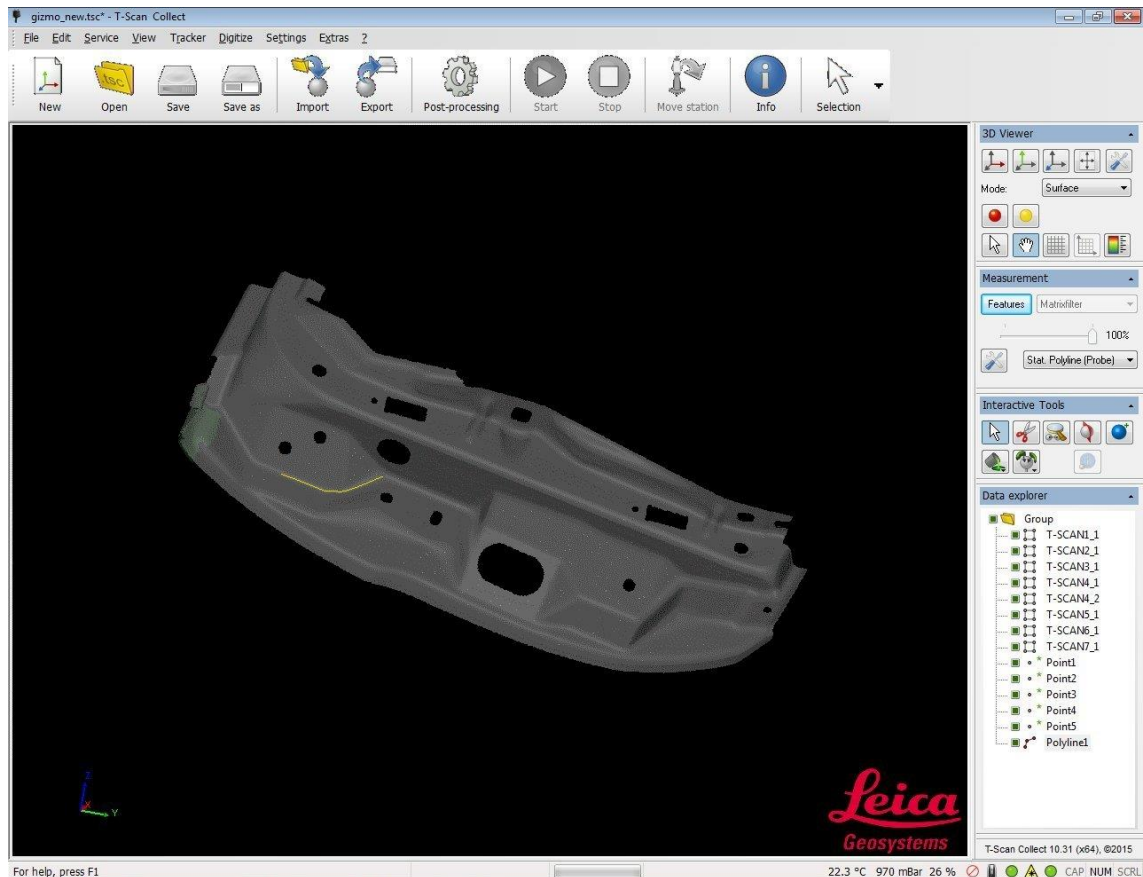
Note

Polylines are saved in data sets named "Polyline[X]", where [X] is a sequential number. If you rename such a data set, it will not be treated as a polyline anymore.

Pressing <D> deletes points from the "Polyline[Y]" data set, where [Y] is the highest existing number. If the last point of a polyline is deleted, the polyline itself will also be deleted. The next delete action will open the data set with the now highest number and delete the last measured point from that line.

7.3.4 Dynamic Polyline Mode

In *Dynamic Polyline Mode* 3D points can be measured continuously with the T-Probe. The points will be connected automatically to form line segments. According to the settings (see

chapters 17.13 and 17.14) the T-Probe coordinate window and/or the wire frame model of the T-Probe will be displayed as soon as the measurement starts.

As long as the right mouse button or the button of the T-Probe is pressed, points will continuously be acquired with the chosen settings (see chapter 7.2.6). By releasing the button, the polyline will be completed. Pressing the button anew will create a new poly line.
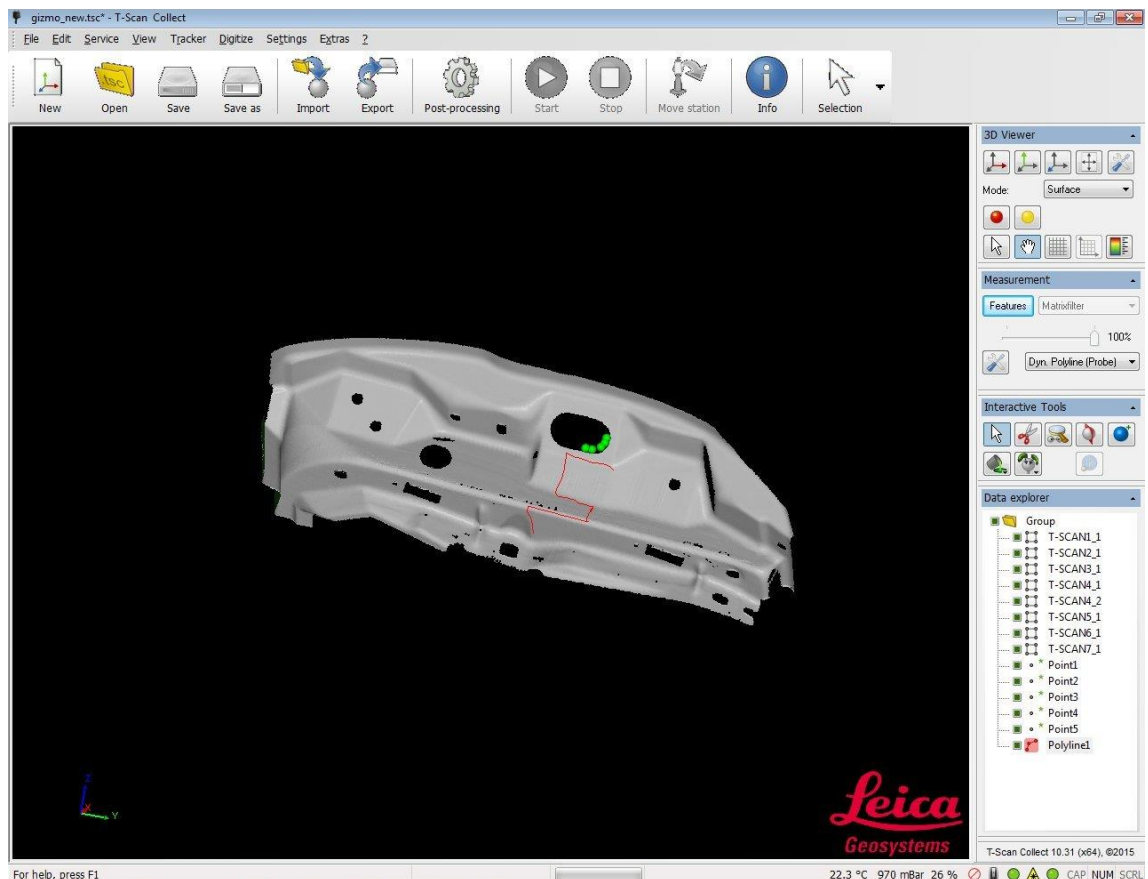
Remarks

Lines will be stored in data sets named „Polyline[X]" where [X] is a running number. If you rename such a data set it will not be treated as a poly line anymore.

7.4   Stopping a Measurement

To stop a measurement, click the *Stop* icon in the standard toolbar or choose *Digitize* → *Stop*. The infrared LEDs are switched off and the scanner goes into sleep mode. The *Start* and *Stop* icons change their states accordingly.

You can continue the measurement at any time. All you need to do is restart the measurement. The measured data is added to the existing data.

8 Interactive Tools

There are several interactive tools you can use for editing existing data sets. These tools are provided in the 'Interactive Tools' toolbox and can also be accessed by selecting *Edit* → *Interactive Tools* or the drop down menu in the toolbar.

The toolbox contains the following tools:
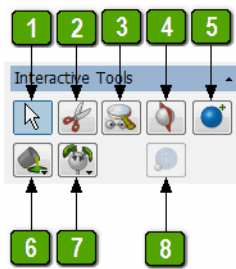
[1] Selection mode ("No Tool")

[2] Interactive clipping

[3] Check accuracy

[4] Cross sections

[5] Create reference points

[6] Fill holes / tools for editing triangle meshes

[7] Alignment calibration

[8] The Data Analysis tool is currently not used and therefore is not available

The tools [6] and [7] have a small arrow in the bottom right corner. The arrows indicate that a submenu can be opened when clicking the icons and keeping the mouse button pressed.
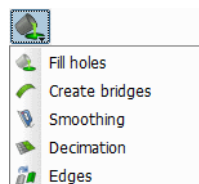
The functions of these icons can be modified by selecting a menu item.

The tool currently in use is shown in the toolbar. A blue background indicates whether the tool is active ("tool mode") or inactive ("view mode"). By pressing <Space Bar> in the 3D viewer, you can switch between these modes.

8.1 Selection Mode

This icon activates the selection mode. When you click a data set in the 3D viewer, it is displayed in the assigned selection color and shown in red in the data explorer (see chapter 5.4).

8.2 Interactive Clipping

This tool allows you to clip data of any type. If all data of an object or entire objects are clipped, the corresponding entry is removed from the data explorer.

The toolbox offers the following options:
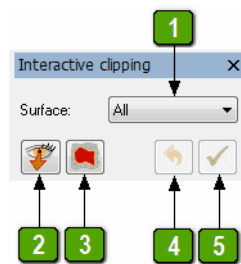
1 Choose whether the clipping action will apply to *All* faces, only *Front* faces or only *Back* faces. This option is only effective for objects that have back and front faces.

2 If this option is enabled (symbolized by a "normal" eye), no data will be deleted that is currently invisible, e.g. because it lies outside the 3D viewer or is hidden by other data. If the option is disabled ("X-ray eye"), all the data within the selection will be deleted.

3 If this option is enabled, data within the selected area will be retained and all data outside the selection will be deleted. The selected area will then be shown in green instead of red.

Move the mouse cursor over the area containing the data you want to edit. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer.

Define the area to clip with the left mouse button. By left-clicking, you can define a straight line. If you keep the left mouse button pressed, you can draw a continuous line. Close the area by double clicking with the left mouse button or pressing the right mouse button. You can also select multiple areas in this way.

Press **5** or the <Enter> or <Del> key to clip the selected areas. Press **4** to undo the last action or actions you made.

8.3 Check Accuracy

With this tool, you can check the measurement accuracy of the *T-SCAN* system.
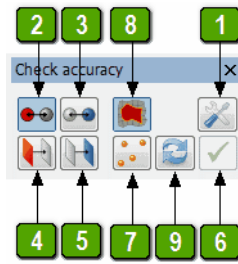
1. Before running an accuracy check, it is advisable to review the nominal values in the settings dialog 1:
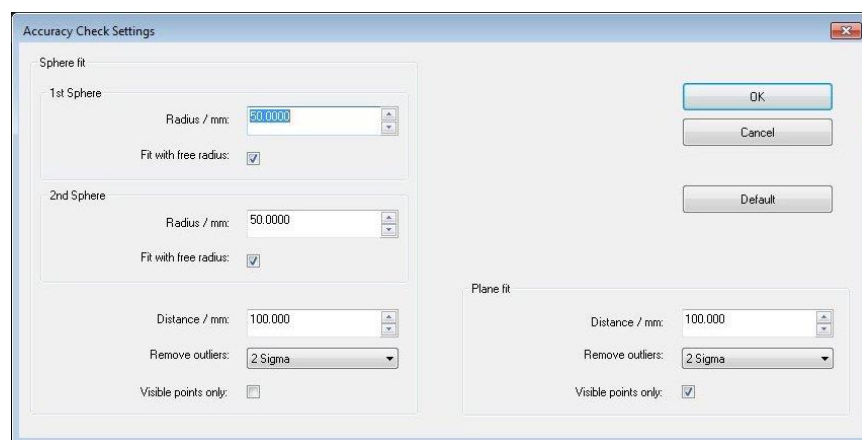
1. If you are using a barbell, enter the sphere radii into the corresponding fields. *Sphere 1* corresponds to the red, *Sphere 2* to the blue sphere displayed in the toolbox. The correct *Distance* between the two spheres also needs to be specified.
   If you are using a single sphere, you are free to choose which field to use for the nominal radius. Take care to select the correct sphere (red/blue) later in the evaluation.

2. If *Fit with free radius* is disabled when you perform the accuracy check, the software will try to fit a sphere with the given radius into the data. This may lead to an error message if it is not possible to calculate a reasonable result, e.g. because the given radius differs significantly from the actual data.

3. In the *Remove outliers* fields, you can specify how many of the worst fitting points will be removed before sphere fit and plane fit. A setting of *1 Sigma* will eliminate approximately 32%, *2 Sigma* approximately 5% and *3 Sigma* approximately 3 ‰ of the most off-size points.

4. For the sphere fit, *Visible points only* is disabled by default because, in most cases, you will want to check the entire sphere, including the rear side.
   For the plane fit, *Visible points only* is enabled by default because usually only the

selected points are part of the plane, whereas surfaces or data sets located behind the plane are to be left unconsidered.

2. Now use the lasso [8] in the 3D viewer to select the first or the only sphere/plane you want to check. Make sure that the correct sphere or plane is selected in the toolbox (red/blue). Then start the evaluation by pressing <Enter> or clicking [6].

3. The following result dialog appears:

Example
for one sphere: for two spheres:

Accuracy Check Results		
Sphere analysis		
	Sphere 1	Sphere 2
Sphere X:	1245.877	
Y:	3453.695	
Z:	-347.381	
Nominal radius / mm:	25.0000	
Measured radius / mm:	24.9918	
Radius deviation / mm:	-0.0082	
Standard deviation / mm:	0.0359	
Mean deviation / mm:	0.0290	
Negative maximum / mm:	-0.0982	
Positive maximum / mm:	0.0970	
Points:	23036	
Sphere		
Nominal distance / mm:		
Measured distance / mm:		
Length deviation / mm:		
Close	Save log...	

Accuracy Check Results		
Sphere analysis		
	Sphere 1	Sphere 2
Sphere X:	1245.877	85.815
Y:	3453.695	3457.025
Z:	-347.381	-375.753
Nominal radius / mm:	25.0000	25.0000
Measured radius / mm:	24.9918	24.9981
Radius deviation / mm:	-0.0082	-0.0019
Standard deviation / mm:	0.0359	0.0314
Mean deviation / mm:	0.0290	0.0257
Negative maximum / mm:	-0.0982	-0.0795
Positive maximum / mm:	0.0970	0.0811
Points:	23036	23436
Sphere		
Nominal distance / mm:	1160.4130	
Measured distance / mm:	1160.4142	
Length deviation / mm:	0.0012	
Close	Save log...	

The results can be saved to a text file which can be imported into third party software (e.g. Microsoft Excel).

Note

If you click the currently selected button ([2], [3], [4], [5]) in the toolbox once more, the result dialog is reopened and you can again view or save the available results. The values remain stored even after the toolbox is closed, but will be deleted when you exit the application.

4. Fitted objects (spheres or planes) are displayed in the 3D viewer. Their colors help to identify their correspondences to the first fit (red) or the second fit (blue).
These objects are removed when the toolbox is deactivated or closed, or when switching from sphere fit to plane fit (or vice versa).

5. If you want to check a second sphere or plane, select the correct icon in the toolbox (for spheres normally [3], for planes normally [5]) and repeat step 2.
On completion of a calculation for spheres, a line connecting the two sphere centers in the

3D viewer displays the determined distance.

Repeating a calculation will overwrite existing results.

6. The icon **7** clears all results of the sphere and plane fits as well as the objects created in the 3D viewer.

7. By default, the lasso **8** is selected for creating the set of points for the fit. However, it is also possible to create single points that will be used as the basis for the calculation. To do so, switch to point mode **7**.

You can now select points by left-clicking on the data. These points are displayed as small spheres. As soon as at least four valid points have been selected, a preview object (sphere or plane) is created and displayed in the 3D viewer. Every time you add a new point, the preview is recalculated to make the changes visible. The final calculation is carried out when you press <Enter> or click **6** in the toolbox.

8.4 Cross Sections

When the cross sections toolbox is active, the following tools for creating cross sections are available:
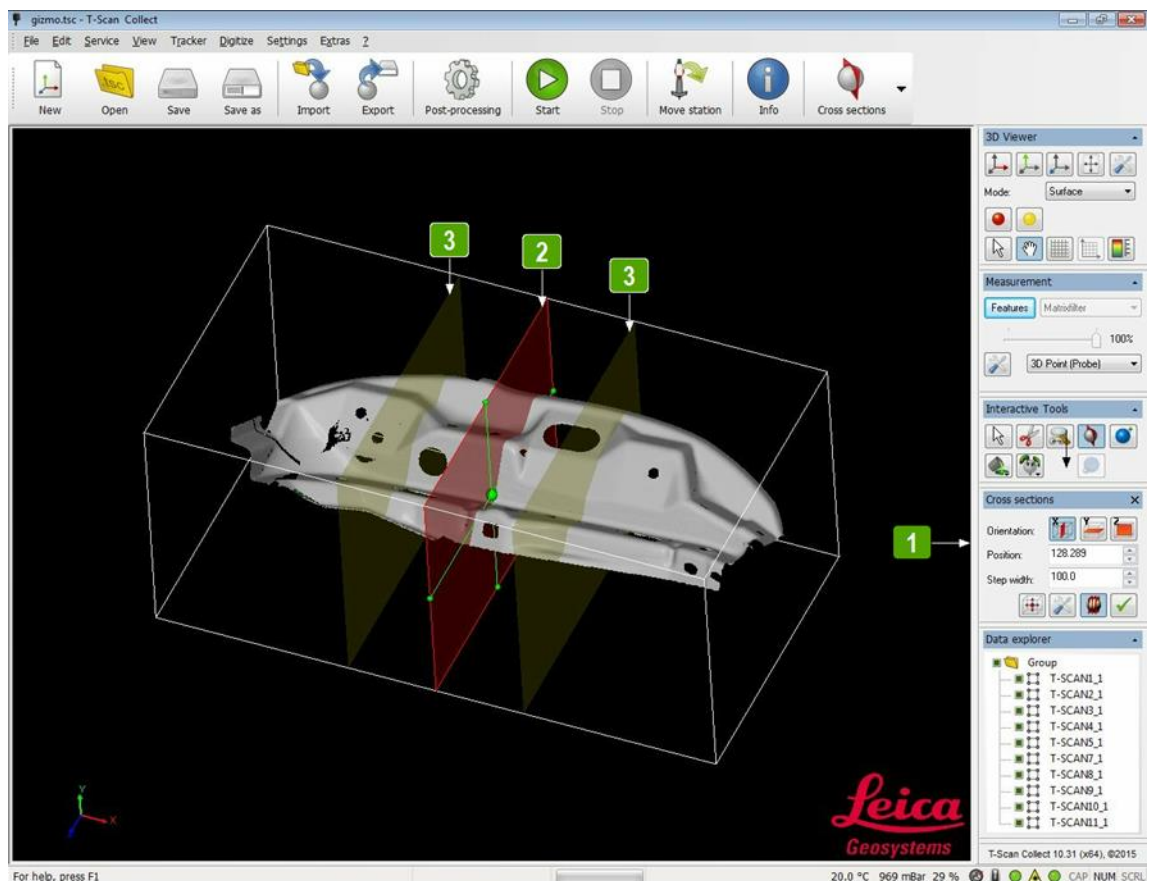
The cross sections toolbox **1** provides access to manipulation functionality for the cross section plane **2**. The cross section plane and, if activated, the auxiliary planes **3** for multiple sections, are visualized.
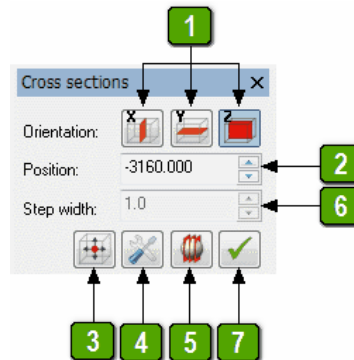
1 Selection of the cross section plane direction (along the coordinate axes)

2 Position of the cross section plane along the selected coordinate axis

3 Center the cross section plane in the data set

4 Open the dialog with the cross section settings; see chapter 17.1

5 Create multiple sections. If this mode is active, the step width field **6** defines the distance between multiple sections. Two auxiliary planes are displayed at the specified distance to the main plane in the 3D viewer, providing a preview of the settings.

7 Calculate and display cross sections

Note

The following display settings will be used:

- o *Red, semitransparent: cross section plane*

- o *White: bounding box*

- o *Green: center lines of the cross section plane as well as the start and end points and the intersection point of these lines.*

8.5 Create Reference Points

With this tool, you can use sphere fits to create reference points from measurement data. The following toolbox appears:
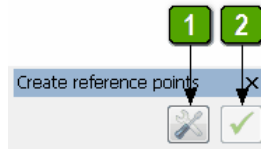
1. Before creating a reference point, it is advisable to review the settings. Click on the *Settings* button **1** to open the dialog:
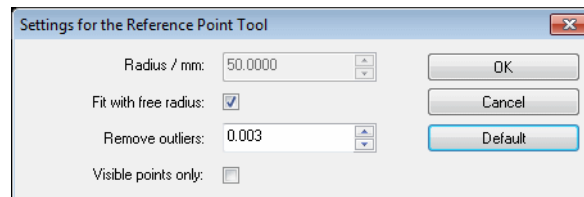
-   If *Fit with free radius* is disabled, the software will try to fit a sphere with the given radius into the data. This may lead to an error message if it is not possible to calculate a reasonable result, e.g. because the given radius differs significantly from the actual data.

-   In the *Remove outliers* field, you can specify a factor which defines how many percent of the worst fitting points will be removed in a sphere fit. The default value is 0.003, meaning 3 ‰.

-   The *Visible points only* checkbox is deselected by default because, in most cases, you will want calculate the fit for the entire sphere, including the rear side. This checkbox can be selected if required.

2. Now use the lasso in the 3D viewer to select the data you want to use for the sphere fit and press <Enter> or the toolbox icon .

3. The reference sphere is displayed as a blue sphere in the 3D viewer.

8.6 Fill Holes (Triangle Mesh)

This tool provides options for interactively or automatically filling holes in a triangle mesh. This functionality is available if a single triangle mesh is loaded. The following toolbox appears:
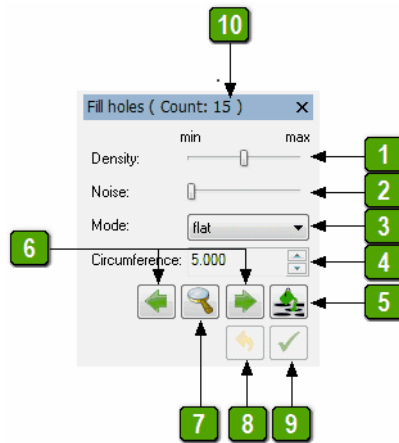
Holes can be selected in two different ways:

- Move the mouse cursor over the triangle mesh area you want to edit. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer. Holes in the data set in the area near the mouse cursor are detected automatically and shown with a yellow border. When you left-click a detected hole, the hole is filled and a preview of the result is displayed.

- Click the *Previous Hole* or *Next Hole* **6** icon to select one hole after the other. The 3D view can be automatically centered on the selected hole if icon **7** is activated.

With the *Density* **1** and *Noise* **2** sliders, you can adjust the data generated for the filled hole to optimally fit the characteristics of the surrounding measurement data. If multiple holes are selected and filled, these settings are applied to each of them.

The *Mode* **3** list box provides the following filling strategies:

- *flat* fills a hole as plane as possible; works very fast

- *curved* incorporates the orientations of the surrounding mesh into the calculations, delivering excellent results e.g. when filling holes in spheres; calculation takes longer

To accept the generated areas, press the *Apply* **9** icon or right-click in the 3D viewer. If you are not satisfied with the result, click *Undo* **8** to undo the last action or actions you made.

Holes can also be filled automatically. For this, enter the maximum *Circumference* **4** in mm and click **5**. All holes up to this size are filled automatically using the specified settings.

The title bar of the toolbox shows information **10** on how many holes have been detected in the triangle mesh.

8.7 Create Bridges (Triangle Mesh)

This tool provides the possibility to interactively create bridges over gaps in a triangle mesh. This functionality is available if a single triangle mesh is loaded. The following toolbox appears:
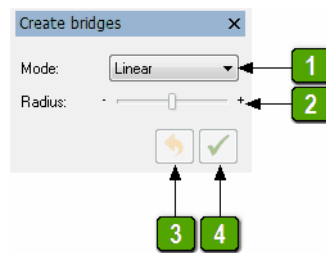
Move the mouse over the triangle mesh area you want to edit. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer.

To create bridges, move the mouse over the area to be edited until the area is highlighted in yellow. Click on it with the left mouse button to define the start edge for the bridge. Then select the end edge with the left mouse button. A preview bridge with the currently selected parameters is displayed between the two selected edges.

In the *Mode* list box [1], you can choose between the following bridge modes:

Linear
This mode creates a linear data bridge as the shortest connection between the selected edges.

Tangential
This mode creates a data bridge between the selected edges, taking the surface orientation of the adjacent triangles into account.

Corner
This mode creates two linear data bridges, one from each selected edge. The surface orientation of the adjacent triangles is taken into account. The edge that is formed where the two linear bridges meet can then be rounded with the *Radius* slider [2].

The selected mode can be changed even after you have selected the start and end edges. This way you can visually adjust the generated bridge to the surrounding area.

To accept the generated bridge, press the *Apply* [4] button or right-click in the 3D viewer. If you are not satisfied with the result, click *Undo* [3] to undo the last action or actions you made.

8.8   Smoothing (Triangle Mesh)

With this tool, you can interactively smooth areas in a triangle mesh. This functionality is available if a single triangle mesh is loaded. The following toolbox appears:
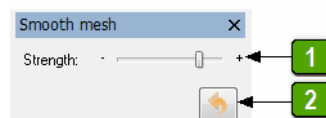
Move the mouse over the triangle mesh area you want to edit. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer.

A circle is displayed around the mouse cursor showing the radius of effect of the tool ("brush size"). It can be resized with the mouse wheel or by drag and drop with the middle mouse button.

To smooth an area, left-click on the beginning of the area and keep the mouse button pressed. Now drag the cursor over the desired area on the object. The data in this area is smoothed while you move the cursor. Releasing the mouse button stops the smoothing process.

The smoothing strength can be adjusted with the *Strength* slider **1**.

If you are not satisfied with the result, click *Undo* **2** to undo the last action or actions you made.

Note

Take care to only smooth surfaces that face the user in the 3D viewer. Smoothing transitions from front side to back side (e.g. on spheres), or on faces which can only be seen under a very small angle, can lead to artifacts.

8.9 Decimate Mesh (Triangle Mesh)

This tool is used to interactively decimate a triangle mesh. This functionality is available if a single triangle mesh is loaded. The following toolbox appears:
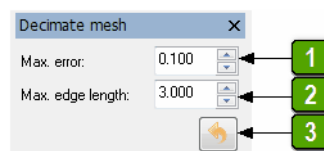
Move the mouse over the triangle mesh area you want to decimate. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer.

A circle is displayed around the mouse cursor, showing the radius of effect of the tool ("brush size"). It can be resized with the mouse wheel or by drag and drop with the middle mouse button.

To decimate an area, left-click on the beginning of the area and keep the mouse button pressed. Now drag the cursor over the object to mark the area you want to decimate. When you release the left mouse button, the marked area is automatically decimated.

The *Max. error* **1** parameter defines the maximum permissible deviation in millimeters; the *Max. edge length* **2** parameter defines the size of the resulting triangles in millimeters.

If you are not satisfied with the result, click *Undo* **3** to undo the last action or actions you made.

8.10 Sharpen Edges (Triangle Mesh)

This tool provides the possibility to sharpen edges of triangle meshes interactively. This functionality is available if a single triangle mesh is loaded. The following toolbox appears:

Move the mouse over the triangle mesh area you want to edit. If view mode is currently active, change to tool mode by pressing <Space Bar> in the 3D viewer.

A circle is displayed around the mouse cursor, showing the radius of effect of the tool ("brush size"). It can be resized with the mouse wheel or by drag and drop with the middle mouse button.
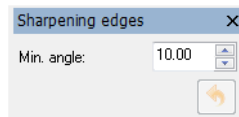
To sharpen edges, left-click on the beginning of the area and keep the mouse button pressed. Now drag the cursor over the object to mark the area you want to modify. When you release the left mouse button, the edges in the marked area are automatically sharpened.
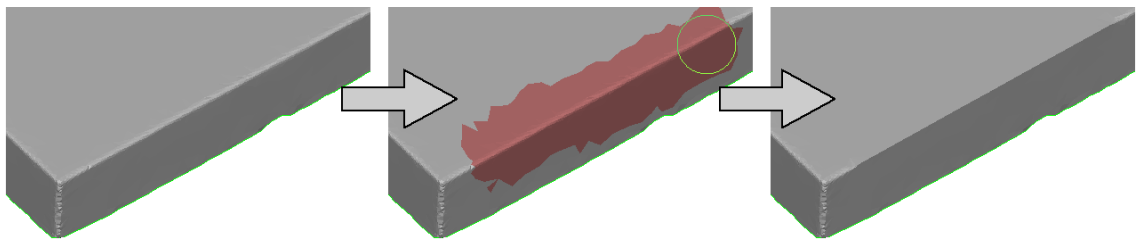
The *Min. Angle* parameter defines the minimum angle an edge must have in order to be sharpened. A theoretical value of 0 means that all data is modified, a value of 180 means that no data is modified.

If you are not satisfied with the result, click *Undo* to undo the last action or actions you made.

8.11 Alignment Calibration

How to use this toolbox is described in detail in chapter 0 Calibration.

9 Calibration

Environmental influences (such as temperature changes or transport of the system) may lead to a gradual decrease in data quality.

To increase data quality, T-Scan Collect offers the alignment calibration feature which the user can – and should – perform on site if:

- an upcoming measurement requires highest accuracy

- local accuracy has decreased. This can manifest itself, for example, in poor statistics when calculating a sphere fit

- gaps occur in the data

- wavy data is measured on actually planar surfaces

9.1 Calibrate measurement range

9.1.1 Preparation

In order to perform the measurement range calibration, you need a planar surface with diffuse reflection.

Not suited are surfaces containing scratches, bumps, dents or similar faults. In general, a table board should suffice the requirements. However, it is recommended to use a specified plane which is optionally available.

9.1.2 Procedure

Open the Measurement range toolbox. The measuring mode *Alignment* will automatically be activated. When opening the toolbox the status – symbolized by the traffic light – will be set to red by default, indicating that no action has been performed yet.

To perform the calibration, two measurements have to be made. Take care to account for the following:

- The distance between scanner and plane has to be as constant as possible

- The laser should hit the plane perpendicularly

- The markers of the used scanner face should point as directly as possible towards the camera

- The orientation of the scanner must not change during measurement

- Each measurement must contain at least 5000 points

- For both measurements, the same scanner face must be used

- Both measurements must be made on the same area of the plane
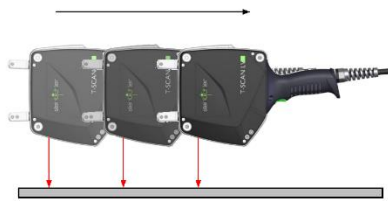
First measurement: keep scanner distance constant (Close End*)
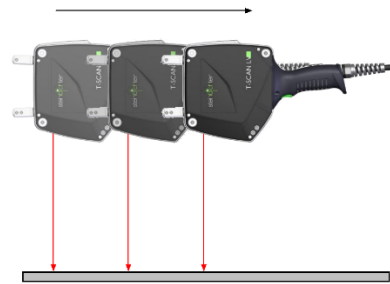
Second measurement: keep scanner distance constant (Far End*)

*: For the definitions of these terms, see graphics in chapter 17.12.
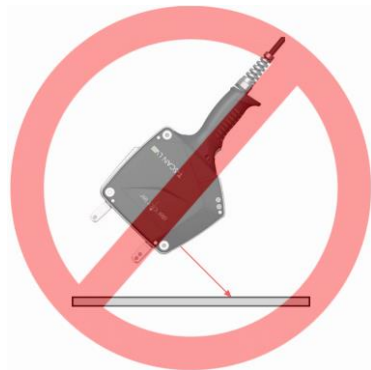
Avoid measurements where the laser does not hit the surface perpendicularly

Avoid measurements where the laser does not hit the surface perpendicularly

Avoid tilting the scanner, so the markers do not point directly towards the camera
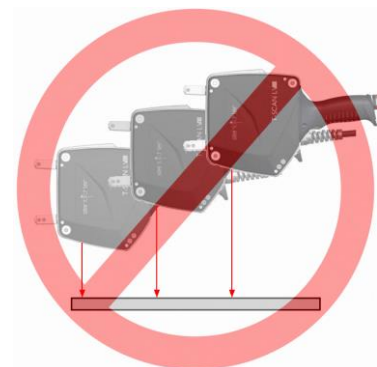
Avoid changing the distance of the scanner to the surface

The above sketches are drawn from the camera's perspective.

Now measure the plane one time using the Close End of the scanner and another time using the Far End of the scanner ("Calibration measurement").

In the 3D viewer, select the two data sets using the lasso. Take care not to select points which do not lie on the plane. If other data (e. g. from previous measurements) is located at the same area, hide or delete it in the Data Explorer toolbox prior to using the lasso.
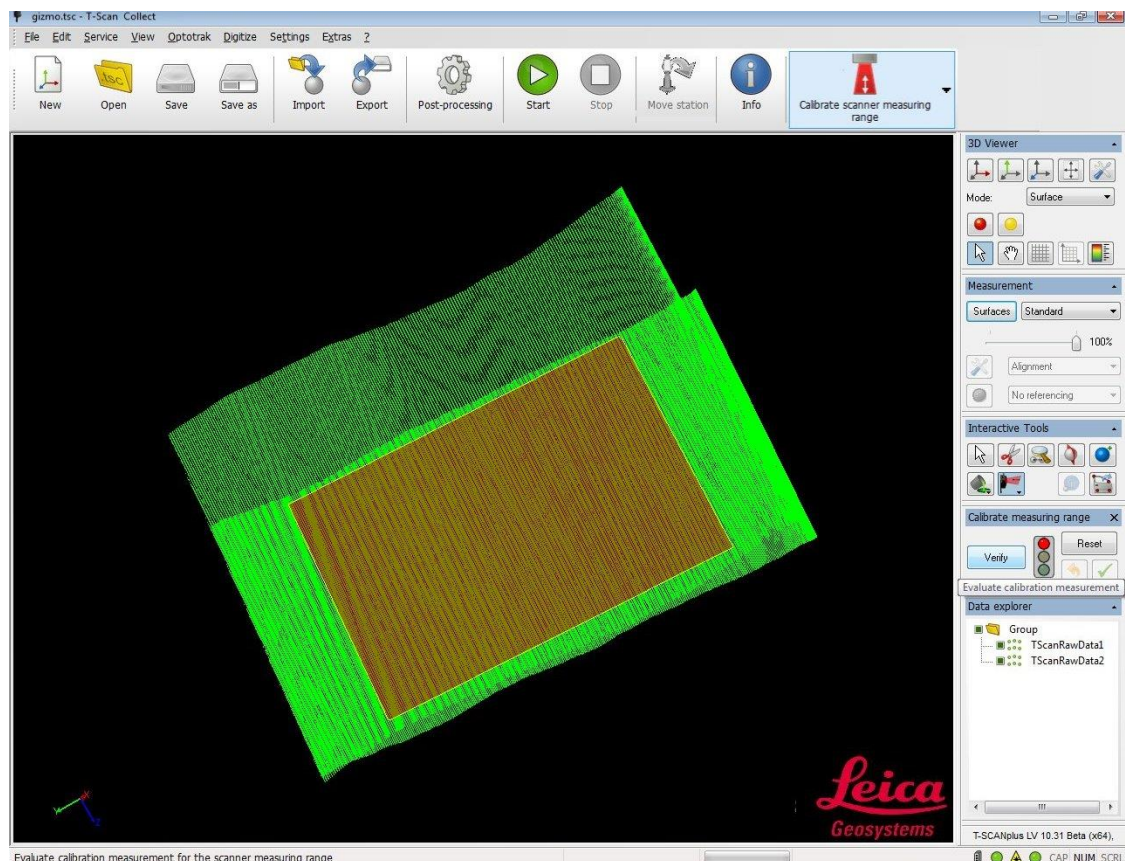
Press *Verify*. If the data does not suffice the requirements an adequate message box will open. In this case, repeat the measurement. Otherwise the status indicator will show you the status of the measurement range calibration:

- Green state: The scanner is calibrated optimally. No further steps are necessary and you may close the toolbox.

- Yellow state: The scanner calibration has been optimized using the measured data. To make sure that the optimization improved the calibration it is necessary to measure the plane again (in the same way as described above) one time using the Close End of the scanner and another time

- Red state: The scanner calibration has not been started yet or the *Undo* button was pressed.

If the Verification measurement confirms an improvement, the status indicator will switch to green. In this case, save the new calibration by pressing *Apply*.

Otherwise the status indicator will remain yellow. You now have the possibility of either repeating the verification measurement or undoing the calibration measurement by pressing *Undo*.

If you are not able to reach the green status, you can close the toolbox. All changes will be undone after a confirmation message box is displayed.

When closing the toolbox, the measuring mode which has been active before will be restored.

9.2 Alignment Calibration

9.2.1 Requirements for an Alignment Sphere
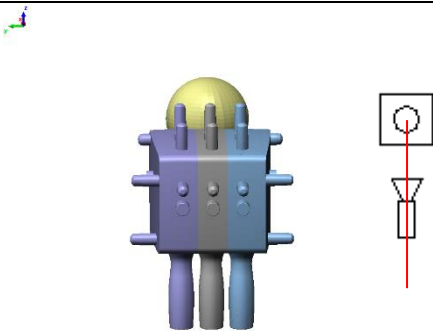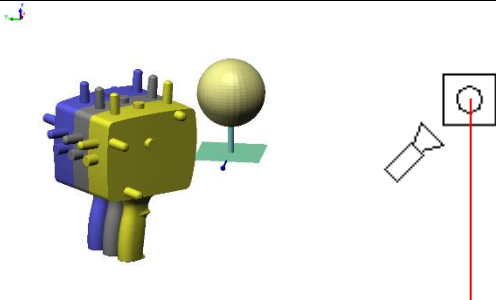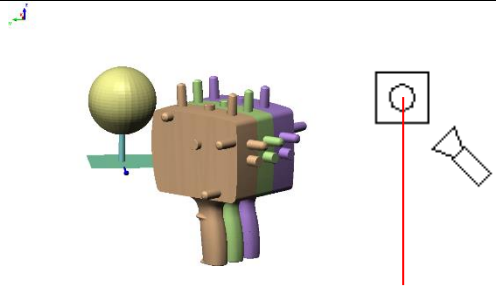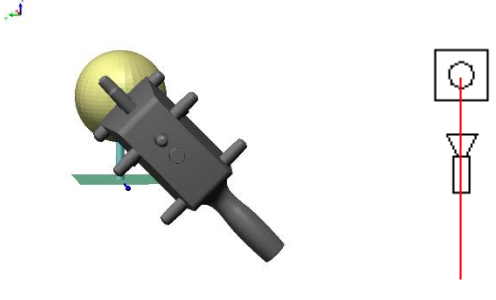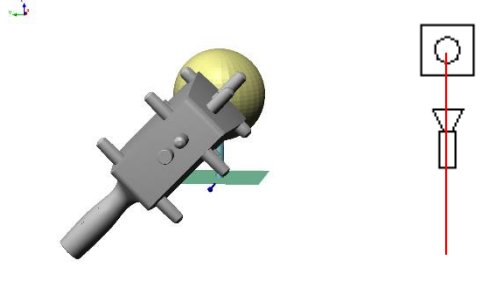
To do an alignment calibration, a high-precision calibration sphere is needed. It is recommended to use a sphere made of ceramic, steel, aluminum or other suitable materials with a diffusely reflecting surface (non-volumetric scattering), specified by Leica Geosystems. These spheres are between 40 mm and 105 mm in diameter. The form error and the radius of such a sphere are known with an uncertainty of less than 20 µm. A calibration certificate is shipped with these spheres (e.g. Leica Reference Sphere – Article number: 576190).

9.2.2 Measurement Strategy

The Alignment has to be done for each face separately. For the left, top and right face there are 5 main positions, for the back face there are 4 main positions:

Left face

Top face

Back face

Right face
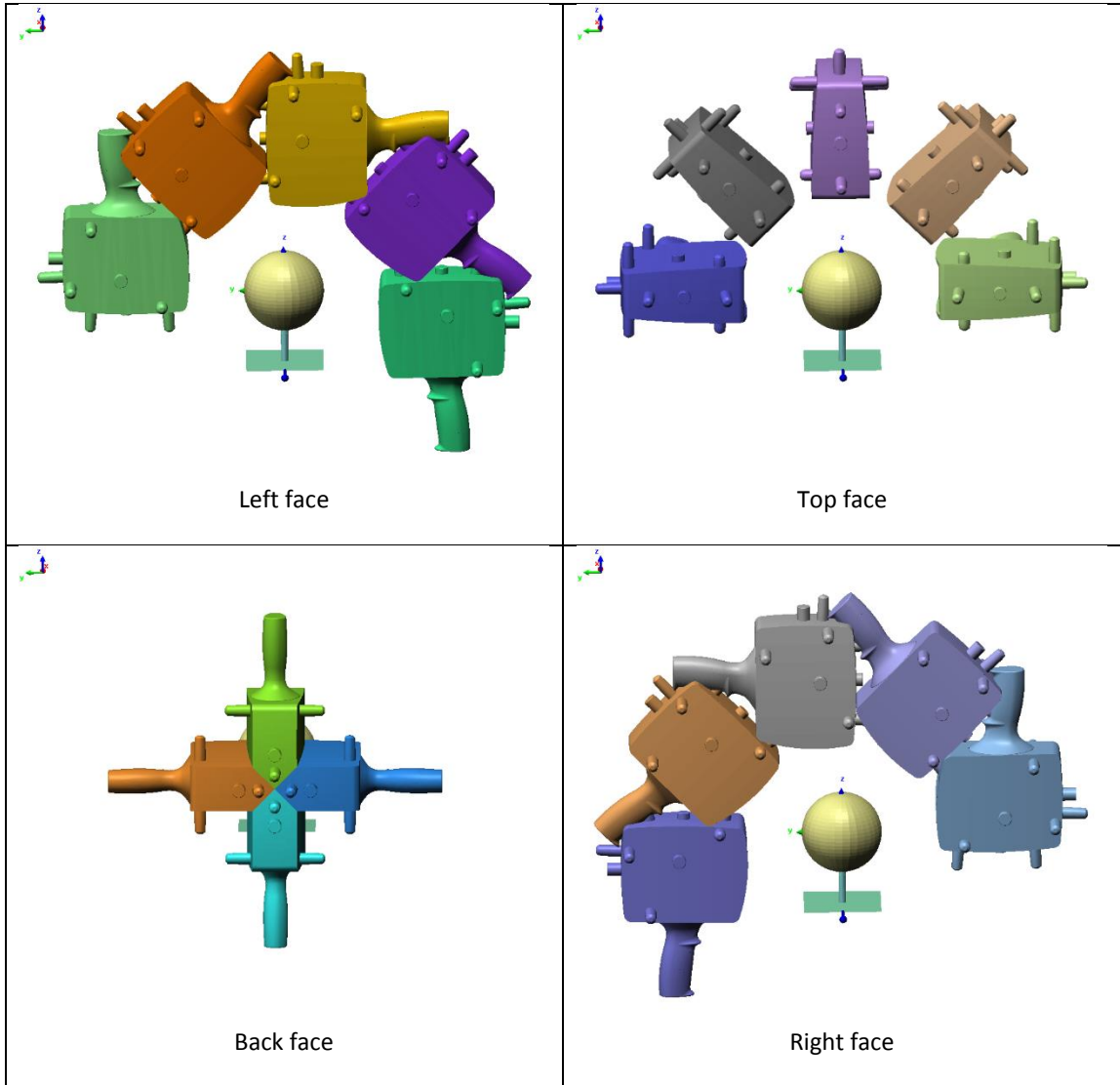
For each main position 11 scans need to be acquired:

Center position
3 scans: Middle, left, right

Rotated to the left
3 scans: Middle, left, right

Rotated to the right
3 scans: Middle, left, right

Centered, diagonal
1 scan

Centered, diagonal
1 scan

9.2.3 Step-by-Step Guide

1. Open the Alignment toolbox.
   The *Alignment* measurement mode is activated automatically.
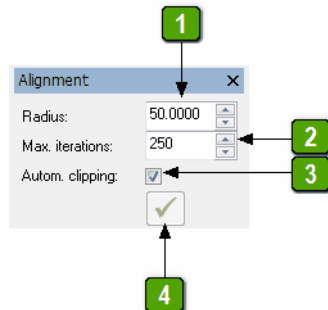
2. Press *Start*.

3. Measure the alignment sphere as described in chapter 9.2.3. Data measured in *Alignment* mode is displayed in green by default to distinguish it from other data.

4. Press *Stop*.

5. If data was acquired that does not belong to the sphere (e.g. mounting, table, ...) you can manually remove this data (see "Interactive Clipping" toolbox) or have this data removed automatically. For this, activate *Autom. clipping* [3] in the toolbox.

Note

Automatic clipping might lead to unexpected results if the initial data is inappropriate. For example, if the majority of the data consists of the mounting or table, etc. (but not the sphere) or if the scanner is way out of calibration, data fragments might remain which normally should be removed. In this case, the data needs to be clipped manually.

6. You need to measure at least 400 000 points in order to continue. It is recommended to measure between 400 000 und 1 000 000 points.

7. In the toolbox, enter the radius of the sphere [1] given in the calibration certificate and the maximum number of iterations [2]. The recommended value is 250; however, the algorithm often takes fewer iterations.

8. Press *Start calculation* [4].

9. The calculation may take several minutes, depending on the performance of the computer and the initial calibration state.

10. After this calculation, a sphere fit is executed automatically.

11. A dialog appears, showing the results of the calibration and of the sphere fit:
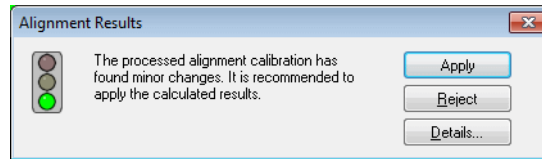
Check the color of the traffic light symbol and follow the displayed advice.

If a red traffic light appears, please check whether a correct radius value was entered. A yellow light indicates a significant improvement in comparison to the previous state. It is recommended to repeat the calibration in order to verify the result. A green state means that some minor changes have been identified in the calibration, but no significant improvement, i.e. the scanner is either optimally calibrated or this calibration method cannot achieve further improvement.

The quality of the calibration result can only be assessed from the Before and After information. To view this information, click *Details…*
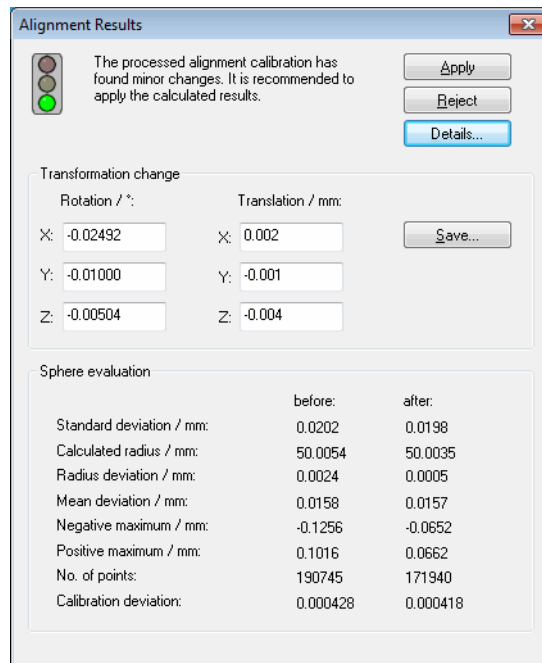
In the details view, you can save the calibration results to a file.

If you cannot achieve the green state, even after repeatedly measuring the sphere, this may be for the following reasons:

- Bad measurement strategy; measure as described in chapter 9.2.3.

- Setup changed, vibrations, instable setup; stabilize the setup

- Scanner out of calibration; contact service personnel

12. When you click *Apply*, the result of the calculation is applied to the actual calibration.

13. To verify the alignment, repeat the alignment measurement from step 3 as described above.

14. When you close the alignment toolbox, the measurement mode that was active before is restored.

10 Working with Files

All file-related commands are provided in the *File* menu. The available commands are described in detail below.

10.1 Create New Project

To create a new project, click *New* in the standard toolbar or choose *File → New*. If new or changed data is still in memory, a dialog appears asking if you want to save the changes. If you press *No*, all data will be cleared from memory; if you choose *Yes* they will be saved to a file.

If an online transformation was created (see chapters 11.1 and 15.9), a dialog appears asking if you want to keep or discard the current transformation.

10.2 Open Existing Project

To open a saved project (*.TSC* file), click *Open* in the standard toolbar or choose *File → Open…*.

A file selection dialog appears where you can select the file path and the name of the project.

10.3 Save Project

To save the current project under the given project name, click *Save* in the standard toolbar or choose *File → Save…*.

If you have not yet assigned a project name, the *Save as…* function described below is invoked automatically.

10.4 Save Project Under New Name

To save the current project as a *.TSC* file under a new name, click *Save as* in the standard toolbar or choose *File → Save as…*.

A file selection dialog appears where you can enter the file path and the name of the project. You do not have to add the file extension *.TSC* explicitly; it is automatically added by the software.

Example: Generate a project $...\backslash Module\backslash Module.tsc$ by changing to (or creating) the file path '..\Module' and entering 'Module' in the *File Name* text field.

10.5 Import Data

You can import data in different formats into your current project by using the *File* menu or the data explorer.

10.5.1 <u>Using the File Menu</u>

If you want to add data to a specific group, you first have to select the group or a data set from that group. Then click *Import* in the standard toolbar or choose *File* → *Import....*

In the file selection dialog, select the file path and the file(s) to be imported, then click the *Open* button. The selected files are imported and inserted into the selected group.

10.5.2 <u>Using the Data Explorer</u>

In the data explorer, right-click on a group or a data set within a group. Choose *Import...* from the menu that opens. Then continue as described in chapter 10.5.1.

Note

To facilitate finding the file(s) to be imported, select the appropriate file type in Files of type *first. The file selection dialog will then only show files of this type.*

You can import several files (of the same file type) in a single step by multiple selection in the selection dialog.

10.5.3 <u>Using Drag and Drop</u>

Files known to the software can also be imported by drag and drop to the 3D viewer or the data explorer toolbox. When importing to the data explorer, the data will be placed in the group over which the mouse button was released.

10.6 Export Data

You can export the loaded data sets in different formats. This is especially useful if you want to process the data in a different application.

You can export data using the *File* menu or the data explorer.

10.6.1 <u>Using the File Menu</u>

This function is available if measurement data is provided in memory. To export 3D data, press *Export* in the standard toolbar or choose *File* → *Export....*

In the file selection dialog, select the file path on which the created file is to be saved. Enter a name and select the file format to be generated.

Set the other options as required and then click the *Save* button.

Note

Please keep in mind that, with some file types, not all the data can be exported in the selected format. The following table shows the data supported by the individual file types.

	TSC	AC	ASC	TXT	IGS106	IGS110	IGS116	SBP	SBT	STL	VDA	LST	CGK
Hierarchy (Data Explorer)	🟢	🔴	🔴	🔴	🔴	🔴	🔴	🔴	🔴	🔴	🔴	🔴	🟢
Unstructured Pointset	🟢	🟢	🟢	🟢	🟢	🔴	🟢	🟢	🔴	🔴	❸	🔴	🟢
LineScanSet	🟢	🟢	❶	❶	❶	🔴	❶	❶	🔴	🔴	❷	🔴	🟢
RangeImage	🟢	🟢	❶	❶	❶	🔴	❶	❶	🔴	🔴	❷	🔴	🟢
PolyLine	🟢	🟢	❶	❶	❶	❷	❶	❶	🔴	🔴	🟢	🔴	🟢
PolyLineSet	🟢	🟢	❶	❶	❶	🟢	❶	❶	🔴	🔴	🟢	🔴	🟢
Point	🟢	🟢	❶	❶	❶	🔴	❶	❶	🔴	🔴	🟢	🟢	🟢
Triangle Mesh	🟢	❶	❶	❶	❶	🔴	❶	❶	🟢	🟢	❸	🔴	🟢
Alignment Data	🟢	❶	❶	❶	❶	🔴	❶	❶	🔴	🔴	❸	🔴	🟢

🟢: Export fully supported

🔴: Export not supported

❶: Data will be exported as Unstructured Pointset

❷: Data will be exported as PolyLineSet

❸: Data will be exported as PolyLine

10.6.2 Using the Data Explorer

In the data explorer, select the data sets you want to export. Right-click on one of the selected data sets and select *Export…* in the menu. Then continue as described in Chapter 10.6.1.

11 Processing of Data Sets

All processing functions are provided in the *Edit* menu.

11.1 Transform Data

To transform all loaded data sets, choose *Edit → Transform data*. The following submenu appears:
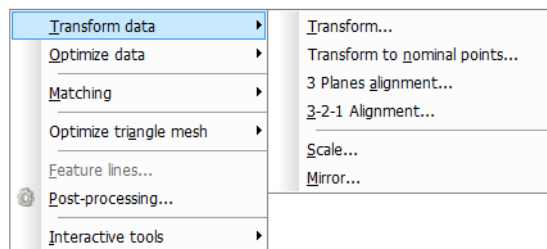
11.1.1 Transform

Choose *Edit → Transform data → Transform*. The following dialog appears:
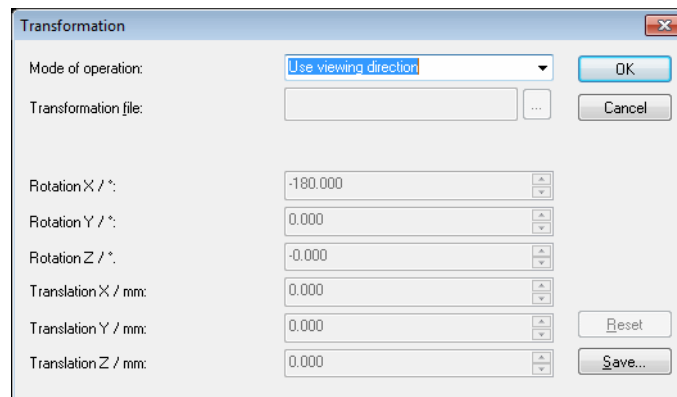
The different entries have the following meanings:

Mode of operation

With this parameter, you can choose how to determine the transformation matrix that will be applied to all data sets. The following modes are available:

- *Online-Transformation*
  In this mode, the currently used online transformation is displayed. By clicking the *Reset* button, you can undo the displayed online transformation.

- *Use transformation file*

  In this mode, you need to specify a transformation file. The rotation angles and the translation values are stored in this file.

- *Use given values*

  In this mode, the rotation angles and the translation values to be applied need to be entered in the input fields.

- *Use viewing direction*

  In this mode, you can rotate the data sets in the 3D viewer before calling the function. The data sets can be rotated in such a way that the viewing direction represents the future positive Z axis. The X and Y axes will then be parallel to the monitor edges. During execution of the function, a corresponding transformation matrix is calculated and applied.

Transformation file

With this parameter, you determine the transformation file to be used. You can also determine the file interactively in a file selection dialog by using the ... button next to the file name. The field is only active if the mode of operation is set to *Use transformation file*.

Rotation, Translation

In these fields, you can define the rotation angles and the translation values from which the transformation matrix is generated. The fields are only active if the Mode of operation is set to *Use given values*.

Note

The ⌊⌐ symbol in the status bar indicates that an online transformation is active.

Please note that repeated execution of this function may lead to unexpected results!

11.1.2 Transform to Nominal Points

This wizard helps you to transform your data to the object's coordinate system if nominal points (e. g. from a CAD file) are available for the data set.

11.1.2.1 Edit the nominal points

Choose *Edit → Transform data → Transform to nominal points*. A wizard will appear to guide you through the process. You can cancel the wizard at any time by clicking the *Cancel* button.

With the wizard, you can load, save and manually edit nominal point lists.

The following functions are available:

Input box: ID
This input box defines the ID that will be assigned to the next point to be created.

Input box: X | Y | Z
These input boxes define the X, Y and Z components of a nominal point. As soon as one of the input boxes loses the focus (by pressing <Tab> or clicking in a different area), a nominal point with the respective ID is either created and added to the list, or an already existing point is modified.
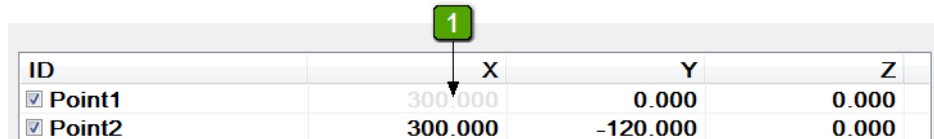
Nominal scaling (1:N)
The nominal scaling helps you if the measurement object and the nominal points are scaled differently (e. g. if the scale of the object is 1:4 and that of the nominal points is 1:1). The nominal data is automatically scaled to the correct size. For example, if the scale of your model is 1:4, enter a value of 4; if the scale is 2:1, enter 0.5.

Columns: ID | X | Y | Z
This table shows the nominal points. By using the checkbox in the ID column, you can specify whether a point will be used for calculating the transformation. If you deselect the checkbox, all three coordinates of the point are grayed out and the point will not be used in the calculation.

With the right mouse button, you can exclude individual coordinate components of a point from the calculation. When you right-click the component of a point, its state changes from active to passive, and vice versa. Use this possibility if not all coordinates of a nominal point are known or are to be used.

ID	X	Y	Z
☑ Point1	300.000	0.000	0.000
☑ Point2	300.000	-120.000	0.000

1 Excluding the X coordinate of a nominal point

When you click an ID in the first column, the point is selected and the ID is displayed with a blue background. You can use the input boxes to modify the point you last selected.

Delete
Use this button to delete the currently selected points from the list.

Note

You can also delete selected points by pressing <Del>.

Delete All
Use this button to delete all points in the list.

Load
Opens a file selection dialog to load a point list file.

Save
Opens a file selection dialog to save the point list file.

The *Next* button is available as soon as enough points or components have been selected.

Note

If you press Next without having excluded any coordinate components of a nominal point, you will not be able to exclude them later. If this has become necessary, however, please close the wizard and re-open it (save your nominal points beforehand, if required).

11.1.2.2 Measure the actual points

On this page, you create the corresponding measurement points for the points you selected on the *Edit the nominal points* page.

The next steps in the wizard depend on whether you have excluded coordinate components of the nominal point from the calculation.

If no coordinate components were excluded, the correspondences between the measured points and the nominal points are determined automatically.

If coordinate components were excluded, you need to define the correspondences manually through the ID, i.e. by measuring the points in the correct order.

The following functions are available:

Column: ID
If coordinate components were excluded, the IDs of previously measured points are displayed. When measuring points, please make sure the correct ID is selected.

If no coordinate components were excluded, the ID column is initially empty. The IDs will be computed automatically in the next step when the transformation is calculated.

Columns: X | Y | Z
Displays the coordinates of measured points. If coordinate components were excluded, points that have not yet been measured are marked with a "---" symbol.

From Scene
If no coordinate components were excluded, all points from the data explorer are entered in the table. If coordinate components were excluded, only those points are entered that match the name of a loaded point.

Measure

When you click this button, the point you last selected in the table is measured with the T-Probe. If the measurement succeeded, the next point in the list is automatically selected.

Note

You can also start a measurement by clicking the right mouse button or pressing <M>.

Delete

Use this button to delete the coordinates of the currently selected points from the list.

Note

You can also delete the coordinates of selected points by pressing <Del>.

Delete all

Use this button to delete the coordinates of all points in the list.

Load

Opens a file selection dialog to load a point list file. If coordinate components were excluded, only those points are imported that match the given IDs. The other points are ignored.

Save

Opens a file selection dialog to save the point list file.

The *Next* button is available as soon as enough measurement points have been acquired.

11.1.2.3 Transformation results

This page shows the calculated best fit transformation as well as statistics for the best fit.

Columns: ID | X | Y | Z

The first column of the table shows the ID, the other columns show the X, Y and Z coordinates.

Three rows are displayed for each ID:

- Nom: This row shows the nominal values as entered on the *Edit the nominal points* page of the wizard (if applicable, scaled by the nominal scaling factor).

- Act: This row shows the measured points after they have been transformed to the nominal points.

- Dev: This row is displayed in red on a gray background and shows the deviation between actual point and nominal point.

Coordinates that have been grayed out on the *Edit the nominal points* page will be excluded from the calculation and are therefore displayed in gray.

Rotation / Translation

These fields display the calculated rotation (in degrees) and translation (in millimeters).

Statistics

The statistics provide information on the best fit. The first line shows the maximum 3D deviation and the ID for which the deviation was determined. The second line indicates the minimum 3D deviation and the corresponding ID, followed by the mean 3D deviation and the 3D standard deviation.

By pressing the *Back* button, you can access the previous pages of the wizard and make modifications. If no coordinate components were excluded, the corresponding IDs that have been calculated automatically are now displayed as well.

For example, you can exclude the ID that shows the maximum 3D deviation from the calculation by returning to the *Edit the nominal points* page and deselecting that specific point. Alternatively, you can also measure the point again.

When you click the *Finish* button, the measured data is transformed to its origin with the inverse of the current online transformation. The newly calculated transformation is then applied. In this way, you can repeatedly execute the *Transform to nominal points* wizard.

Log file
Use this button to save the results to a log file.

Note

When you apply the calculated transformation, you can visualize the nominal points in the 3D viewer. Click the Nominal points *icon in the 3D viewer toolbox.*

The *symbol in the status bar indicates that an online transformation is active.*

11.1.3 3 Planes Alignment
To align all loaded data sets by using reference planes, select *Edit → Transform data → 3 planes alignment*. The following dialog appears:
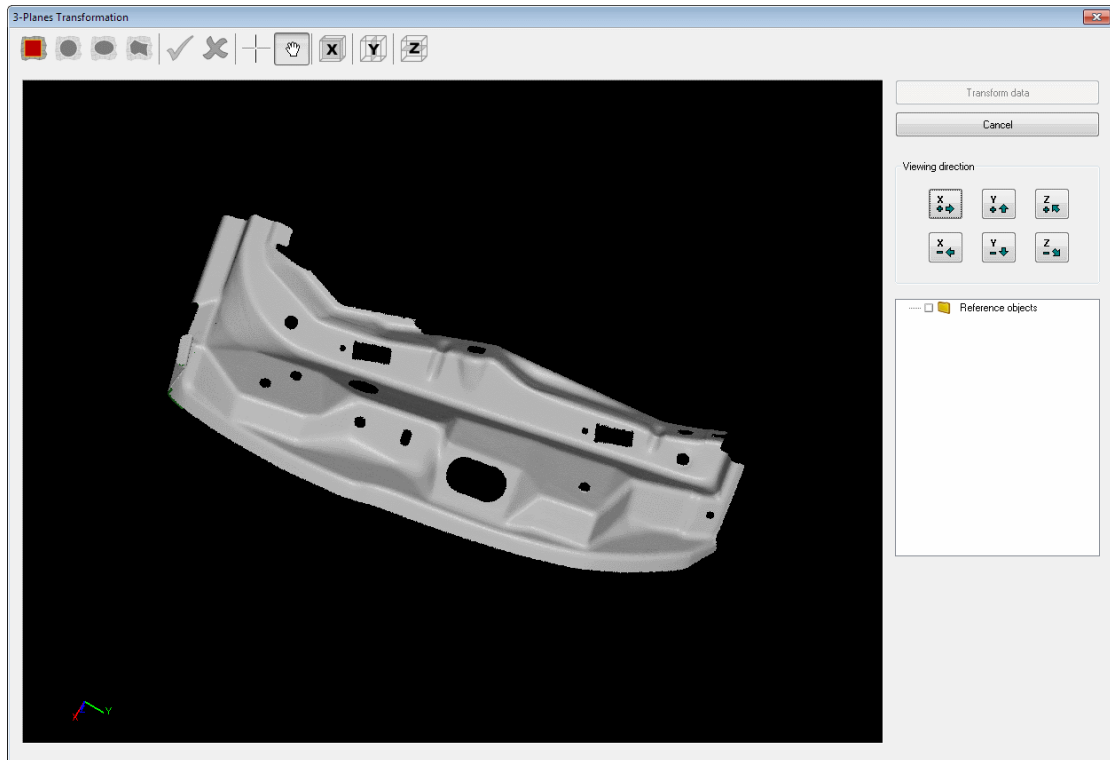
Use the provided tools to select all reference planes one after the other and assign the respective point data to paraxial nominal planes.

The following drawing tools are available:
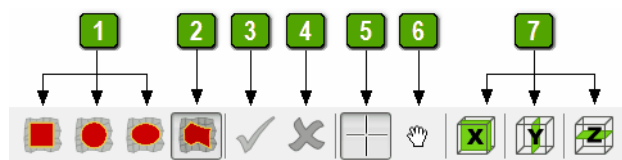
1 Shape of selection: rectangle, circle, ellipse

2 Shape of selection: polygon

3 Confirm the shape created with tool **1**

4 Discard an unconfirmed shape created with tool **1**

5 Drawing mode

6 Viewing mode

7 Assign the object data of the masked area to a nominal plane perpendicular to the X, Y or Z axis

As soon as you click one of the tools **7**, you need to define the position of the plane in the following dialog:
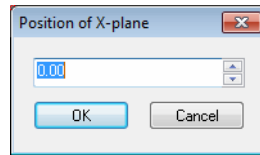
When you click *OK*, a plane fit is calculated based on the selected points, and plausibility checks are performed for planarity of data, angles to already created planes, etc. After successful completion of the best fit, the plane object is added to the list of reference objects. The masked points of the object are displayed in green.

As soon as at least one object has been assigned to each main plane, the *Transform data* button is available. When you click the button, the data is transformed and the alignment is completed.

Note

When using one of the tools **1***, you can add areas in the following way:*

1. *Move the cursor to the top left corner of the area to be drawn.*

2. *Press and hold the left mouse button.*

3. *Move the cursor to the bottom right until the object has the required size.*

4. *When you release the left mouse button, the object is surrounded by a border that allows manipulating the object: When the cursor is inside the object, you can move the whole object; when the cursor is above one of the small squares, you can move the corresponding border(s). To do so, press and hold the left mouse button, move the element to the required position and release the mouse button.*

5. *To accept the object, use tool* **3** *or double-click outside the drawn object with the left mouse button. To delete the object, use tool* **4***.*

When using tool **2***, proceed as follows:*

1. *Move the cursor to the starting point of the polygonal chain and click the left mouse button.*

2. *When you move the cursor, you can see a line between the last point and the cursor. Each time you click the left mouse button, you add a new point to the polygonal chain. Repeat this step until all points have been added.*

3. *To end the polygonal chain, double-click the left mouse button. Please keep in mind that the point that is double-clicked is also added to the polygonal chain.*

Basically you can assign any number of reference planes to a main axis plane – also at different axis positions. You need to make sure, however, that you only select the data of a single plane and a single position in each step. Also be careful that no data from other areas is accidentally selected as well. This mainly happens if data on the back of the object is hidden by front areas.

*The* ⌐ *symbol in the status bar indicates that an online transformation is active.*

11.1.4 <u>3-2-1 Alignment</u>

To do a 3-2-1 alignment for all loaded 3D data sets, choose *Edit →Transform data → 3-2-1 Alignment.* The following dialog appears:
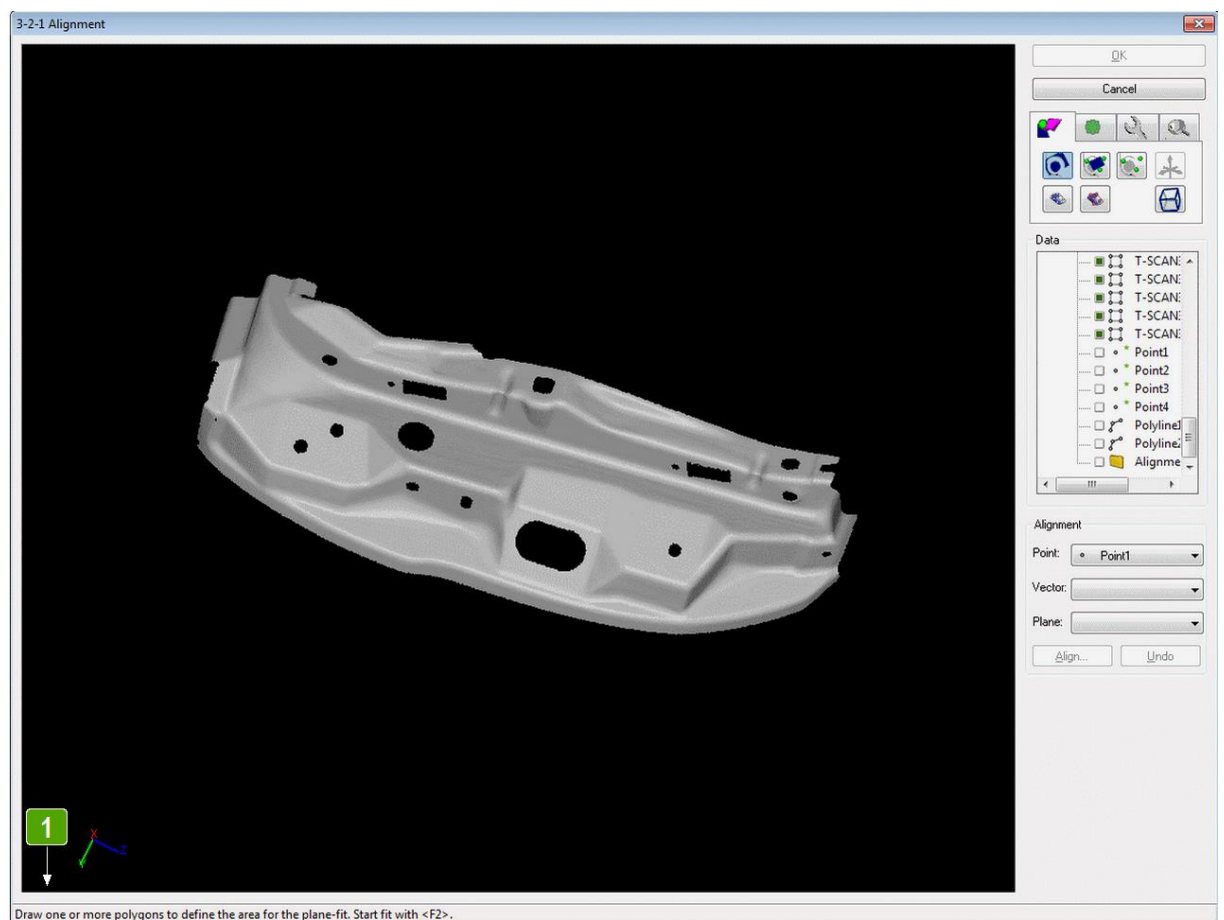
In the left part of the window, the loaded 3D data is displayed. Below the 3D viewer, the status bar 1 indicates how to use the individual tools. On the right, you will find the data explorer with the project. Above it, there are several tabs with the different tools for orientation of the 3D data. During 3-2-1 alignment, alignment elements (planes, vectors, points) are created and saved in a separate group of the 3D scene. These elements will be deleted when you leave this dialog.

The tools are grouped as follows:

Functions and tools for the extraction of alignment elements (planes, vectors, points) from 3D data.

Functions and tools for the calculation of alignment elements from existing alignment elements.

Functions and tools for the manipulation of alignment elements.

Functions and tools for 3D viewer control.

Functions for the Generation of Alignment Elements from 3D Data

Tools for the generation of a best fit plane for the selected area in the 3D data. Activate this tool and use the mouse to draw one or more polygons in the 3D data. The polygons define the area for the calculation of the best fit plane. A polygon point is defined by left-click. Double-clicking the left mouse button ends a polygon. To start the calculation of the best fit plane, press the <F2> key.

Tool for the generation of a best fit plane for selected points of the 3D data. Activate this tool and use the mouse to define points in the 3D data. These points will be used for the calculation of the best fit plane. A point is defined by left-click. To start the calculation of the best fit plane, press the <F2> key.

Function for the generation of points from the 3D data. Activate this tool and use the mouse to define points in the 3D data. A point is defined by left-click.

Tool for the calculation of the main axes of inertia for the selected area in the 3D data. Activate this tool and use the mouse to draw one or more polygons in the 3D data. The polygons define the area for the calculation of the main axes of inertia. A polygon point is defined by left-click. Double-clicking the left mouse button ends a polygon. To start the calculation of the main axes of inertia, press the <F2> key.

Important: This function is only available for triangle meshes.

Tool for the generation of a best fit cylinder for the selected area in the 3D data. Activate this tool and use the mouse to draw one or more polygons in the 3D data. The polygons define the area for the calculation of the best fit cylinder. A polygon point is defined by left-click. Double-clicking the left mouse button ends a polygon. To start the calculation of the best fit cylinder, press the <F2> key. Since not the cylinder but its axis will be used for the alignment, the cylinder axis is displayed as a vector after the fit has been performed.

Tool for the generation of a best fit cylinder for selected points of the 3D data. Activate this tool and use the mouse to define points in the 3D data. These points will be used for the calculation of the best fit cylinder. A point is defined by left-

click. To start the calculation of the best fit cylinder, press the <F2> key. Since not the cylinder but its axis will be used for the alignment, the cylinder axis is displayed as a vector after the fit has been performed.

Tool for the calculation of the boundary planes of the 3D data for a selected area of the 3D data. The generated boundary planes are parallel to the XY, XZ and YZ-planes, respectively, of the current coordinate system.

<u>Functions for the Generation of Alignment Elements from Existing Elements</u>

Depending on the selection status of existing elements in the 3D data explorer, different functions are available for the generation of additional elements.

Function for the generation of a point

- For **3 Planes**, the generated point is the point of intersection of the 3 planes

- For **Vector and Plane**, the generated point is the point of intersection of vector and plane

- For **Point and Plane**, the generated point is the projection of the selected point on the selected plane

- For **Circle**, the generated point is the center of the circle

Function for the generation of a vector

- For **Plane**, the generated vector is the normal vector of the plane

- For **Polyline**, the generated vector represents the line of best fit through the points of the polyline

- For **Two Points**, a vector is generated with the selected starting and end points

- For **Two Planes**, the generated vector represents the line of intersection of these two planes

- For a **Point** and a **Plane**, the generated vector has the selected starting point, and it is normal to the plane

- For a **Vector** and a **Plane**, the generated vector is the projection of the selected vector on the selected plane

Function for the generation of a plane

- For **Three Points**, a plane is generated which is defined by the selected points

- For **Circle**, the generated plane is a circle plane

- For **Vector**, the generated plane is normal to the vector and passes through the starting point of the vector

Functions for the Manipulation of Alignment Elements

This function is used to reverse the direction of all selected vectors in the scene explorer

This function is used to reverse the direction of all selected planes in the scene explorer

This function is used to translate all selected planes in the scene explorer in normal direction. The translation distance is the value set in the input field

Functions for 3D Viewer Control

The buttons and the functions they represent are identical to those in the main window. Please refer to that chapter for detailed information.

Execution of the Alignment

After the elements required for the alignment have been created with the above tools, the elements that are to be used for the alignment of the 3D data are specified in the selection fields. You can also drag and drop elements to be used for alignment directly into the selection fields from the scene explorer.

Start the alignment by clicking the *Align* button. The dialog for the alignment settings is displayed:
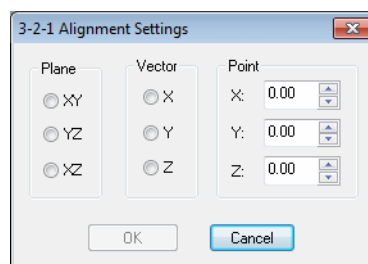

In this dialog, you can define the plane that will be used as the best fit plane in the new coordinate system (please note that the direction of the plane's normal vector will be taken into account in the alignment). In addition, you can specify the axis that will be used as the best fit vector in the new coordinate system, and the coordinates of the selected point in the new coordinate system. Click *OK* to execute the alignment. You can undo the last performed alignment with the *Undo* button.

Note

The symbol in the status bar indicates that an online transformation is active.
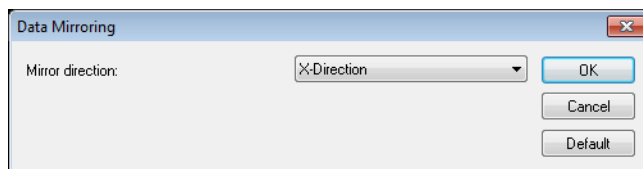
11.1.5 <u>Scale</u>

To scale all loaded data sets, choose *Edit → Transform data → Scale*. The following dialog appears:
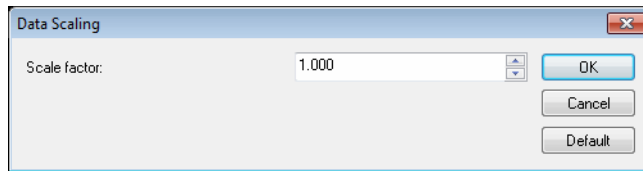
Enter the factor and start the calculation by pressing OK.

11.1.6 <u>Mirror</u>

To mirror all loaded data sets, choose *Edit → Transform data → Mirror*. The following dialog appears:

Select the mirror direction and start the calculation by pressing OK.

11.2 Optimize Data

11.2.1 <u>Smoothing</u>

To optimize all loaded, rasterized data sets, choose *Edit → Optimize data→ Smoothing*. The following dialog appears:
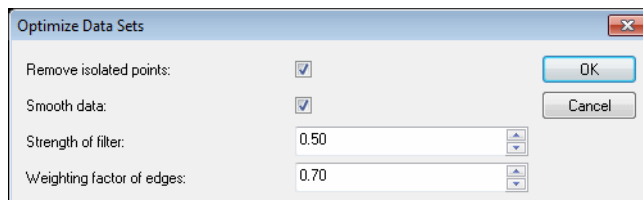
<u>Remove isolated points</u>

If you choose this option, isolated points will be removed from the data.

<u>Smooth data</u>

If you choose this option, the data will be filtered with the following filter parameters:

Strength of filter

Determines the strength of the filter. Valid values are in the range of 0 (no filtering) to 1 (strong filtering).
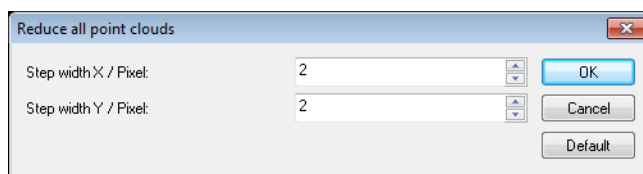
Weighting factor of edges

Determines the weighting of object edges. Valid values are in the range of 0 (object edges will not be specially treated and will be filtered) to 1 (object edges will not be filtered).

11.2.2 Decimating

To systematically decimate visible data sets, choose *Edit → Optimize data→ Decimating*. The following dialog appears:


When you click *OK*, only every x^{th} value in each column and every y^{th} value in each row of each data set will be kept.

This function is effective for the following data types: rasterized and non-rasterized data, unstructured point clouds and polylines.

11.3 Matching

11.3.1 Global Matching

If you used the Move Station Wizard (see chapter 15.9), all data will be in the same coordinate system. However, depending on the accuracy and position of the reference points, the result may still show deviations in the overlapping area. These deviations can be minimized by matching. To start global matching, choose *Edit → Matching → Global Matching...*. The following dialog appears:
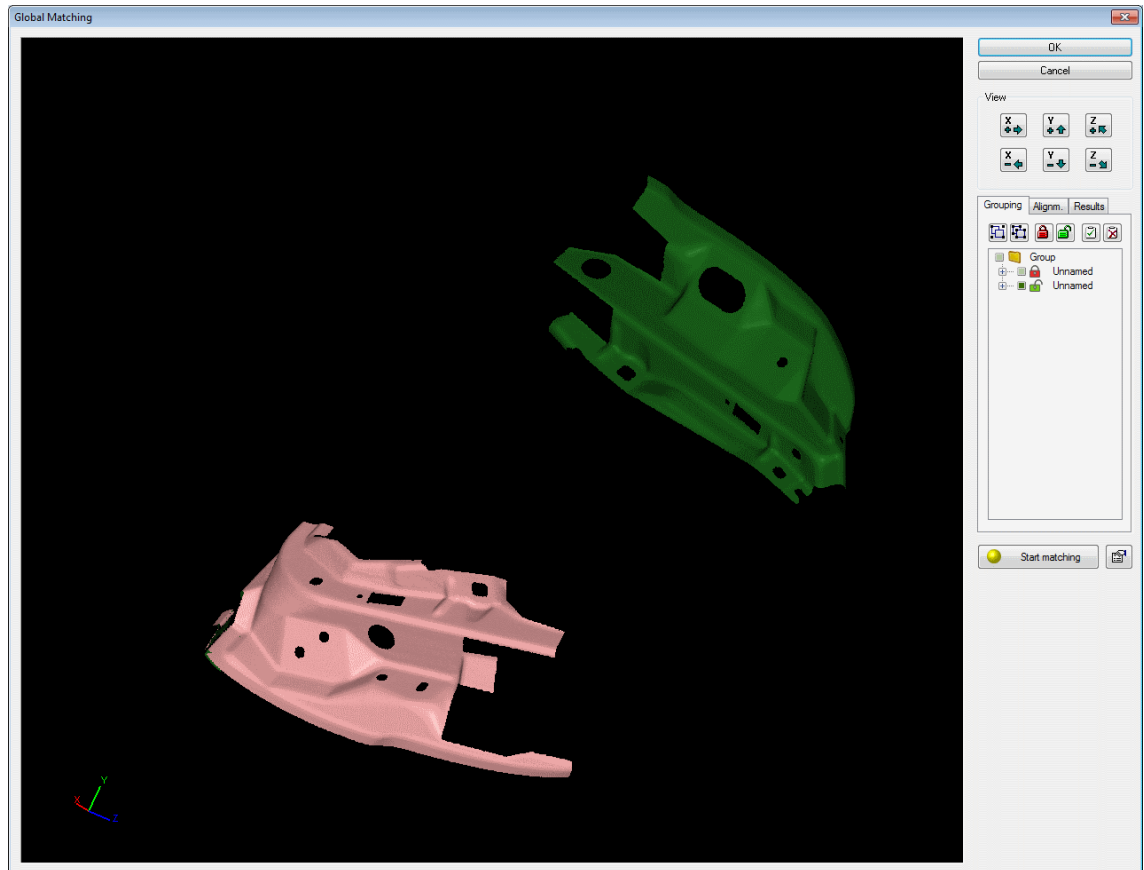
The current 3D view with all measurement data is displayed; each measurement is in a group and the first measurement is locked. You can now *group* or *ungroup* data sets, *lock* or *unlock* individual groups, and *use* or *ignore* them. When you have made the required settings, you can start the matching. The algorithm tries to iteratively minimize the distances between all groups in the overlapping area.

If you have already created groups in your project, these groups are kept when global matching is started. The groups generated here are included in the project when you leave the dialog.

For grouping data sets, the following tools are available:

All selected data sets (also those in a selected group) are combined to a new group. Its name is the name of the first data set in the group followed by the number of data sets within the group (if more than one data set is contained in the group).

All selected data sets (also those in a selected group) are grouped separately, i.e. after execution of this function, each data set is in its own group.

All selected groups are locked, i.e. their position is not changed during matching

All selected groups are unlocked, i.e. their position is optimized during matching.
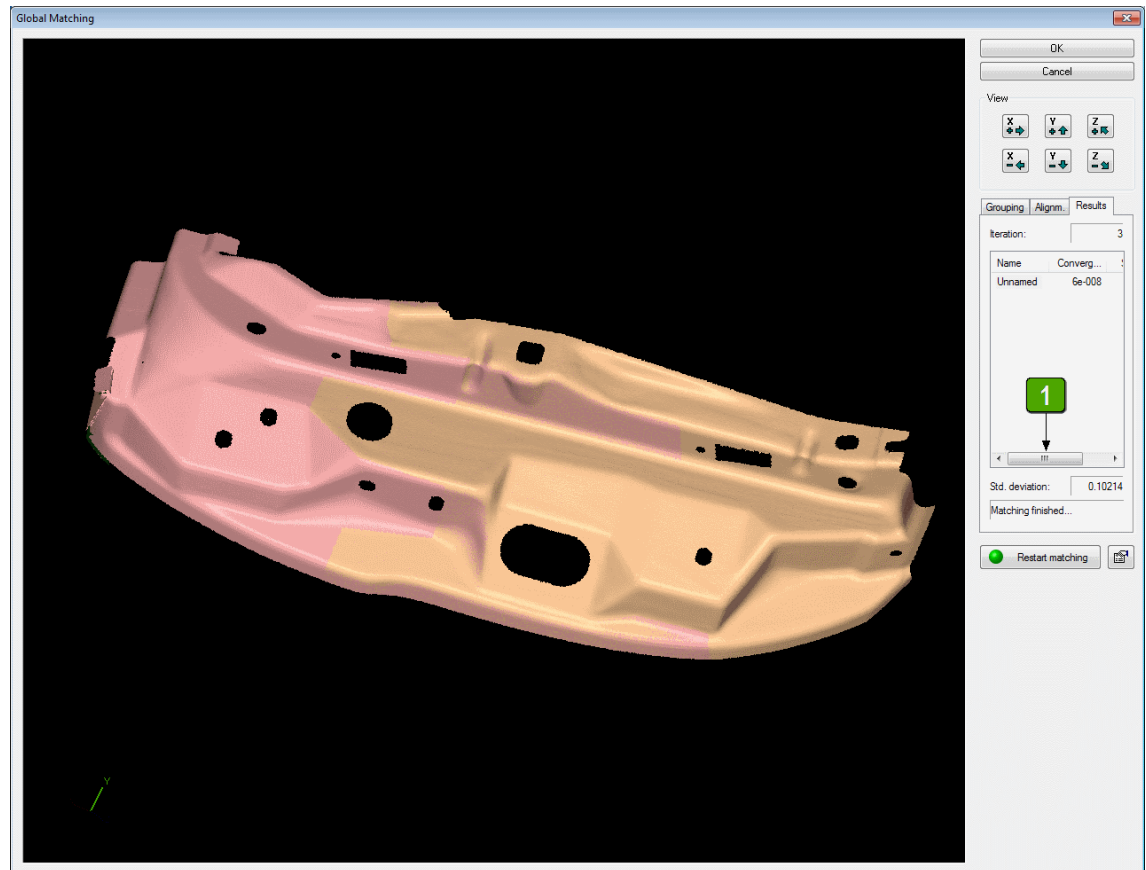
	All selected groups are used during matching.
	All selected groups are ignored during matching and will not influence the match result.

When you press *Start Matching*, the optimization is started. The dialog will then look like this:

After each iteration, key information about the matching process is shown in the list (center right) and in the text fields (bottom right):

- The number next to *Iteration* indicates how many iterations have already been executed.

- The number next to *Convergence* shows the current convergence level. The smaller this number is, the closer the measurement comes to the required target position.

- The number next to *Std. Deviation* displays the current standard deviation for the mean deviation, i.e. how much the single deviations of points in the overlapping area differ from the mean deviation. This value largely depends on the noise in the measurement data and on the systematic deviation to the previous data sets.

You can specify the interruption criteria for the iteration in the dialog for the surface matching parameters. These parameters are described in more detail in chapter 17.2. You can also set this parameter from this dialog by clicking the button to the right of *Start Matching*.

You can also interrupt the iteration of the surface matching by pressing the *Stop Matching* button. If the quality is not sufficient, you can restart matching with the *Start Matching* button.

The values for *Std. Deviation* are also shown in the list. To make them visible, move the horizontal scroll bar **1** to the right.

You can have these values displayed in the text fields below the list again at any time by selecting an entry in the list with the mouse or the arrow keys.

If you have groups of data that have not yet been aligned to each other, you can do this interactively. Select the *Alignment* tab. The dialog will then look like this:
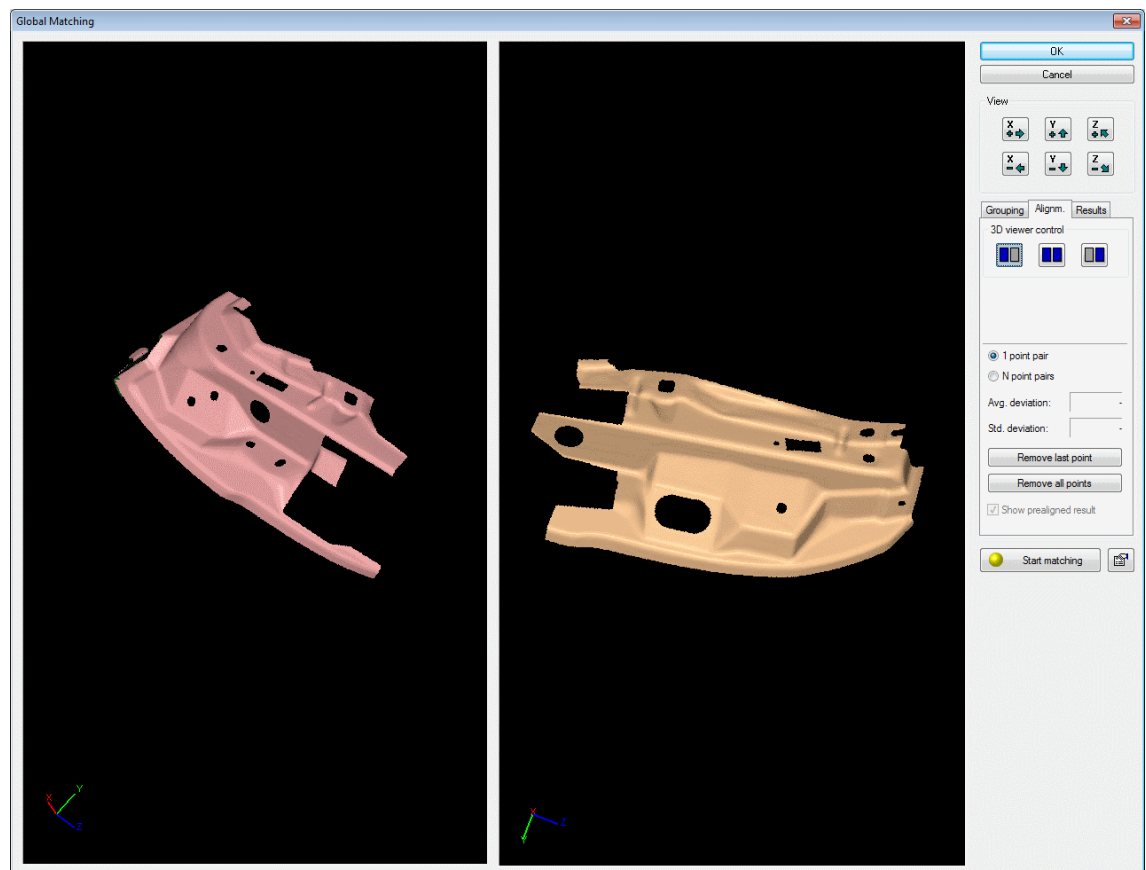
Pre-alignment by 1 corresponding pair of points

This dialog contains two different 3D views. The left window shows all locked data (which is already in the target coordinate system) and the right window shows all other data.

For successful matching, one and the same surface point has to be marked in each of the two windows. You can mark such a surface point by clicking the surface position once with

the mouse. At this position, a small sphere is displayed for better orientation. It does not matter in which of the two windows you mark the point first.

When the surface point is marked in both windows, the system automatically starts the surface matching. If the object surface has enough structure, the matching converges after few iterations and the result is displayed.

You can assess the quality of the match result both visually and by checking the values given in the text fields (bottom right):

- The number next to *Iteration* indicates how many iterations have already been executed.

- The number next to *Convergence* shows the current convergence level. The smaller this number is, the closer the measurement comes to the required target position.

- The number next to *Average Deviation* shows the mean deviation of all points in the overlapping area compared to all other measurements. Ideally, this value is close to 0.

- The number next to *Std. Deviation* displays the current standard deviation for the mean deviation, i.e. how much the single deviations of points in the overlapping area differ from the mean deviation. This value largely depends on the noise in the measurement data and on the systematic deviation to the previous data sets.

You can interrupt the iteration of the surface matching by pressing the *Stop Matching* button. If the quality is not sufficient, you can restart matching with the *Start Matching* button.

Pre-alignment by N corresponding pairs of points

If the object surface does not provide sufficient structure, you need to mark several corresponding point pairs for pre-alignment. You can mark surface points alternately in both windows, or you can mark all points in one window first. It is only important that the points are marked in the same order. For better orientation, the selected points are visualized as spheres in different colors.

When at least three corresponding points have been selected, the program calculates the approximate position and, optionally, displays the unlocked data transparently in the left window (*Show prealigned result* checkbox). You can assess the quality of the pre-alignment result either visually or by checking the values given in the text fields (center right). There the average deviation and the standard deviation of the point-to-point matching are indicated. If the quality is not yet sufficient, you can select further corresponding pairs of surface points.

When you are finished, start the surface matching manually with the *Start Matching* button.

If the matching failed completely, or if you made a mistake in the order in which you selected the points, you can remove all point pairs by clicking *Remove all points* and start again.

Note

To simplify assessment of the match result, each group and each measurement are displayed in a different color. This makes it easier to identify how well the measurements overlap in the overlapping area.

By pressing <Space Bar> (if the cursor is in one of the two 3D views), you can toggle the mode of operation for the two 3D views between point definition (cursor) and viewing mode (hand). To synchronize the display selection for the two 3D views, activate the middle button *in the* 3D Viewer Control *group.*

You can access the surface matching parameters directly from the dialog by pressing the button to the right of Start matching. *The parameters for surface matching are described in more detail in chapter 17.2.*

11.3.2 <u>Tolerance Based Matching</u>

As opposed to global matching, tolerance based matching takes additional constraints into account in the process. A threshold defines the maximum movement of a data set during matching. Tolerance based matching helps to prevent global scaling errors or an uncontrolled deformation of the object shape, and achieves a high global accuracy of the match result. To execute tolerance based matching, choose *Edit → Matching → Tolerance based matching…*. The following dialog appears (the dialog may take some time to open, depending on the amount of data):
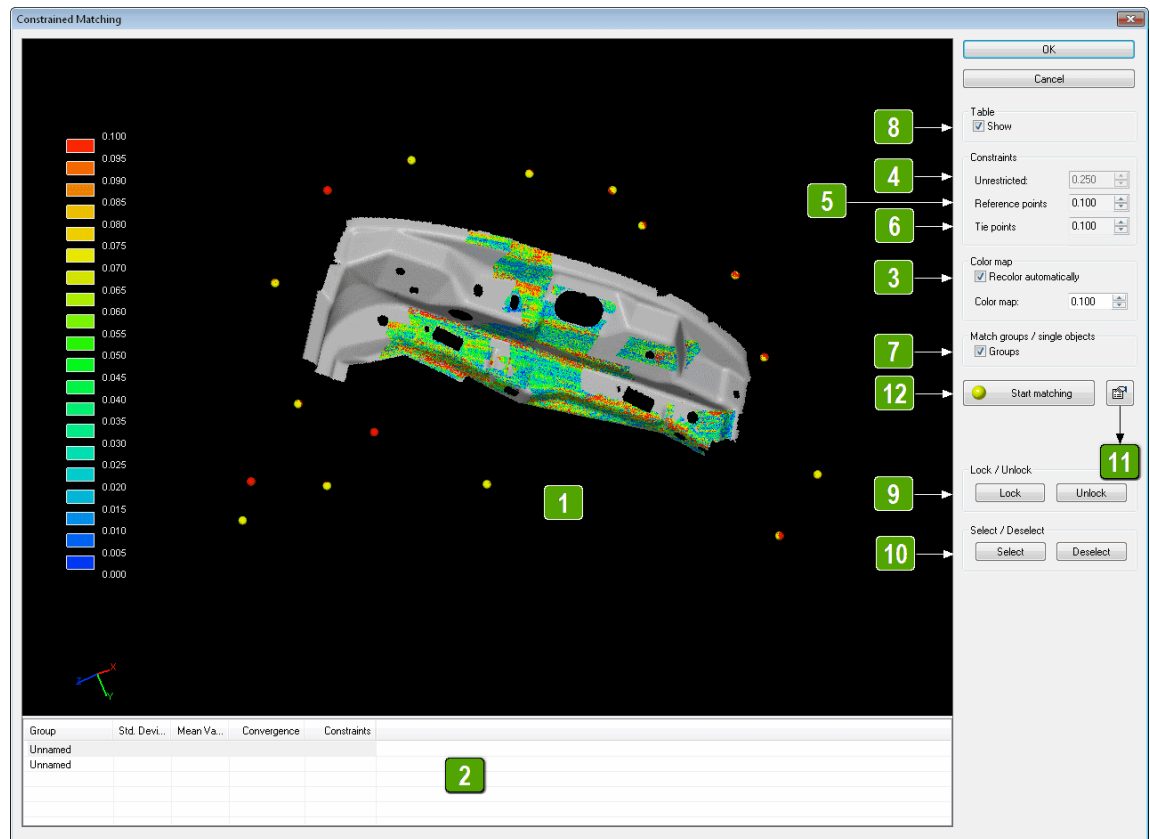
In the main part of the dialog, a 3D viewer **1** visualizes the measurement object. The table below it displays the match results **2**. The size of the result table can be modified by placing the cursor over the dividing line between the 3D viewer and the result table. When the cursor changes to a vertical double-headed arrow, press and hold the left mouse button and resize the table.

If you select the *Recolor automatically* checkbox **3**, deviations (in millimeters) between overlapping scans are displayed in color. Deviations that exceed the entered value are shown in the default color like scans that do not overlap.

If *Groups* **7** is deselected, all overlapping scans are colorized; if *Groups* is selected, only overlapping scans that are in separate groups are colorized.

Depending on the data, you can apply different matching strategies:

- If the data is not grouped, the value entered for *Unrestricted* **4** defines the maximum distance in millimeters that **each data set** may be moved. If *Recolor automatically* **3** is active, all overlapping scans are colorized.

- If the data is in at least two groups, the behavior depends on the setting of the *Groups* **7** checkbox. When deselected, the behavior is as described above. When selected, data sets within a group will not be moved. In this case, the *Unrestricted* parameter **4** defines the maximum distance in millimeters that a **group** may be moved. If *Recolor*

automatically `3` is active, only overlapping scans that are in separate groups are colorized.

- If you created groups with the Move Station wizard before starting tolerance based matching, and if the *Groups* checkbox `7` is active, the *Tie points* `6` value defines the distance the **groups** may be moved. The groups are moved in such a way that the tie points are no further away from their original positions than the specified value (in millimeters). If *Groups* `7` is activated, tie points are displayed as yellow spheres.

  This only works as long as no data sets are manually moved from a group created by the Move Station wizard to another group (which is generally not advisable in any case).

- If you created groups with the Move Station wizard and also transformed the data with the Transform to Nominal Points wizard before starting tolerance based matching, the *Reference points* `5` field can be used. In this case, **groups** are moved in such a way that neither the reference points nor the tie points are further away from their original positions than the respective specified values (in millimeters). If *Groups* `7` is activated, reference points are displayed as red spheres.

Note

When tolerance based matching is restarted repeatedly, previous actions will be discarded, i.e. transformation calculations will start from scratch. However, this is not true for matching with Groups `7`*, which will remain unchanged until a new matching with* Groups `7` *is executed.*

This gives you the possibility to match the groups first and then the individual measurements.

With the *Show* checkbox `8`, you can toggle the display of the table containing the scans and groups.

Selected data sets and groups can be locked for matching. This means that movement of the selected data set or group is prevented. To lock a data set, select the data set or group in the result table and click *Lock* `9`. If you select multiple data sets, all selected data sets are locked. Locked data sets are shown gray in the result table. The first data set is locked by default.

To unlock data sets, select them and click the *Unlock* button.

With the functions *Select* `10` and *Deselect*, data sets or groups can be highlighted in another color in the 3D viewer. This is basically done in the same way as locking / unlocking data sets. Data sets can also be selected directly in the 3D viewer (see chapter 5.1). Selected data sets are marked with an asterisk (*) in the result table.

By clicking `11` you can open the settings dialog for tolerance based matching. For further information, see chapter 17.2.

Before you start matching, specify the required tolerance value. Then click the *Start matching* button.

During matching, the standard deviation, mean value and convergence for each data set are displayed in the result table. The Constraints value indicates the distance by which each data set was moved from its original position. All values are given in millimeters.

After the matching has finished, deviations in the surface measurement data sets are visualized according to an updated color lookup table if the *Recolor automatically* checkbox is active.

Note

---

*The data quality can generally be increased with tolerance based matching. Therefore it makes sense to **always** apply tolerance based matching – after scanning, but before post-processing. For best results, do relatively short scans and avoid changing the scanning direction. In other words, do not move the scanner back and forth over the object in a single scan. Instead, do two scans with the scanner moving first in one direction and then in the opposite direction.*

11.4 Optimize Triangle Meshes

To access the triangle mesh optimization functions, choose *Edit → Optimize triangle mesh*. The following submenu appears:
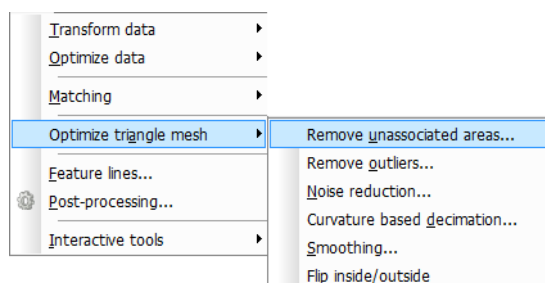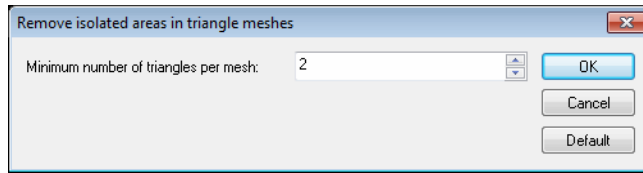

11.4.1 <u>Remove Unassociated Areas</u>

This function is used to remove isolated meshes or parts of meshes which consist of only a few triangles.

The menu entry *Edit → Optimize triangle mesh → Remove unassociated areas* is available when at least one triangle mesh has been loaded.

The following dialog appears:

A mesh – or part of it – is identified as isolated and thus removed if its triangle count is less than *Minimum number of triangles per mesh*.

11.4.2 <u>Remove Outliers</u>

Using this function, individual triangle mesh knots that protrude significantly from their environment can be moved towards the surrounding surface.

The menu entry *Edit → Optimize triangle mesh → Remove outliers* is available when at least one triangle mesh has been loaded.

The following dialog appears:
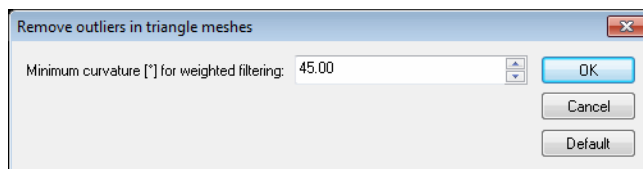
In order to remove outliers, the sum of the angles between surrounding mesh edges is calculated for each knot. If the difference between the angle sum and 360° exceeds the *Minimum curvature*, this knot is transformed by projection towards the surrounding surface.

11.4.3 <u>Noise Reduction</u>

This function is used for smoothing triangle meshes in such a way that individual knots are transformed along their normal vector towards the surrounding surface.

The menu entry *Edit → Optimize triangle mesh → Noise reduction* is available when at least one triangle mesh has been loaded.

The following dialog appears:
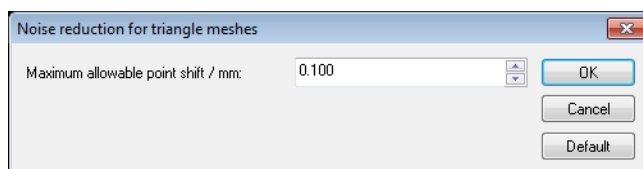
Maximum allowable point shift defines the maximum distance a mesh knot is allowed to be shifted by smoothing.

11.4.4 <u>Curvature Based Decimation</u>

This function is used for decimating triangle meshes in such a way that the resulting mesh does not differ from the original mesh by more than the specified tolerance, and the resulting edges of the triangle do not exceed the specified length.

The menu entry *Edit → Optimize triangle mesh → Curvature based decimation* is available when at least one triangle mesh has been loaded.

The following dialog appears:
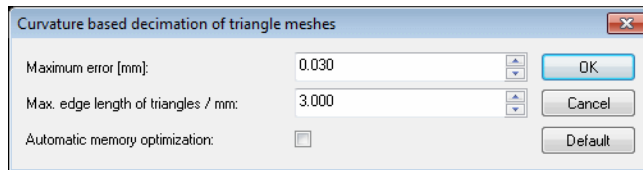
The *Maximum error* defines the maximum permissible difference between the resulting mesh and the original mesh.

The *Max. edge length of triangles* defines that the mesh is decimated in such a way that resulting triangle edges do not exceed the specified value.

For a detailed description of *Automatic memory optimization*, see chapter 12 Post-Processing.

11.4.5 <u>Smoothing</u>

This function is used to smooth a triangle mesh. For best results, the filter does not just alter single points, but larger areas of a mesh.

The menu entry *Edit → Optimize triangle mesh → Smoothing* is available when at least one triangle mesh has been loaded.
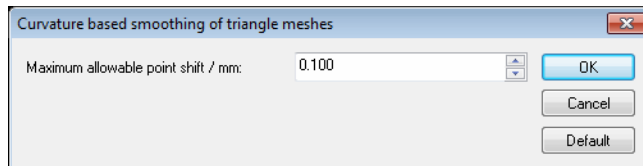
The following dialog appears:

Maximum allowable point shift defines the maximum distance a mesh knot is allowed to be shifted by the filter.

11.5 Interactive Tools

The interactive tools are described in detail in chapter 8.

12 Post-Processing

T-Scan Collect offers the possibility to automatically post-process the acquired 3D surface data in order to create a triangle mesh.

To automatically post-process the data, click the *Post-Processing* icon in the standard toolbar or choose *Edit → Post-processing*. The following dialog appears:
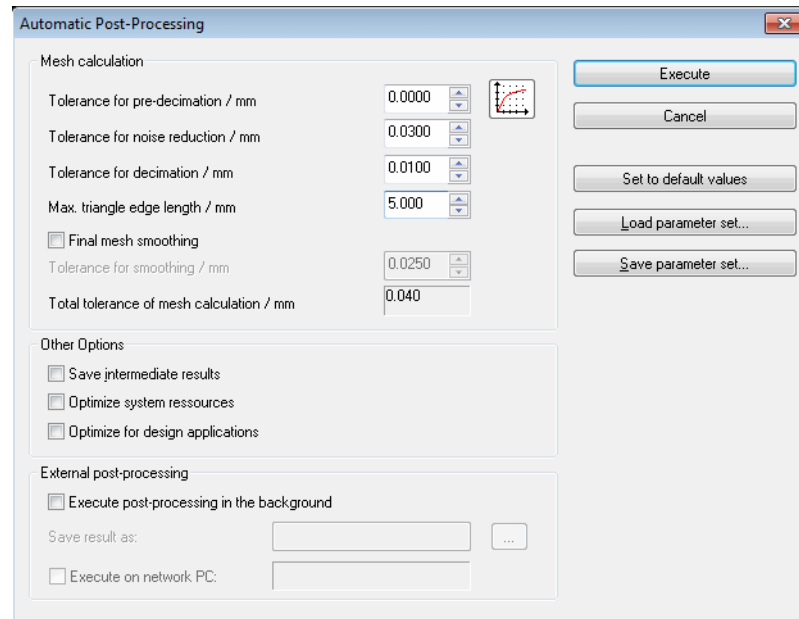
During automatic post-processing, the separate surface measurement data sets are combined to an overall triangle mesh (data such as reference points or polylines are not affected). In the process, the individual data sets are pre-decimated and then combined to a single triangle mesh. The result is further processed to remove small isolated fragments, outliers and to reduce noise. Afterwards, the mesh is decimated once again to reduce the number of triangles. Optionally, the mesh is smoothed in a final step.

Tolerance for pre-decimation
To use reasonable values for pre-decimation, it is recommended to run a data analysis first. To do so, press the button for *Data analysis* . The following dialog appears:
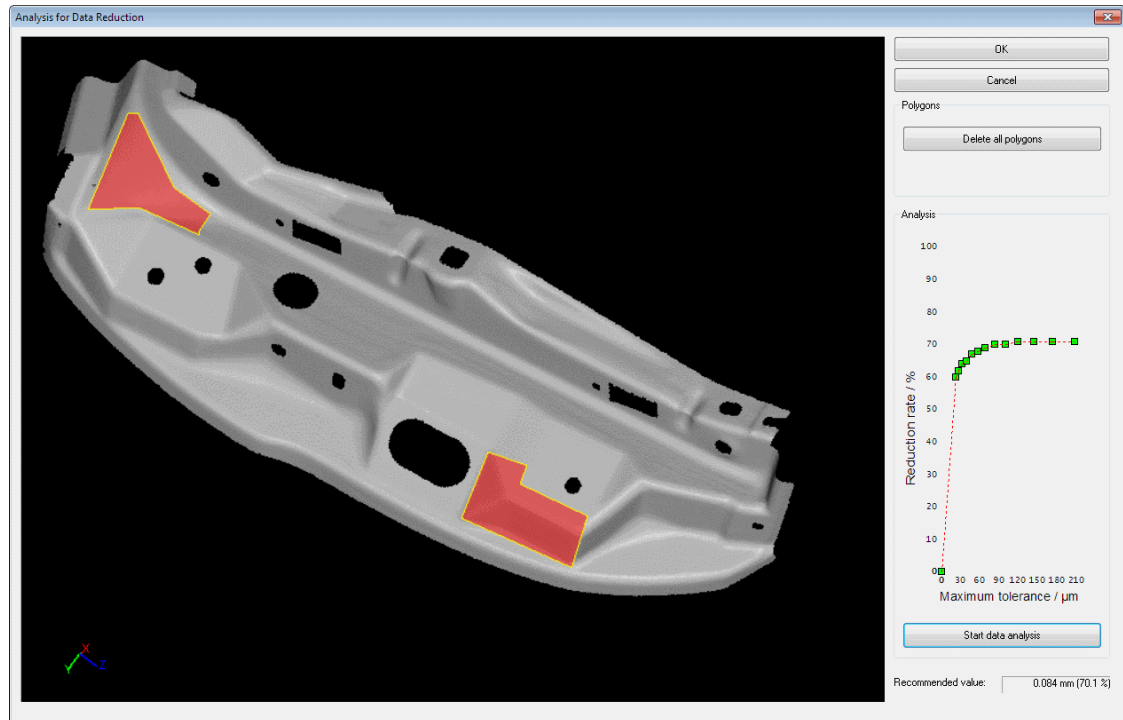
To perform a data analysis, mark characteristic areas of the object by creating polygons on the object. First rotate the object to the desired viewing angle and change to the tool mode of the 3D viewer by using <Space Bar>. With a single left-click you define a polygon point, and with a double-click you end the polygon. After selecting one or more areas in this way, start the data analysis. You will get a characteristic curve which shows the decimation rate depending on the tolerable error. After the analysis, the recommended error value for pre-decimation is displayed. By pressing OK, you can accept this value as a post-processing parameter.

If you do not want to use the value returned by the data analysis or if you want to change it, you can close this dialog and manually enter a value for the tolerable error.

Tolerance for noise reduction
This parameter defines the maximum permissible shift of points in the triangle mesh relative to the original data.

Tolerance for decimation
This parameter defines the maximum permissible shift of the processed data relative to the original data.

Max. triangle edge length
This parameter defines the maximum edge length a triangle can have after the calculation.

Final mesh smoothing
When activated, a smoothing filter is applied to the triangle mesh at the end of post-processing.

Tolerance for smoothing

This parameter can only be changed if *Final mesh smoothing* is active. It defines the maximum permissible shift of points in the triangle mesh relative to the original data.

Total error of mesh calculation

This value is calculated automatically and indicates the sum of the permissible tolerances of the individual steps.

Save intermediate results

If this option is selected, a separate backup copy of the project is made after every calculation step. The backup files are located in the current project directory.

Optimize system resources

This option should be activated if large data sets are to be post-processed and the computer may not have enough memory (independent from the operating system).

When activated, a data analysis step will be performed prior to the post-processing. If the analysis determines that there is not enough free physical memory for the operation, some steps of the post-processing are modified to reduce the amount of required memory, but this will increase calculation time.

If the analysis determines that no optimization is necessary, the post-processing will run as usual.

Note

When handling very large data sets, the post-processing might fail – even if resource optimization was enabled – because there is not enough memory available in general. In this case, the original data must be reduced or a more powerful computer must be used.

Optimize for design applications

If this option is activated, higher priority is given to more uniform triangles and a smoother surface.

However, if features (e.g. hole diameters, hole positions, etc.) are to be extracted from the triangle mesh, it is recommended to deactivate this option. This is the default.

Load/Save parameter set

These functions enable you to save or load the parameters used for post-processing. When the dialog appears, the parameters last used are loaded automatically.

Set to default values

Press this button to use default values for the post-processing. The parameters will be set depending on the point-to-point distance of the existing 3D data. Generally you should obtain reasonable results when using these values.

External post-processing

When the *Execute post-processing in the background* option is active, the measurement data is sent to another instance of T-Scan Collect and processed there. In the meantime you can work normally in the main instance and, e.g., start new measurements.

Enter a valid path into the field *Save result as* or use the respective button to select the path in a file selection dialog.

External post-processing can also be executed on a remote network computer. For this purpose, the following requirements must be met on the remote machine:

- Installation of the T-Scan Collect application (this requires an additional dongle)

- Configuration of the required DCOM access permissions (see chapter 20.1)

- Folder share with read and write permissions

Furthermore, some modifications need to be made to the Postprocessing_RangeImageBased.ini file. This file is located

- in `%AllUsersProfile%\Application Data\Steinbichler\T-SCAN\T-Scan Collect 10.3x` (Windows XP)

- or in `%ProgramData%\Steinbichler\T-SCAN\T-Scan Collect 10.3x` (Windows 7)

Open this file with a text editor and adjust the following section accordingly:

```
[External]
EnableExternalPostproc=TRUE
ResultsPath=\\some-server\some path\filename.stl
EnableRemotePC=TRUE
RemotePC=\\some-server
```

- Set EnableRemotePC=TRUE

- Set RemotePC ("some-server") to the name of the computer on which the post-processing will be executed.

Save this file (under a different name, if necessary) and load it by clicking the *Load parameter set...* button. The *Execute on remote PC* option is activated and the remote computer name is displayed.

After clicking *Execute*, you can continue using the first instance of T-Scan Collect as usual, while a second instance runs the post-processing. When the external post-processing has finished, a message box appears in the first instance. Clicking *OK* in this message box closes the second instance.

13 Calculation of Feature Lines

For the calculation of feature lines, choose *Edit → Feature lines...*. The calculation of feature lines only works on triangle meshes.

The following dialog appears:
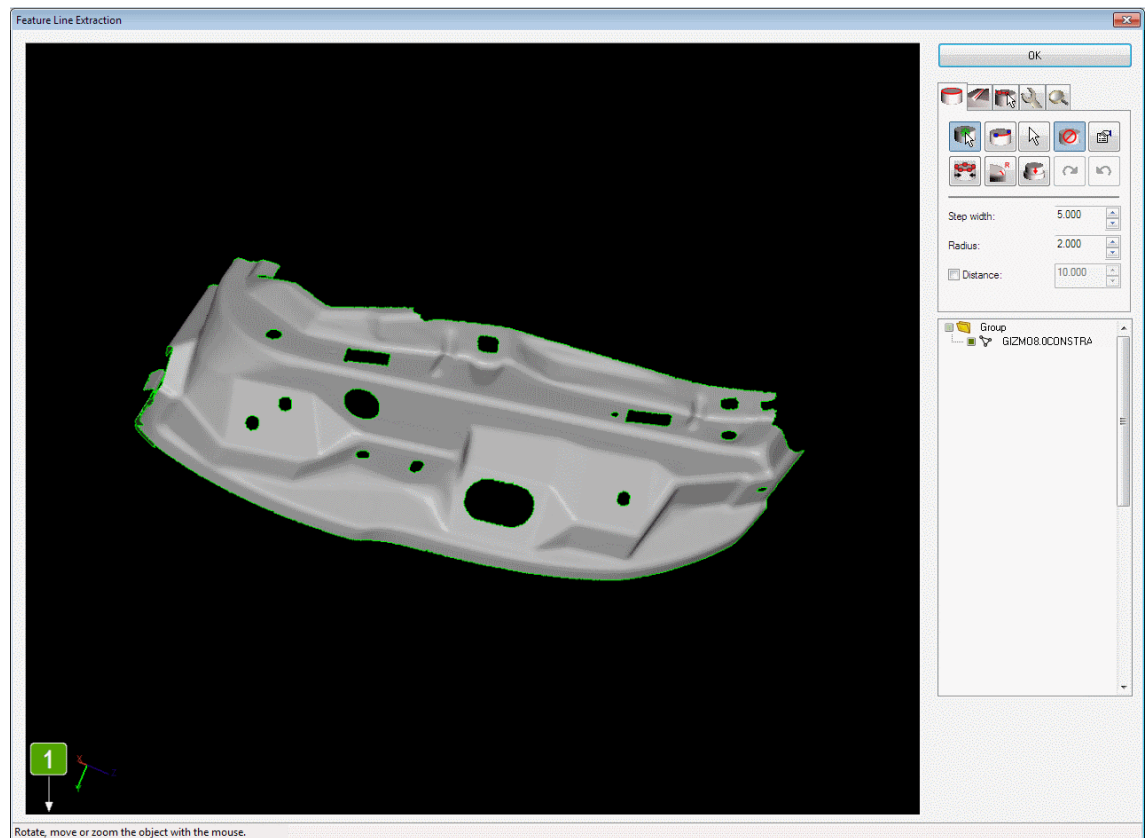
In the left part of the window, the loaded triangle meshes are displayed. Below the 3D viewer, the status bar ![1] indicates how to use the individual tools. On the right, you will find the data explorer with the project. Above it, there are several tabs with different tools for the calculation of feature lines. The tools are grouped as follows:

Functions and tools for the calculation of feature lines (fillets); the theoretical intersections as well as the surface data can be extracted

Functions and tools for the calculation of V-lines

Functions and tools for the interactive generation of polylines

Functions and tools for the processing of polylines and triangle meshes

Functions and tools for 3D viewer control

In the project, the generated feature lines, V-lines and polylines are grouped as follows:

- Theoretical_Intersections: Contains all objects that were generated from the section of planes

- Surface_Data: Contains all objects on the surface

- Borderlines: Contains all border lines of an object

13.1 Calculation of Feature Lines

The tab with the functions for the calculation of feature lines looks like this:
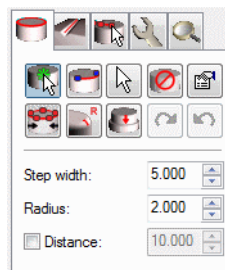
The buttons represent the following tools:

Extraction of a new feature line:

First, adjust the values for step width, radius and, if applicable, distance. Then click two relatively close points on the feature line. The program now automatically extracts the feature line from the data. Depending on the setting of the extraction parameters (see below), the theoretical intersections and/or the surface data can be extracted. If an extraction is not possible, a warning message will be displayed. In this case, change the values for the step width and/or radius and/or distance, and click the *Recalculation* button (see below).

The extracted feature line is displayed in the 3D viewer as follows:
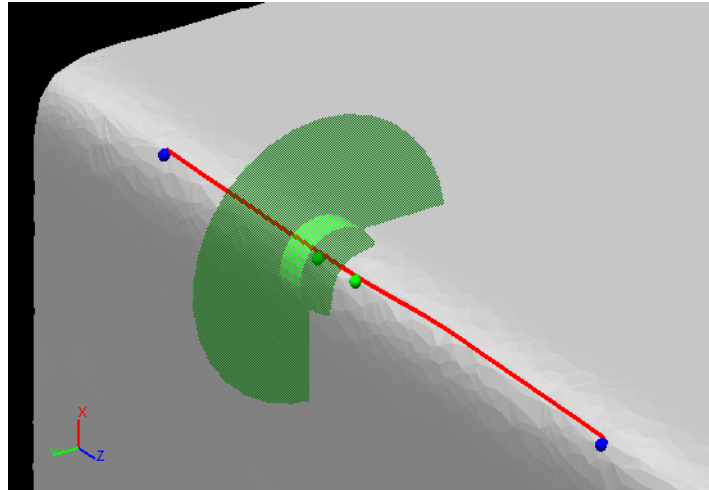
For a better graphical orientation, the two points you clicked are displayed as small green spheres. At the first point, a cylinder of the length *Step Width* and with the specified *Radius* is shown as a semi-transparent graphic. If the distance limit is active, a disk with the radius from *Distance* is also displayed. These help graphics change their size according to changes made in the parameters *Step Width*, *Radius*, or *Distance*.

You can limit the evaluation area around the edge by activating the distance criterion and specifying the height of the step in the object.

This button is used to define the start and the end of the feature line to be calculated. For this reason, the button is only enabled when the tool for the calculation of feature lines is active.

First click the starting point, and then the end point of the feature line. Both points are displayed as blue spheres in the 3D viewer. When both points are defined, the tool is automatically deactivated, and you can extract the feature line without further action (see above).

If you wish to change or delete the start or end point, activate this tool again and click the *Undo* button (see below). Then you can redefine the points or deactivate the tool (by clicking it once more).

Selection of any data objects (triangle meshes, feature lines, polylines, etc.)

With this button, all displayed triangle meshes can be hidden and shown again later.

This button opens the settings dialog with the parameters for the extraction of feature lines:
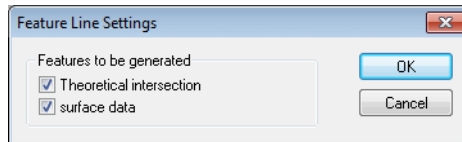
Here, you determine whether only the theoretical intersections or the surface data or both are to be extracted.

 This tool is used to define the step width interactively in the 3D viewer. Click two points one after the other (preferably on the feature line) in a distance that corresponds to the step width. The distance of these two points is displayed graphically in the 3D viewer and directly inserted as *Step Width* (rounded).

 This tool is used to define the radius interactively in the 3D viewer. This is basically done using the same procedure as described for the above tool.

 This tool is used to determine the applied distance interactively in the 3D viewer. This is basically done using the same procedure as described for the above tool.

 Use this button to extract the feature line once again, e.g. after changes to the parameters.

 Use this button to undo the last action (depends on the selected tool).

13.2 Calculation of V-Lines

The tab with the functions for the calculation of V-lines looks like this:

The buttons represent the following tools:

 Extraction of a new V-line:

First adjust the values for step width, width and depth. Then click two relatively close points on the base of the V-line. The program automatically extracts the V-line from the data. If an extraction is not possible, a warning message will be displayed. In this case, change the values for the step width and/or width and/or depth, and click the *Recalculation* button (see below).

Selection of any data objects (triangle meshes, feature lines, polylines, etc.)

With this button, all displayed triangle meshes can be hidden and shown again later.

At present, there are no global settings for the calculation of V-lines.

This tool is used to define the step width interactively in the 3D viewer. Click two points one after the other (preferably on the V-line) in a distance that corresponds to the step width. The distance of these two points is displayed graphically in the 3D viewer and directly inserted as *Step Width* (rounded).

This tool is used to define the width of the V-line interactively in the 3D viewer. This is basically done using the same procedure as described for the above tool.

This tool is used to determine the depth of the V-line interactively in the 3D viewer. This is basically done using the same procedure as described for the above tool.

Use this button to extract the V-line once again, e.g. after changes to the parameters.

Use this button to undo the last action (depends on the selected tool).

13.3 Interactive Generation of Polylines

The tab with the functions for the interactive generation of polylines looks like this:
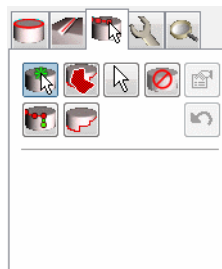
The buttons represent the following tools:

Use this button to generate a new polyline. On the object surface, click all points, one after the other, that define the polyline. End the polyline by double-clicking the last point.

For a better graphical representation, each point you click is displayed as a small green sphere in the 3D viewer.

This tool is used to limit the area for the extraction of borderlines. First position the area of interest of the object in the 3D viewer, then draw a polyline around this area and activate the button *Extract Borderline* (see below).

	Selection of any data objects (triangle meshes, feature lines, polylines, etc.)
	With this button, all displayed triangle meshes can be hidden and shown again later.
	At present, there are no global settings for the interactive generation of polylines.
	With this button, you can extend an existing polyline. First, select the polyline that is to be extended by clicking onto it in the 3D viewer. For a better graphical representation, each point of the line is displayed as a small green sphere.
	Then click all points on the object surface, one after the other, that are to be added to the polyline. End the polyline by double-clicking the last point. The first new point is added to that end of the polyline that is closest to the point you clicked.
	With this button, you can extract borderlines of the object. You can also limit the area in which the borderlines are to be extracted (see above).
	Use this button to undo the last action (depends on the selected tool).

13.4 Processing of Polylines and Triangle Meshes

The tab with the functions for processing polylines and triangle meshes looks like this:

The buttons represent the following tools:

	This tool is used to clip polylines. Position the area of interest of the object and the polylines to be clipped in the 3D viewer, then draw a polygon around the area of interest and use the <Del> key to clip the polylines located underneath. You can also draw several polygons and then clip the data below these areas in a single step.
	Please note that this function can only be used to clip polylines but not the triangle meshes that might be displayed.
	This function is used to smooth polylines. First, select the polyline to be smoothed by clicking it in the 3D viewer. Modify the number of control points until the polyline meets your requirements. The line becomes smoother by reducing the number of control points.

Please keep in mind that the smoothed line will be saved as a sampled polyline with the original number of nodes.

Selection of any data objects (triangle meshes, feature lines, polylines, etc.)

With this button, all displayed triangle meshes can be hidden and shown again later.

At present, there are no global settings for the processing of polylines and triangle meshes.

With this button, you can combine two polylines into one. Select the first polyline by clicking it in the 3D viewer. Then select the polyline you want to combine with the first one. The two polylines will always be connected in the shortest possible way. You can then select additional polylines that are to be added to the already combined lines.

With this button, you can split a polyline. Click the polyline at the point where it is to be split.

With this button, you can add new points to an existing polyline. Select the polyline to which you want to add new points by clicking it in the 3D viewer. For a better graphical representation, each point of the line is displayed as a small green sphere.

Now click points on the polyline. At the clicked points, new nodes are added to the polyline.

With this button, you can delete points from the polyline. Select the polyline from which the points are to be removed by clicking it in the 3D viewer. For a better graphical representation, each point of the line is displayed as a small green sphere.

Then click those nodes of the line that are to be removed.

With this button, an open polyline can be closed. Select the polyline that is to be closed by clicking it in the 3D viewer. If the selected polyline is already closed, a warning message will be displayed.

With this button, you can convert a polyline to a NURBS curve. Click the polyline you want to convert to a NURBS curve.

With this button, you can fit a plane into different – preferably planar – areas of the object and intersect these planes. The result is a polyline along the intersection.

First, draw a polygon around the area of the object you want to use for fitting the first plane. Then draw a polygon around the area of the object that is to define the second plane. The two planes are automatically intersected.

The fitted planes are temporarily displayed as plane objects. It is recommended to position the object in the best possible way in the 3D viewer before the corresponding plane is fitted.

With this button, you can fit the corner. As a result, you will get three polylines along the edges of the corner, with the corner point as the start of the polylines.

Draw a polygon around each area of the object through which you want to fit a plane and which represents a boundary surface of the corner. When the third plane has been fitted, they are automatically intersected.

The fitted planes are temporarily displayed as plane objects. It is recommended to position the object in the best possible way in the 3D viewer before the corresponding plane is fitted.

With this tool, the edges and borders of an object can be sharpened. First, determine a radius of action, i.e., the area where the triangle mesh is to be modified. Then select the feature line (theoretical intersection) or borderline (preferably smoothed) by clicking it in the 3D viewer. The triangle mesh is moved towards the selected polyline so that rounded edges are eliminated or noisy borders are smoothed.

If you are not satisfied with the result, you can undo the last action or actions you made. You can then, e.g., adapt the radius and repeat the edge smoothing.

With this tool, you can limit areas in which edge smoothing is to be applied. Position the area of interest of the object and the edge to be sharpened in the 3D viewer, then draw a polygon around this area and select the tool for edge sharpening. The edge sharpening is then only performed for that area.

Use this button to undo the last action (depends on the selected tool).

13.5 3D Viewer Control

The tab with the functions for the control of the 3D viewer looks like this:
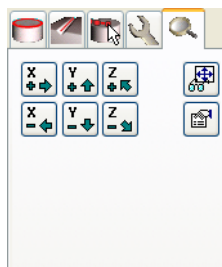
The buttons and their functions correspond to those in the main window (see chapter 5.1).

14 Service

Service functions are password protected and only accessible to service personnel.

15 Tracking System

15.1 Initialize

To initialize the tracking system, open the Tracker menu and select *Initialize*. The tracking system will reference its axes and the status display of the tracking system changes to green or yellow.

15.2 Disconnect/Connect

The *Disconnect* command can be used to disconnect the software from the emScon server to allow other software to access the emScon server. Once the connection is interrupted the menu item changes to *Connect*. Clicking on *Connect* re-establishes the connection to emScon.

15.3 Environmental Parameters

Before measuring you should check the environmental conditions. To do this, open the Tracker menu and select *Environmental conditions*. The following dialog opens:
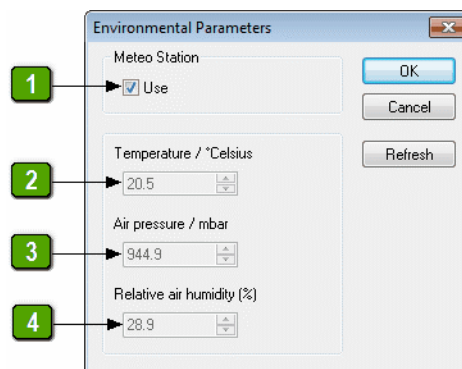
If you activate *Use Meteo Station* **1** the temperature, air pressure and humidity are obtained from the Meteo station. Activate this checkbox if you have a Meteo Station attached to your system. Otherwise you need to enter the values by hand.

If a weather station is attached to the system the temperature will be read out and displayed. Otherwise you should enter the current temperature **2** as precisely as possible.

If a weather station is attached to the system the air pressure will be read out and displayed. Otherwise you should enter the current air pressure **3** as precisely as possible.

If a weather station is attached to the system the humidity will be read out and displayed. Otherwise you should enter the current humidity **4** as precisely as possible.

Press *OK* to accept the displayed values or *Cancel* to abort.

15.4 Laser Control

In the Tracker menu select *Laser Control*. The following dialog opens:
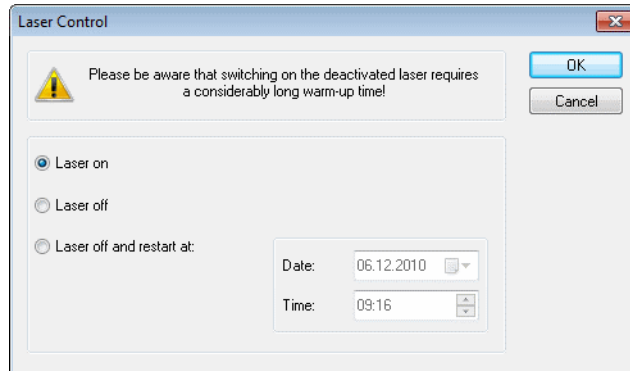
Laser On
This will turn the laser on. Please note that for trackers before AT generation, turning on the laser will take a warm-up time of 15 - 20 minutes.

Laser Off
This will turn the laser off. Please note that after the laser has been turned off, tracker systems before AT generation will only be ready to measure after a warm-up time of approximately 15 - 20 minutes.

Laser Off and restart at
You can use this option to immediately turn off the laser and to turn it back on at a specified time to avoid unnecessary delays before starting to work.

15.5 Reflectors

In the Tracker menu select *Reflectors* ... This brings up the following dialog:
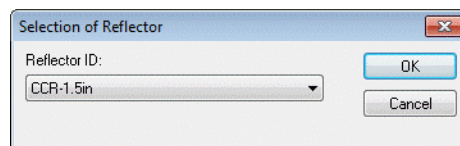
Choose the type of reflector to be used and click *OK* to activate the reflector in the software.

15.6 Information

In the Tracker menu select Information.

You cannot change the displayed values. However the information is important for service and support purposes.

15.7 Go Birdbath

This command sends the laser beam to the Home Point (Birdbath) to lock the beam onto a reflector. The reflector has to be placed onto the magnetic holder of the Home Point in advance.

15.8 Go Zero Position

This command sends the laser beam into a position in front of the Laser Tracker where the beam can be caught with a prism of the T-Scan or T-Probe.

15.9 Move Station

The wizard for moving the station assists you when you need to (repeatedly) move the tracker, for example because of the size of the object. Before and after changing the position of the tracker, reference points need to be measured with the T-Probe. Based on the so created point lists, transformations will be calculated which allows you to measure the object in one coordinate system.

15.9.1 <u>Measuring the Pass Points</u>

Start measuring the data as usual. After you have acquired all data from this tracker position select *Move Station* from the standard toolbar or choose *Tracker → Move Station*. A wizard will open.

If the wizard is opened for the first time, the page *Measuring the pass points* will appear. If possible, measure **all** pass points which are accessible from this tracker position. For this, the following elements are available:

ID
You can define what ID the next point to be measured will obtain.

Measure point
Use this button to measure a pass point with the T-Probe. When measuring, make sure that the ID is correct and change it, if necessary.

From Scene
All points from the data explorer are inserted into the table.

Delete
Use this button to delete the currently selected pass points from the list.

You can also delete selected pass points by pressing <D> or <Del>.

Delete All
Use this button to delete all points in the list.
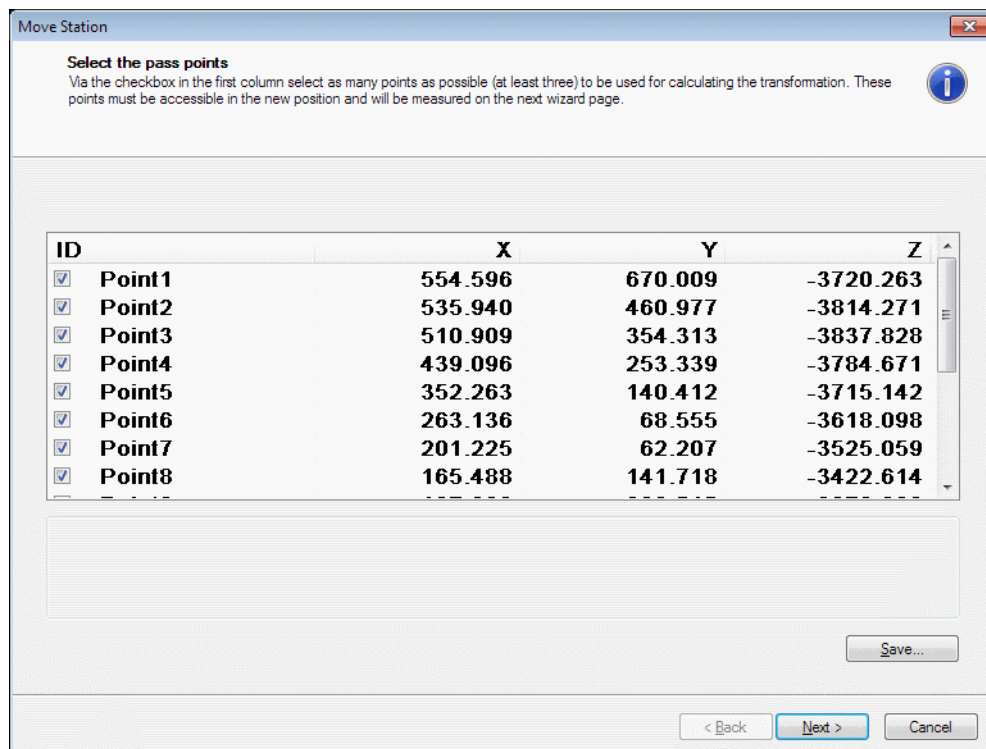
Load / Save
Opens a file selection dialog to load or save a pass point list.

You need to acquire at least three pass points to be able to continue. Make sure the pass points are spatially well distributed and do not form a straight line or an equilateral triangle. The more pass points you acquire, the better the determination of the transformation will be.

Press *Finish* after you have acquired all pass points that are accessible from this tracker position. The wizard is then closed for the time being, giving you the possibility to acquire and manipulate data. To continue with the changing of the setup, start the wizard again.

15.9.2 Selection of Pass Points

The next time the wizard is started, the page *Select the pass points* is displayed. This page displays all the pass points you measured before and offers you the possibility to select points. The selected points will be measured in the next step of the wizard and used to calculate the transformation. To obtain optimal transformation results, select as many pass points as possible by checking the checkboxes in the first column. When at least three pass points have been selected, you can press the *Next* button to continue.

15.9.3 Moving the Station

Change the tracker position now. You can also change the object's position as long as you are sure that all reference points have a fixed relation to the object (i. e. the reference points do not move in respect to the object; this, for example, is the case if the reference points are attached to the object itself).

Press *Next*.

15.9.4 Measuring the Pass Points

You can now measure or import the pass points you selected on the previous page. You can also measure other points (which you might need if you want to change the setup a second time). When at least three points are entered, you can press the *Next* button to proceed to the *Transformation results* page.

Move Station

Measure the pass points
Measure or load all accessible points of the new position.

ID	X	Y	Z
	546.534	-194.516	-3231.342
	536.581	-198.966	-3120.320
	589.647	-206.452	-2980.131
	622.935	-261.727	-2888.688
	635.509	-389.155	-2847.344
	611.683	-536.548	-2900.016
	686.123	-633.286	-2875.509
	685.398	-684.085	-2873.959
	589.860	-633.651	-2950.948
	575.281	-556.273	-2994.634
	520.127	-598.597	-3136.923
	554.321	-628.446	-3044.641

Measure point — From scene — Delete — Delete all — Load... — Save...

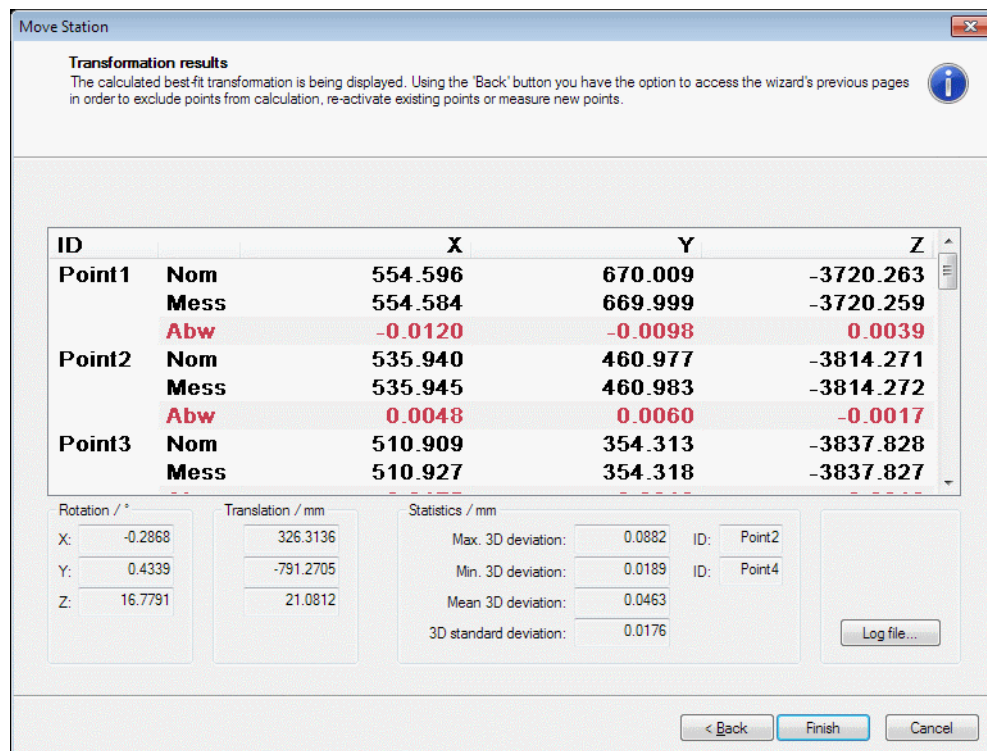< Back — Next > — Cancel

ID

The ID column is initially empty because the assignment of IDs will be calculated automatically in the next step (transformation results).

15.9.5 <u>Transformation Results</u>

The calculation of the transformation automatically determines which points correlate with each other. On completion of this calculation, the points acquired in the previous step *Measure the pass points* automatically obtain an ID. Points for which no correspondences could be determined, are not assigned an ID.

The transformation results page shows you the calculated best fit transformation as well as statistics for the best fit.

<u>Columns: ID | X | Y | Z</u>

The first column of the table shows the ID, the other columns show the X, Y and Z coordinates. The table only lists the IDs for which correspondencies were determined.

Three rows are displayed for each ID:

- Nom: This row shows all the nominal values.

- Act: This row shows the measured points after they have been transformed to the nominal points.

- Dev: This row is displayed in red on a gray background and shows the deviation between actual point and nominal point.

Rotation / Translation

These fields display the calculated rotation (in degrees) and translation (in millimeters).

Statistics

The statistics provide information on the best fit. The first line shows the maximum 3D deviation and the ID for which the deviation was determined. The second line indicates the minimum 3D deviation and the corresponding ID, followed by the mean 3D deviation and the 3D standard deviation.

By pressing the *Back* button, you can access the previous pages of the wizard and make modifications. For example, you can exclude the ID that shows the maximum 3D deviation from the calculation by returning to the page *Select the pass points* and deselecting that specific point. Alternatively, you can also measure the point again.

Press *Finish*. A new group named "Setup1" is created in the data explorer. All existing data sets are moved to this group. Another group named "Setup2" is also created and activated.
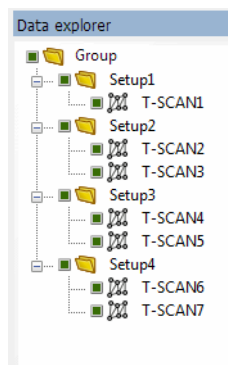
Note

Do not move data sets in or out of the "Setup" groups. Data sets in one of these groups were measured in a specific tracker position and should remain in that group. However, you can still create subgroups and delete or move data sets within the "Setup" groups.

15.9.6 Invoking Move Station again

If the wizard is started anew, the page *Select the pass points* (and not *Measure the pass points* of the initial position) will open after pressing the button *Next* because the reference points for the current tracker position have already been acquired during the last execution of the wizard. This also applies if you restart the T-Scan Collect application and load a file that involved working with the wizard.

15.10 Release Motors

The Release Motors command unlocks the Tracker head if it was locked before, e.g. on a specific target.

The user is now able to point the laser to another position by moving the Tracker head manually.

16 Digitize

16.1 Start

To start a measurement, click the *Start* button in the standard toolbar or choose *Digitize →
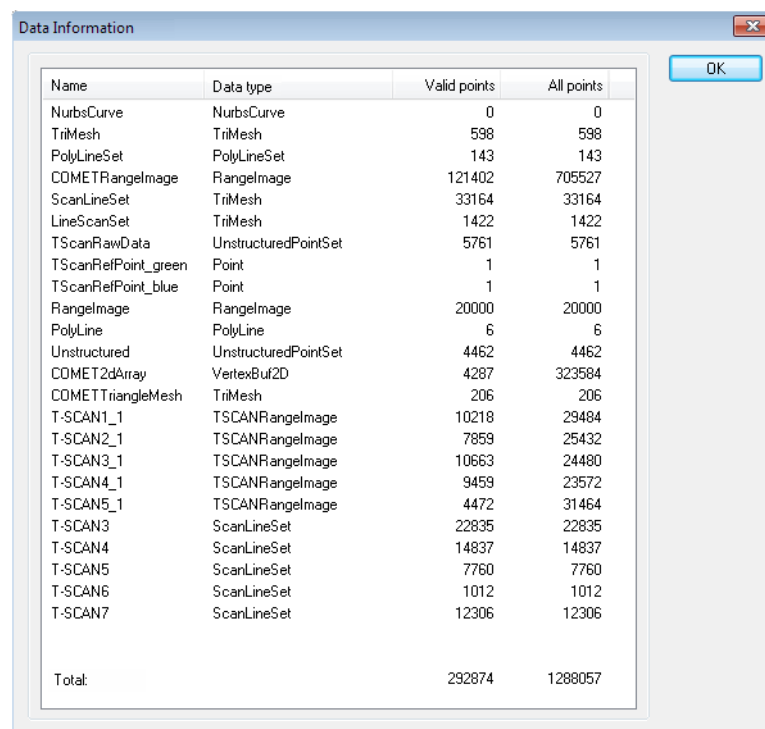Start*.

See chapter 7 for more detailed information on measuring.

16.2 Stop

To stop a measurement, click the *Stop* button in the standard toolbar or choose *Digitize →
Stop*.

16.3 Information

To display information about the currently loaded data, click the *Data Info* button in the
standard toolbar or choose *Digitize → Information*. An information box with details about the
data sets will appear, looking something like this:

The *Name* column contains all the entries that are also shown in the data explorer.

The *Data type* column shows the type of each data set.

The *Valid points* column indicates the number of valid points of a data set. Some data types (e.g. RangeImages) may also contain invalid points.

The *All points* column shows the total number of points of a data set, including invalid points. Since invalid points take up memory, too, this column indicates the total amount of memory used by a data set.

17 Settings

17.1 Cross Sections

To open the dialog for configuring the cross section settings, choose *Settings* → *Cross Sections....*
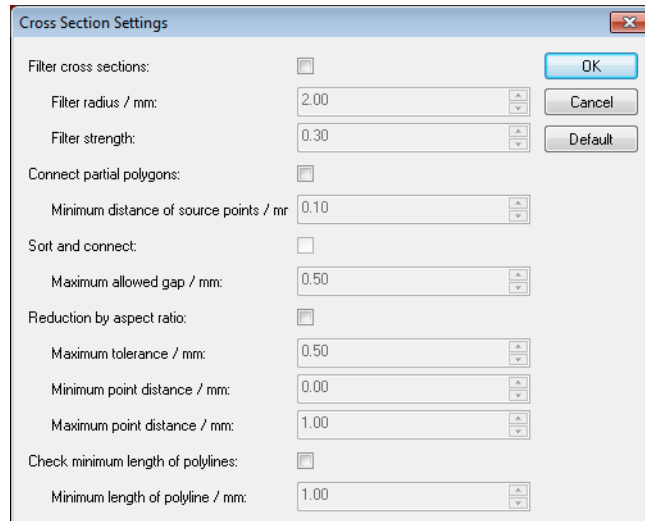
The different fields have the following meanings:

Filter cross sections

Select this checkbox to smooth the point data of the cross section with a Gaussian filter.

All points within the selectable *Filter radius* are filtered. Depending on the distance of the points to the unfiltered original point, these points are weighted according to the Gaussian distribution. The weighting factor of points with the *Filter radius* distance (and, thus, the mathematical function of the Gaussian distribution) is defined by the *Filter strength* parameter. This allows the creation of a wide variety of filters, e.g., a true average filter by entering a filter strength of 1.0. Points for which the distance **d** is greater than *Filter radius* are not considered for filtering.
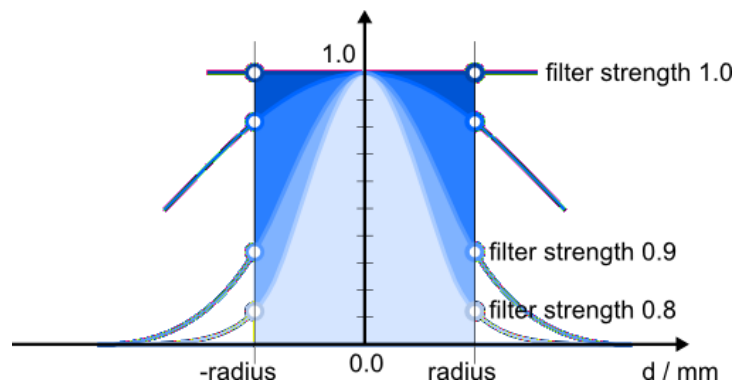
Connect partial polygons

Select this checkbox to connect cross section segments or point cloud segments of several data sets to a closed polyline. With *Maximum distance of source points*, you specify the point distance on the polyline to be created. Please enter a value that is greater than or equal to the point distance of the original data.

Sort and connect

Select this checkbox to sort the individual partial cross sections (scans) in connected polygons and to fill gaps between partial cross sections. This option is only available if the *Connect partial polygons* checkbox is selected. The *Maximum allowed gap* parameter defines the distance in mm up to which gaps between consecutive partial scans will be connected by a straight line. Please note that no new points are generated on these connections, but that the last point of the first scan is followed by the first point of the next scan.

Reduction by aspect ratio

Select this checkbox to reduce the point data of the cross section segments based on the aspect ratio.

The reduction by aspect ratio analyzes the distance between the point before and the point after the current point, as well as the distance from the current point to the line connecting the predecessor and successor points (*Maximum tolerance*). A point is treated in the following way (and in the following order):

Distance between predecessor and successor less than *Minimum point distance*:

$\Rightarrow$ Remove point

Distance predecessor - successor greater than *Minimum point distance*:

$\Rightarrow$ Keep point

Maximum tolerance less than allowed

$\Rightarrow$ Remove point

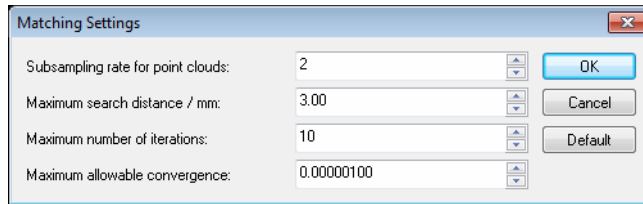*Maximum tolerance* greater than allowed

$\Rightarrow$ Keep point

Tolerance for Point Clouds

This field is currently not used.

17.2 Matching

To configure these settings, choose *Settings* → *Matching...*. The following dialog appears:

The fields have the following meanings:

Subsampling rate for point clouds

Define the subsampling rate you want to use for matching. Since the data for the matching has to be duplicated and processed, the additionally required memory very much depends on the subsampling rate entered here. In addition, the parameter influences the computing time for the matching.

Maximum search distance

This parameter defines the distance within which data from other measurements is considered for the matching. Here, a value between 0.5 and 10.0 should be set. Higher values should only be used in unfavorable cases. Typically the greater the search distance, the longer the time needed for the calculation.

Note

To speed up the matching process, start with a large search distance (e.g. 10 mm) and a high subsampling rate (e.g. 10). Then reduce the search distance (e.g. 2 mm) and the subsampling rate (e.g. 2). Use this iterative method particularly when dealing with large data sets.

Maximum number of iterations

With this parameter, you determine how many iterations maximum are to be executed during matching. If the convergence criterion described below is reached earlier, the matching is stopped before the maximum number of iterations is reached.

Maximum allowable convergence

The convergence criterion defines the minimum distance by which a data set must be shifted during an iteration so that the iteration is continued. The distance is set in millimeters.

Example: You have specified a convergence criterion of 0.0001 (0.1 micrometers). You match 10 data sets by global matching. The algorithm now tries to improve the position of each data set. If at least one data set is moved by more than 0.0001 mm, another iteration is executed – except if the maximum number of iterations has been reached.

17.3 3D Viewer

T-Scan Collect provides two different display modes. The static display mode describes the display of the still object. If the object is moved, e.g. by rotation or zooming, it is displayed in the dynamic display mode.

You can set the visualization parameters separately for the two modes. Typically, a higher subsampling value is used in the dynamic mode to allow smooth rotation and zooming even for large objects.

To adjust these settings, choose *Settings → 3D Viewer….*
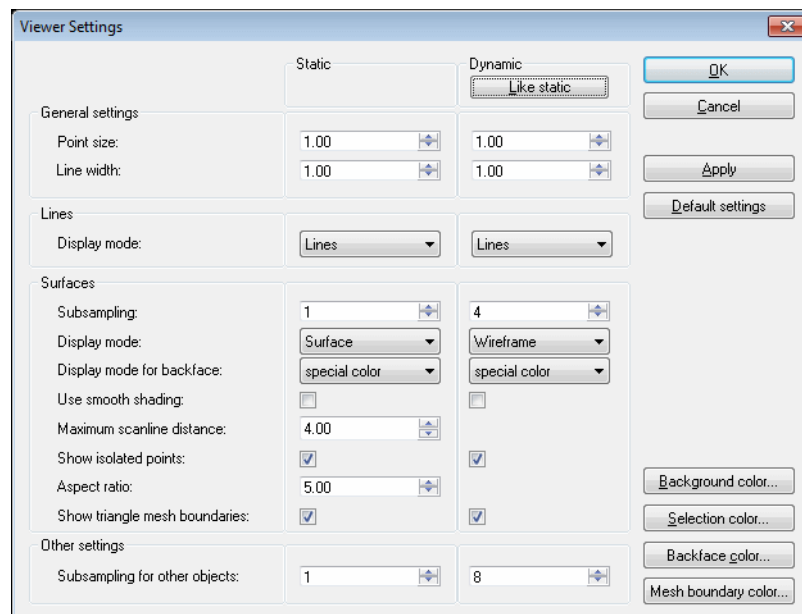
The fields have the following meanings:

17.3.1 <u>General Settings</u>

<u>Point size</u>
This value defines the size in which a 3D point of the point cloud is displayed on the monitor (in monitor pixels).

<u>Line width</u>
This value defines the width in which a line is displayed on the monitor (in monitor pixels).

17.3.2 <u>Lines</u>

<u>Display mode</u>
Select whether lines should be displayed as lines, as lines with points, or as points.

17.3.3 Surfaces

Subsampling

This parameter influences the reduction of the data for visualization. A subsampling value of 4 means that only every 4th point in the row and column directions of the surface is displayed and, thus, detail resolution is reduced by the factor 16 in the visualization.

Display mode

Select whether you want to display the measured surface as a filled surface **1**, as a surface with wireframe **2**, as a wireframe **3** or as points **4**. The shading is not influenced by this selection. However, points can generally be visualized faster than surfaces or wireframes.
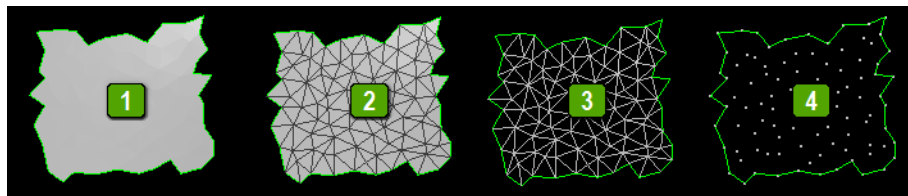
Display mode for backface

Select how you want the back of an object to be displayed. You can either select the same color as for the front (like the front or darker), or you can choose a special color.

Use smooth shading

If this checkbox is selected, the surfaces are smoothly shaded, i.e., the edges of the triangles merge. Usually, this results in the best surface display, but takes longer. For very large point clouds, it is therefore recommended not to use this option.

Maximum scanline distance

If the actual scan line distance exceeds the entered value, points will not be displayed as surfaces but as isolated points. A value of 0 results in all points being displayed as isolated points. To obtain a good representation of the data as surfaces, this value should be at least 3 times the value of the line to line distance (see chapter 0).

Show isolated points

If the Measurement Settings for the Line Scanner is set to *Surfaces* (see chapter 7.2.2) and the display mode is set to Surface, some points might not be displayed as part of a surface but as isolated points. This effect may diminish the appearance on the screen. With the *Show isolated points* checkbox, you can determine if you want to show or hide these points.
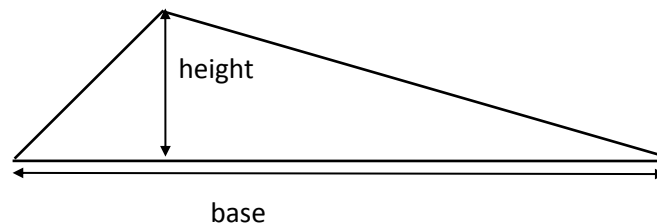

Note

Show isolated points *only refers to the visualization. Even if* Show isolated points *is not active, isolated points will still be used for calculations (e.g. cross sections, masking and clipping).*

If you uncheck Show isolated points *and* Maximum scanline distance *is too small, it may happen that no data is displayed even though there is data. Therefore, you should preferably always have this option checked.*

Aspect ratio

Extremely acute-angled triangles are not displayed. Such triangles will be created, for example, if measurement data is acquired with the scanner across an object edge, because neighboring pixels will then lie on different planes.

A deformed triangle is defined by the aspect ratio (base to height).

height

base

Reasonable values are between 3 and 20.

Show triangle mesh boundaries

If this checkbox is selected, the boundaries of triangle meshes are displayed in the specified color.

Note

This setting only applies to 2D arrays (this data structure cannot be created but only imported).

17.3.4 Other Settings

Subsampling for other objects

This parameter influences the reduction of the point clouds for visualization. A subsampling value of 4 means that only every 4th point in the row and column directions of the objects is displayed and, thus, detail resolution is reduced by the factor 16 in the visualization.

Background color

Sets the color for the background of the different viewers in a color selection dialog.

Selection color

Sets the color for selected objects in a color selection dialog.

Backface color

Sets the color for the back faces of surface objects in a color selection dialog. Please note that this color is only used if you have chosen *special color* in *Display mode for backface*.

Mesh boundary color

Sets the color for triangle mesh boundaries in a color selection dialog.

17.4 Dynamic Camera

Typically, the scene in the 3D viewer is displayed statically while the wireframe of the scanner – if displayed – is moved. However, if you use Dynamic Camera, the position of the scanner (after the measurement has started) determines what is displayed in the 3D viewer. By "fixing" the view to the scanner, the scene is continuously moved while the wireframe of the scanner – if displayed – always stays in the middle of the 3D viewer.

To toggle the dynamic camera and to define the settings, choose *Settings → Dynamic Camera…*. The following dialog appears:
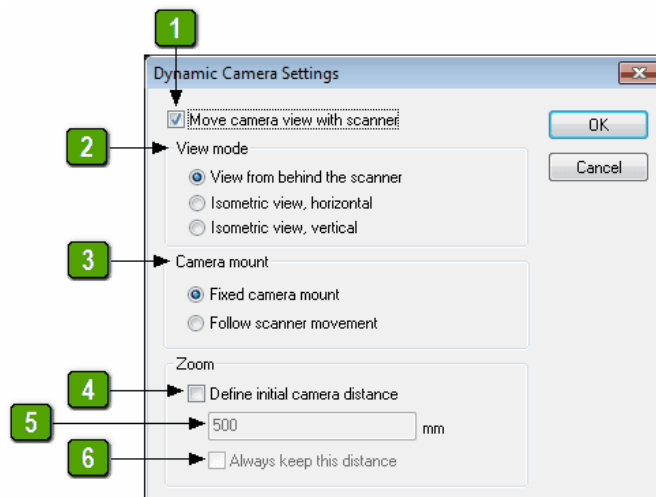
The checkbox *Move camera view with scanner* [1] determines if the dynamic camera is active or not.

If the option *Settings → Adjust size automatically* is active, it will be disabled automatically and you will be notified by a message. This is because using this option contradicts using the dynamic camera.

If *Move camera view with scanner* is active, the following options are available:

View mode [2]

View from behind the scanner: The perspective is chosen in a way that you are always viewing the rear of the scanner from behind.

Isometric view, horizontal: The perspective is chosen in a way that you are always viewing the left, rear, bottom side of the scanner.

Isometric view, vertical: The perspective is chosen in a way that you are always viewing the top, left, rear side of the scanner.

Camera mount 3

Fixed camera mount: The view is fixed to the wireframe of the scanner; the scanner is displayed statically in the middle of the 3D viewer.

Follow scanner movement: The view follows the scanner's movement slightly delayed. The visualization looks smoother and the wireframe of the scanner moves approximately in the middle of the 3D viewer.

Zoom

Define initial camera distance 4 : As soon as you activate the measurement mode and the wireframe of the scanner is displayed, the distance from the camera to the wireframe can be defined. For this, select the checkbox and enter a value 5 . During the measurement, you can adjust the camera distance using the mouse (mouse wheel or middle mouse button) as usual.

If you would like to always keep the entered distance, check *Always keep this distance* 6 . The camera distance then is continuously updated so that changes caused by the mouse (mouse wheel or middle mouse button) will be rejected immediately.

Note

While you keep the trigger button of the scanner pressed, the camera distance will be cut in half so that more details of the area to be scanned can be seen. When the trigger button is released, the previous camera distance will be restored to give you a better overview.

17.5 Coordinate Axes

The settings for the coordinate axes are described in chapter 5.3.

17.6 Accuracy Check

To adjust the settings for the accuracy check, choose *Settings → Accuracy check…*. The dialog box that appears is described in chapter 8.3. Please refer to that chapter for detailed information on the individual parameters.

17.7 Measuring Mode

To change the measurement mode, select *Settings → Measuring Mode*. A dialog box which corresponds to the Measurement toolbox appears. Please see chapter 7.1 for further information.

17.8 Measurement

To change the measurement settings, choose *Settings* → *Measurement*. The dialog box that appears is described in chapter 7.1. Please refer to that chapter for detailed information on the individual parameters.

17.9 Acoustic Feedback

Choose *Settings* → *Acoustic feedback* to assign sound files to specific states or events:
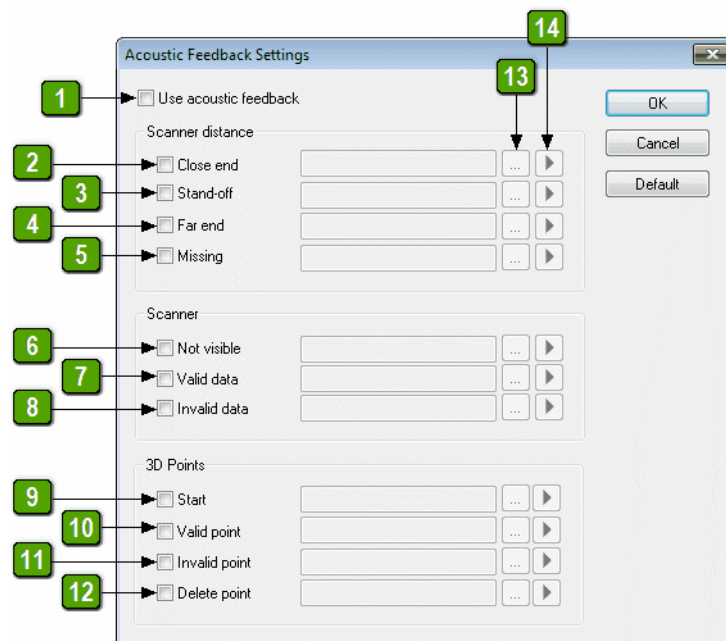
1 Use this button to toggle between three states: All entries are activated; all entries are deactivated; all entries are set to their last state.

The following entries are currently not supported and therefore disabled:

2, **3**, **4**, **5**

The following entries can be customized:

6 The scanner cannot be detected by the tracking system, e.g. because it is hidden by an object

7 After the trigger button is released, the resulting scan data contains valid points

8 After the trigger button is released, the resulting scan data contains NO valid points

9 A measurement with the T-Probe is started

10 A T-Probe measurement was successful, resulting in a valid point

11 A T-Probe measurement was not successful, no point was created

12 A point acquired with the T-Probe is deleted

Note

Please note that the .wav file format allows a lot of different internal settings. T-Scan Collect does not support all of these versions, so it may happen that certain files cannot be used or played, although other media players can play them.

Files saved in the following format can be used in T-Scan Collect: Microsoft WAV signed 16bit PCM format (mono or stereo, 44100 Hz or 22050 Hz).

17.10 Materials

To be able to measure various materials optimally, it might be necessary to adjust specific parameters of the sensor. This can be achieved using the Materials editor. To open it, select *Settings → Materials*. The following dialog opens:
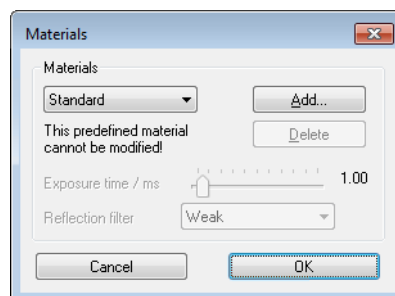
By default, the following predefined materials are available: *Standard, Shiny bright, Dark* and *Shiny dark*.

The predefined materials cannot be modified. However, by pressing the button *Add* you can create new materials: a dialog asking for the name of the new material will open. After pressing *OK*, the entries for *Exposure time* and *Reflection filter* will become active.

Press *Delete* to delete the currently selected material. Note that predefined materials cannot be deleted.

To accept all changes and choose the currently selected material, exit the dialog by pressing *OK*, otherwise press *Cancel*.

Exposure time
The exposure time is the length of time that the camera of the line scanner will be exposed. Adjust the exposure time to suit the object you will be measuring. Use short exposure times (in

a range from 0.25 to 5 ms) for bright objects and longer exposure times (up to 20 ms) for dark objects.

If the exposure time is not adjusted to suit the object's reflection properties, it may happen that no measurement data is generated or that there are gaps in the measured data. An optimum exposure time is the shortest time by which the acquired data is complete and independent of the scanner orientation and the distance to the object.

Note

Please note that the exposure time also influences the data rate. With longer exposure times, less data can be acquired per second. If necessary, adjust the movement speed of the sensor.

Reflection filter

The various reflection filter types are optimized for characteristic reflection properties and can be used to eliminate artifacts or single points which were generated by reflections from specular surfaces or by extraneous light and which do not lie on the object's surface.

The pre-defined filter types are suitable for any measurement task especially for specular surfaces. Choose that type of filter which suits the object you will be measuring. If you are unsure, use the *Weak* reflection filter.

Note

Filter types Standard *or* Weak *can be used for all diffuse reflecting surfaces where* Weak *will typically remove more points than* Standard*. Filter type* Standard *is especially suited for measuring object features since it largely maintains measured points close to edges and borders of an object. For specular surfaces with scattered or secondary reflections filter types* Weak *or* Medium *should be chosen. Besides the majority of erroneous points filter type* Strong *might also remove a lot of „good quality" points. For that reason it should be used advisedly.*

17.11 Adjust Size Automatically

If *Adjust size automatically* is activated, the viewer automatically calculates settings so that all data is completely shown in the current viewer. This is helpful if you start digitizing a new object. Disable *Adjust size automatically* to restore your current viewer settings, e.g. if you want to do detail work*.

To toggle the automatic size adjustment, choose *Settings → Adjust size automatically*.

17.12 Display Scanner

This setting only applies to the surface scan mode (see chapter 7.3.1). When *Display scanner* is active, a wireframe representing the scanner is displayed if the scanner is within the field of view of the tracker. This is especially helpful to check if the scanner can be detected or if it will be hidden by some object.
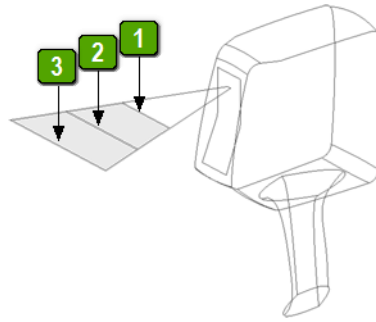
The start ■1, middle ■2 and end ■3 of the scanner's measurement range are indicated by lines. This is especially helpful for controlling the scanner's distance to the object.

To toggle the display of the scanner position, choose *Settings* → *Display scanner*.

17.13 Display T-Probe

This setting only applies to the 3D point mode and the static polyline mode. When *Display T-Probe* is active, a wireframe representing the T-Probe is displayed if the T-Probe can be detected by the tracking system.

To toggle the display of the T-Probe, choose *Settings* → *Display T-Probe*.

17.14 Display T-Probe Coordinates

This setting only applies to the 3D point mode and the static polyline mode. When *Display T-Probe* is active, a window showing the coordinates of T-Probe's origin will be opened when a measurement is started.
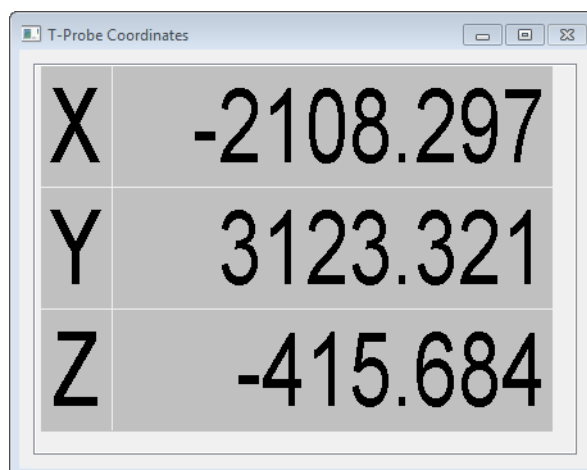
You can resize, move, minimize or restore the window.

17.15 Keep On Top

When *Keep on top* is active, T-Scan Collect will always become the active application as soon as the trigger button on the handle of the scanner is pressed.

18 Extras

18.1 Execute Macro

To execute macros, choose *Extras* → *Execute Macro....*

A file selection dialog appears where you can choose the macro to be executed. Click the *Open* button to execute the selected macro.

19 Help

You can open this manual by pressing <F1> in the T-Scan Collect application. The status bar and the tooltips in the application provide additional helpful information.

19.1 About T-Scan Collect

To display information about T-Scan Collect, choose *?* → *About T-Scan Collect ….* A dialog appears which will look something like this, depending on the components you are using:

[1] Click the *Open license administration* button to open the CodeMeter WebAdmin, which provides information about the currently connected dongle.

[2] Click the *Request for license update* button to create a current license file of your dongle; e.g. *x-xxxxxxx.WibuCmRac*. You can then send this file to support.tracker@hexagonmetrology.com to request a software upgrade.

[3] Displays the serial number of the currently used Tracker and Scanner.

[4] With the buttons, you can display additional information about the individual hardware components. This information is particularly helpful for support.

Note

When you receive the new license file, e.g. x-xxxxxxx.WibuCmRau, save this file and execute it – with the dongle connected – by double-clicking the file. The license will then be activated on the dongle.

20 Appendix

20.1 DCOM Configuration

The COM port of T-Scan Collect can be set to allow sharing over the network. This setting is necessary e.g. to execute post-processing on a network computer ("external post-processing").

The steps described in this chapter require experience in DCOM configuration and security policies. If you are unsure or if you cannot make these settings due local security policies, please ask your system administrator to do the DCOM configuration for you.

Install the same version of T-Scan Collect on both computers. Since the application registers itself on startup, it has to be launched at least once on each machine (on Windows 7, the software has to be started with administrator privileges to register successfully), before you can execute external post-processing.

Ensure that the same user account (i.e. name, group and password) exists on both computers; if this is not the case, please create the required user accounts.

The following steps describe how to configure the DCOM settings under Windows 7. For other Windows versions, please proceed accordingly.

- Start dcomcnfg.exe
- Navigate to My Computer and open Properties:
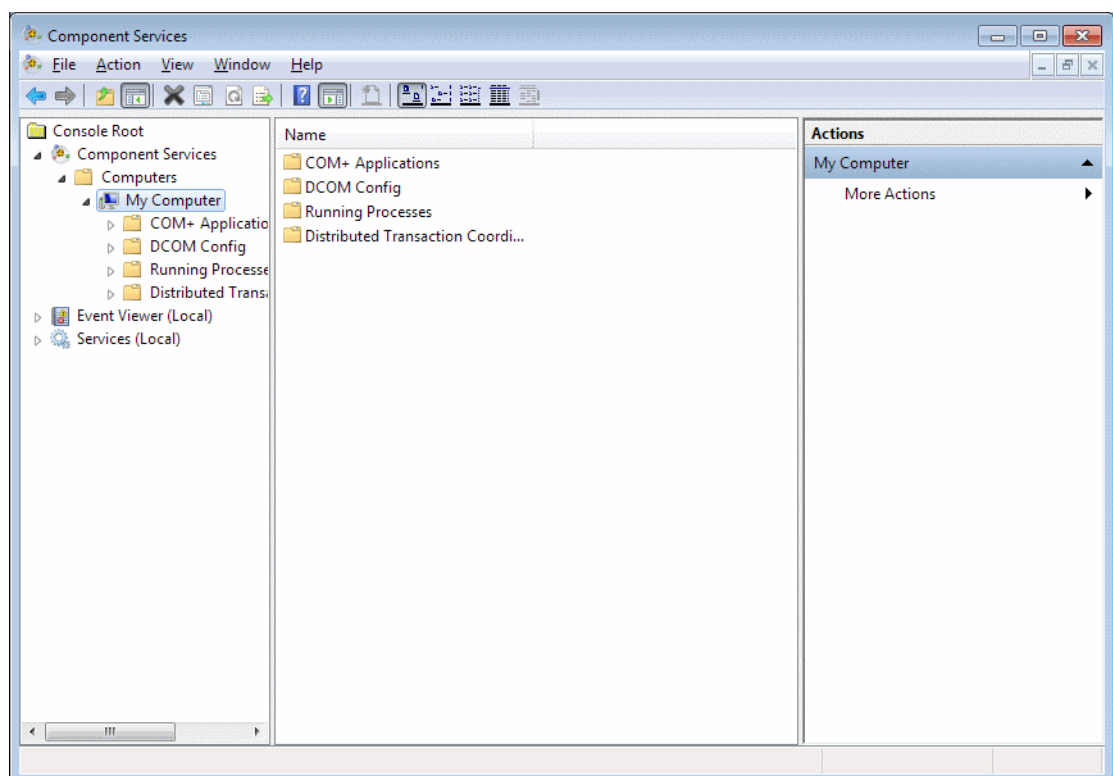
- Select the COM Security tab and click *Edit Limits…* under Access Permissions. Add the user Everyone and set all checkboxes to *Allow*.
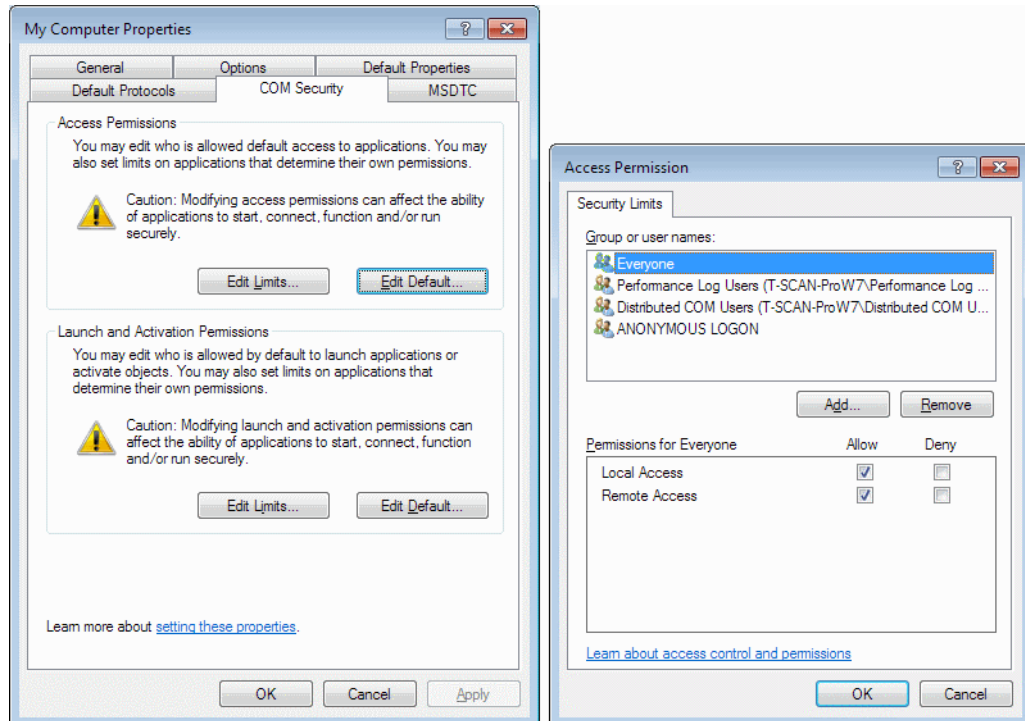
- On the COM Security tab, click *Edit Limits…* under Launch and Activation Permissions. Add the user Everyone and set all checkboxes to *Allow*.
- Navigate to DCOM Config and open the properties of TSCANCOLLECT.NoSensor.Application.
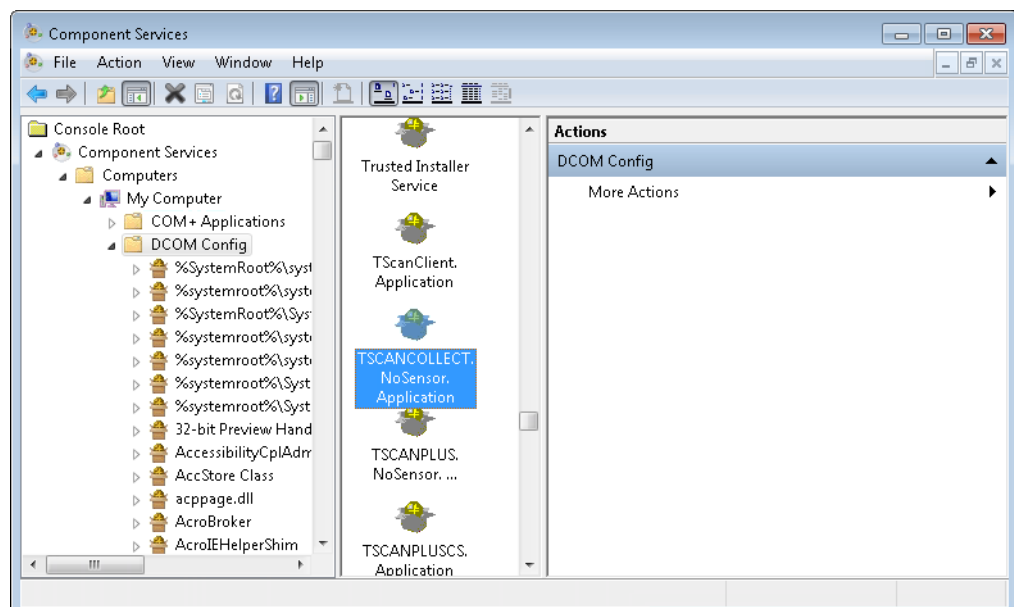
- Switch to the Security tab and select *Customize* for both, the Launch and Activation Permissions and the Access Permissions.
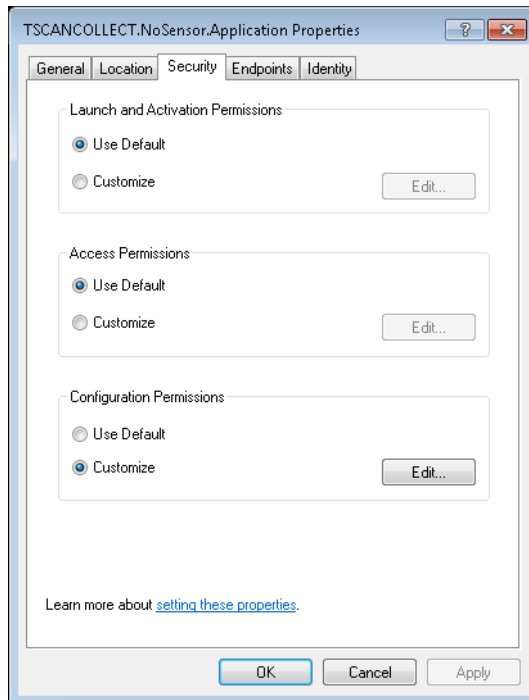
- Click *Edit* under Launch and Activation Permissions, add the user <u>Everyone</u> and set all checkboxes to *Allow*.
- Click *Edit* under Access Permissions, add the user <u>Everyone</u> and set all checkboxes to *Allow*.
- Switch to the Identity tab and select *The interactive user*.
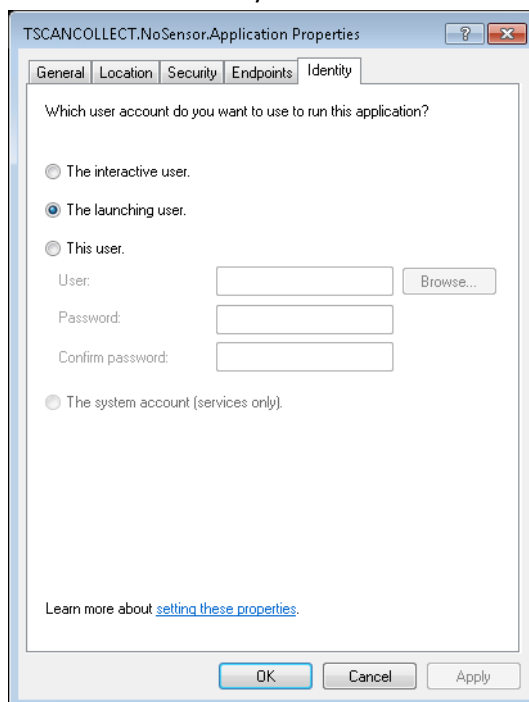
20.2 T-Scan Collect vs. T-Scan Interface

	Collect 10.3x	Interface 10.3x
Toolbars		
New	✓	✓
Open	✓	✗
Save	✓	✗
Save as	✓	✗
Import	✓	✗
Export	✓	✓ (AC only)
Postprocessing	✓	✗
Start	✓	✓
Stop	✓	✓
Move station	✓	✗
Info	✓	✓
Selection	✓	✓
Clipping	✓	✓
Accuracy	✓	✓
Cross sections	✓	✗
Fill holes	✓	✗
Create bridges	✓	✗
Smoothing	✓	✗
Decimation	✓	✗
Sharpening edges	✓	✗
Alignment	✓	✓
Measuring range	✓	✓
Reference points	✓	✗
Data analysis	✗	✗
File Menu		
New	✓	✓
Open	✓	✗
Save	✓	✗
Save as	✓	✗
Import	✓	✗

Export			✅	✅ (AC only)
Exit			✅	✅

Edit Menu

Transform data				
	Transform		✅	❌
	Transform to nominal points		✅	❌
	3 planes alignment		✅	❌
	3-2-1- alignment		✅	❌
	Scale		✅	❌
	Mirror		✅	❌
Optimize data				
	Smoothing		✅	❌
	Decimation		✅	❌
Matching				
	Global matching		✅	❌
	Tolerance based matching		✅	❌
Optimize triangle mesh				
	Remove unassociated areas		✅	❌
	Remove outliers		✅	❌
	Noise reduction		✅	❌
	Curvature based decimation		✅	❌
	Smoothing		✅	❌
	Flip inside/outside		✅	❌
Feature lines			✅	❌
Postprocessing			✅	❌
Interactive tools				
	Selection		✅	✅
	Clipping		✅	✅
	Accuracy		✅	✅
	Cross sections		✅	❌
	Fill holes		✅	❌
	Create bridges		✅	❌
	Smoothing		✅	❌
	Decimation		✅	❌

Sharpening edges	✔	✘
Alignment	✔	✔
Measuring range	✔	✔
Reference points	✔	✘
Data analysis	✘	✘
Service Menu		
Adjust alignment	✔	✔
Start data collection	✔	✔
Contour filter	✔	✘
Accuracy analysis	✔	✔
Change password	✔	✔
Activate/Deactivate	✔	✔
View Menu		
Toolbar	✔	✔
Statusbar	✔	✔
Customize	✔	✔
Reset	✔	✔
Left <-> Right	✔	✔
Tracker Menu		
Initialize	✔	✔
Disconnect/Connect	✔	✔
Environmental parameters	✔	✔
Laser control	✔	✘
Reflectors	✔	✔
Information	✔	✔
Go birdbath	✔	✔ (EmScon only)
Go zero position	✔	✔
Move station	✔	✘
Release motors	✔	✔ (EmScon only)
Digitize Menu		
Start	✔	✔
Stop	✔	✔
Information	✔	✔
Settings Menu		

Cross sections	✓	✗
Matching	✓	✗
3d viewer	✓	✓
Dynamic camera	✓	✗
Coordinate axes	✓	✗
Accuracy check	✓	✓
Measuring mode	✓	✓
Measurement	✓	✓
Acoustic feedback	✓	✗
Materials	✓	✓
Adjust size automatically	✓	✓
Display scanner	✓	✓
Display T-Probe	✓	✓
Display T-Probe coordinates	✓	✗
Keep on top	✓	✓
Extras Menu		
Execute macro	✓	✗
? Menu		
Help	✗	✗
About T-Scan Collect/Interf.	✓	✓

21 Index