Leica Infinity

Help Manual
Version 4.0
English

- when it has to be **right**

Leica Geosystems

PART OF HEXAGON

1 Overall

1.1 Welcome to Leica Infinity

1.1.1 What is Infinity

What is Infinity

Infinity, the intuitive office software solution from Leica Geosystems. Infinity is the ideal place for managing data to and from field instruments, the bridge between field and office:

- Bring together different kinds of data on a field project. TPS, GNSS, images, scans and so much more.
- Numerous data formats can be edited, archived and exported to CAD applications easily, without data loss and without the hassles involved in conversion. It is not only for data from different instrument types, but also from multiple sites and survey teams.
- Infinity structures and processes surveying data for simplified, automated and logically expanded workflows, for a faster overview of the project as a whole.

More efficiency, greater transparency, quicker decision making and simply better.

The aim of Infinity is to support the user throughout a survey project by fulfilling the following five principles:

- **Visualise** - Become familiar with the data and see visually how it is related to each other.
- **Prepare** - Extract and prepare data for the field.
- **Confirm** - Process data, perform quality checks and view results.
- **Report** - Generate and store, proof of quality and completed task reports.
- **Archive** - Know where exported data and reports are stored.

Watch the Infinity overview video which gives you a broad understanding about what modules exist in the software https://www.youtube.com/watch?v=un5WH-sF6I8&list=PL0td7rOVk_IV_al3ziSKuAYA1VVu6W0rM&index=11&t=0s

☞ Some of the features used in the video are licence protected.

To find help, refer to the following:

- Use the Contents tab to browse to a topic.
- Use the Search tab to search for key words.
- Select a link from a topic to go to the related topics.

1.1.2 Training Materials

Training Materials

For training purposes refer to the following training materials:

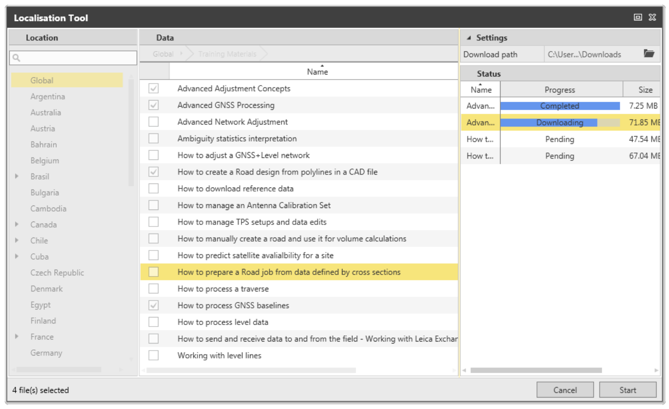
- Video tutorials.
- Tutorials, downloadable in the Infinity software.

Video tutorials

Video tutorials are available on the Infinity YouTube channel:
https://bit.ly/3sZalIK

Tutorial downloads

Tutorials are available for download in the Localisation Tool.

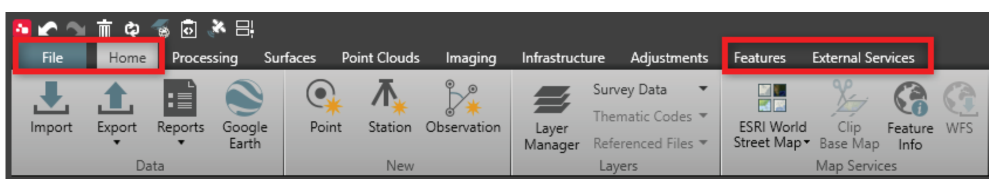


1.2 License Options

1.2.1 Basic Modules

Basic Modules

Basic modules include the core data in and out modules: File, Home, Features and External Services. These modules make for easy field to office or office to field data preparation.



File

- Manage and archive all your Projects.
- Set up local and global settings.
- Send and receive data to and from the field using external services: Leica Exchange, Leica ConX, Autodesk BIM 360.
- Download reference station data from predefined reference stations or by connecting to SmartNet.
- Connect to Map Services: WMS, WMTS, XYZ and WFS.
- Manage and edit coordinate systems.
- Apply styles using the Layer Manager and CodeTables.
- Get Help & Support as well as the access to the Localisation Tool.

Home

- Import and export all supported data formats.
- Generate extensive reports including data source and stakeout.
- View or share data using Google Earth.
- View base maps: With an active CCP use the HxIP accurate orthorectified imagery.
- Check the quality and correcting field errors.
- Use COGO to calculate distance, compute points and shift/rotate/scale.
- Link/unlink images to features, clip base map and georeference image.

Features

- Create and edit thematic code information.
- Automated feature code processing (blocks and layer).
- Create and manage points, lines and areas.
- Copy of entities from CAD, IFC, Shape files and WFS to the project.
- Solve name conflicts using the Rename Tool.

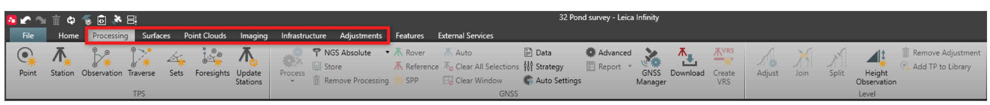
External Services

- Send and receive data to cloud services Leica ConX and Leica JetStream.

1.2.2 Optional Modules

Optional Modules

Optional modules in Infinity, are designed to support the user by grouping product features to workflows. These workflows are grouped to modules, and most modules are optional, allowing you choose what work you perform in Infinity. The optional modules are: TPS-Processing, GNSS-Processing, Level-Processing, Surfaces, Point Clouds (including surfaces), Imaging, Point Clouds from Images (including imaging), Infrastructure and Adjustments.



TPS-Processing

- Create or edit TPS setups to update orientation or positions.
- Support of further point calculations: Sets of angles, measure foresights.
- Build or edit traverses.
- Update stations.

GNSS-Processing

- Process single or multi-frequency GNSS raw static and kinematic data for determining the most reliable and accurate solution.
- View cycle slips, SNR and residual plots with statistics using advanced GNSS data analysis tools.
- Process multiple frequencies of GPS, GLONASS, BeiDou, Galileo and QZSS.
- Use Autoprocessing to build up automatically all possible combination of baselines and process your data within one click.

Level-Processing

- Manage level lines - Edit start/end points, join or split lines.
- Process level lines - Edit staff corrections, reprocess and generate reports.
- Level network adjustment (1D) - Complete levelled height networks.

Surfaces

- Compute 3D or 2.5D surface using individual points and point clouds.
- Add break lines, boundaries or exclusion areas, to edit a mesh.
- Calculate volumes.
- Create contours.
- Cut fill maps and compute tolerance lines.
- Comparison maps.

Point Clouds (including surfaces)

- Measure within point clouds for comparison and checks.
- Visualise scan data in different colour modes - SNR, Intensity, RGB.
- Automatic and manual point cloud cleaning tools.

Imaging

- Manage image data using integrated image group viewer. Sort and organise by groups and features and compute points from images taken from total stations and GNSS sensors.

Point Clouds from Images (including imaging)

- Orientate images.
- Create dense point clouds.
- Generate digital surface models and orthophotos.

Infrastructure

- Import, visualise and organise road design data.
- Repair road data before sending it to the field, for example live edits to road geometry or fixing string line connections.
- Document and report field applications, including stakeout and checks with tolerance flags.
- Manually input a road and compute daylight stringlines.

Adjustments

- Combine TPS, GNSS and level data.
- Full 3D, 2D and 1D computation.
- Compute loops and display of error ellipse and reliability.

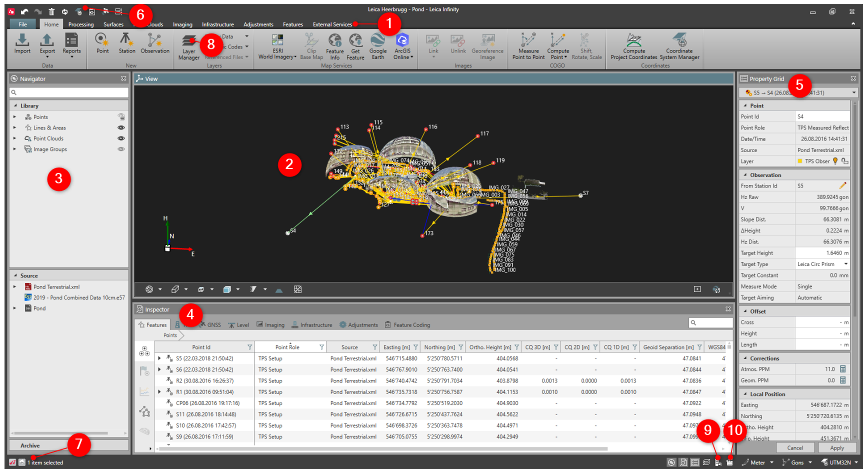
1.3 Getting Started

1.3.1 Working with the User Interface

1.3.1.1 Overview

Working with the User Interface

When we open a project we see the layout of the user interface. The layout of the user interface can be split into several parts which are highlighted in the following graphic.



No.	Name	Description
1.	Ribbon Bar	Explore functions and tools grouped into modules.
2.	Graphical View	View the project content graphically in 2D or 3D.
3.	Navigator	Navigate through the project content.
4.	Data Inspector	View project content in table format.
5.	Property Grid	View the properties of a selected data item.
6.	Quick Access Bar	Get quick access to some important functions.
7.	Status Bar	See current settings.
8.	Layer Manager	Change the layer status and their graphical settings.
9.	BIM Explorer	Inspect the structure of IFC files.
10.	Flythrough Creator	Create video with defined flythrough methods.

See also:

Window Behaviour

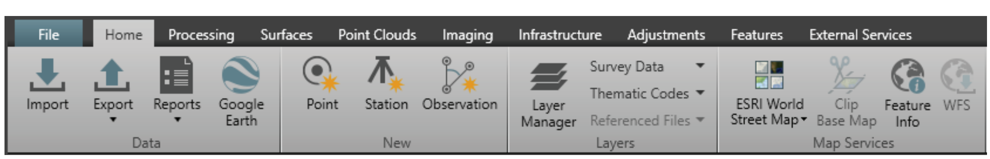
Objects, Point Roles and Symbols

1.3.1.2 Ribbon Bar

Ribbon Bar

The ribbon bar is at the top of the screen and allows access to all the functions and tools.

It is used to group the functions and tools of particular operations together into modules. Each module has its own tab in the main menu at the top. Different tabs allow access to different ribbon bars.



Under the File tab it is possible:

- To perform project management-related operations, such as creating and deleting projects.
- To edit all global settings and global objects.

The Home tab is the core of every project, as that is where the Import, Export and Reports functionalities are located.

For more information, refer to Basic Modules.

Under the Processing, Surfaces, Scanning, Imaging, Infrastructure, Adjustments and Features tabs, operations can be run to work with data and perform edits.

☞ Most of these operations are licence protected and must be ordered separately.

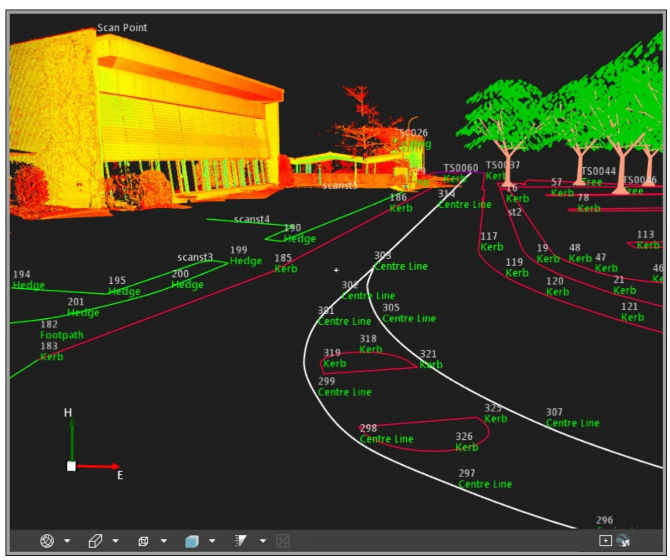
For more information, refer to Basic Modules and Optional Modules.

Under the External Services tab, Infinity manages cloud services such as the Leica ConX and the Publish to Leica JetStream.

1.3.1.3 Graphical View

Graphical View

The graphical view displays the project content graphically in 2D or 3D. It is the main representation of project content and is always visible in the background. It cannot be switched off or hidden.



Zooming/Panning/Rotating the view:

When you open a project, the graphical view is zoomed to the full extent. The whole project content is shown.

To zoom into inspect the data in detail:

Scroll the mouse wheel back and forth to zoom in and out.

Alternatively select **Ctrl+** or **Ctrl-** on the keyboard.

When you have zoomed in and you want to shift the view to neighbouring areas (pan the view):

Press the mouse wheel and move the mouse into any direction until the displayed area suits your needs.

Alternatively press the **Right**, **Left**, **Up** or **Down** arrows on the keyboard.

To rotate the view:

Click into the view with the left mouse button and move the mouse.

The axis in the bottom left corner of the view follows your rotations and indicate the current view direction.

To look around:

By default, the rotation in the graphical view is around the centre of the graphical view, marked by a little white cross. Look around, allows reverse rotation around the current camera position.

Select the  **Look Around** option on the status bar to rotate around the camera position. Click into the view with the left mouse button and move the mouse.

Use **Ctrl+R** to turn on/off the look around.

To lock zooming to the pivot point:

By default, the pivot point is identified by your mouse pointer. When you start zooming the view is zoomed into the area that your mouse points at. But you can also lock the pivot point to the centre of the graphical view marked by the little white cross.

Select the  **Lock the Pivot Point** option at the bottom of the graphical view to lock the direction of view to its centre.

Alternatively, press and hold **Alt** while zooming.

To zoom into the insides of an object:

Switch to  **Perspective View** and zoom until you break through the surface of the object. Once inside you can rotate the view to look at the insides of the object.

Switching between Standard Views:

You can switch between the following standard view options:

Icon	Description
	**2D Plane View:** Select **2D Plane View** to enforce a 2D view from the top down onto the data. The data is displayed as in a map only showing the position information. You cannot rotate the view when it is set to 2D. **3D:** Select **3D** to see your data including its position and height information. **3D Clamped:** Select **3D Clamped** to enforce a projection of the data to the height level of the lowest point in your project.

Icon	Description
	Parallel: Select **Parallel** to enforce a parallel projection of the data in the 3D view. **Perspective:** Select **Perspective** to allow for a perspective view. ☞ Zooming into the insides of an object is only possible in perspective view. ☞ The overview window is only available when the view is set to **Parallel** projection. When you switch to perspective the overview window disappears.
	Select how you want to look at the data, that means select your **Camera** position. In the 3D view, you can freely rotate the view. But if you want to orientate the view such that you can look at your data exactly from, for example, the front or back or from the left or right then you have a choice here. ☞ The 2D view is set to be a top-down view.
	Select whether you want to see the entities shaded (with or without edges) or whether you want to see just the wireframe.
	Select the intensity of shading.
	Enable the 3D terrain in order to view base maps, orthophotos or georeferenced images in 3D. The default **DEM (SRTM)** has a resolution of approximately one arcsecond. ☞ Due to the low accuracy, the DEM is only intended for visualisation purposes.
	Select whether you want the coordinate grid to be shown or hidden.
	Select the **Look Around** feature.

Turn on Google Earth View Synchronisation:

When you turn on **Google Earth View Synchronisation**, then Google Earth follows all zoom, pan and rotation operations you make in Infinity.

Reset the View:

To reset the view to the initial camera position right-click into the background of the graphical view and select **Zoom All** from the background menu.

Alternatively press **Ctrl+Shift+R** on the keyboard.

The zoom status is reset, which means that the view is zoomed to the full extent. Any panning and/or rotation operation is reset, too, so that you have a look at the data as if you had just opened the project.

Managing Layers:

How your data is represented in the graphical view can be modified in the layer manager, which allows you to switch layers on and off, lock them so that they cannot be modified and change the colour and style of their appearance. For more information, refer to Layer Manager.

Hiding Elements from View:

When the data structure of your project is complex you can filter the view by switching elements or whole groups of elements off. This way the graphical view can be reduced to show the jobs and elements that you currently work on or that need revision.

To Switch Elements On or Off:

Select the **Eye** 👁 in the navigator to hide points and objects or even whole jobs from view. When the eye is shown in the navigator then the element or all elements in a job are visible in the graphical view.

Selecting Elements:

Action	Result
Single left mouse click onto an element	Selects the element.
Ctrl +single left mouse clicks onto elements	Selects a series of elements.
Shift (↑) +single left mouse clicks into background	Draws a polygon and selects all elements inside.
Shift (↑)+left mouse button +dragging the mouse	Draws window and selects all elements inside.
Shift (↑)+Ctrl +dragging window/drawing polygon	Selects all elements outside the window/polygon.

The properties of the selected element are loaded to and shown in the property grid.

In the graphical view, selected elements change their colour as specified in the layer manager.

Centring Elements:

To centre the view on a selected element select an element in the navigator or in theiInspector, right-click onto it and select **Centre To** from the context menu.

The view shifts to show the selected element in the centre.

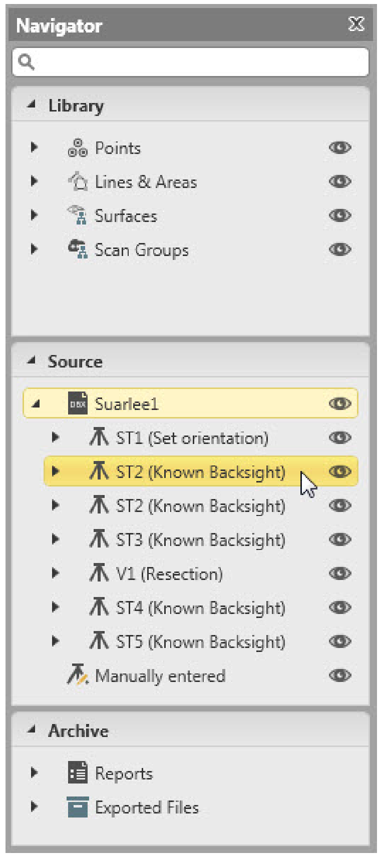
1.3.1.4 Navigator

Navigator

The navigator allows you to navigate through the project content. It shows the whole project content for searching for and locating single data items.

☞ Search has got a filter effect. You can filter the points and other objects by their ID or date and time. All other points and objects are hidden from view.

The project content is structured into three main parts:



In the **Library** section, all objects are grouped by the type of object they belong to and all points and objects are listed by their ID.

In the **Source** section, all imported jobs are listed by date and time. All setups are grouped by the jobs they belong to. Inside each job, all points are grouped by the setups they belong to and listed by their ID.

In the **Archive** section, all reports and exported files are listed in chronological order by their creation date and time.

☞ If you have closed the navigator you can make it reappear by selecting the **Navigator** option in the status bar.

Functions

Showing/hiding items:

To hide points and other objects or even whole jobs from view in the graphical view select the icon. This way the graphical view can be reduced to show the jobs and elements that you currently work on or that need revision.

Recovering points/observations:

Files/objects indicated by the icon, have been deleted externally and cannot be accessed.

Points and/or observations deleted in the field can be restored by selecting **Recover** from the context menu.

1.3.1.5 Data Inspector

Data Inspector

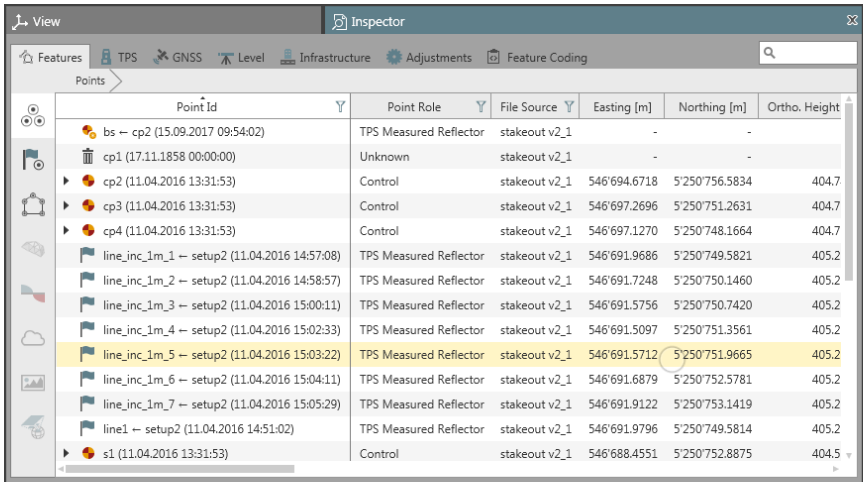
The data inspector shows module-specific project content grouped thematically for detailed investigation. It is designed to focus completely on one type of data object.

The availability of the module-specific tabs depends on which of the modules you have decided to purchase.

See also:

License Options

With the options on the left side, you can switch between subcategories.



☞ If you have closed the inspector you can make it reappear by selecting the **Inspector** option in the status bar.

Functions

Drilling down:

To get one step further into your data select the arrow next to a group of data items.

Navigation to parent:

To navigate back to a higher category of data items, select the breadcrumbs.

Example:

Filtering:

To filter points and other objects by their ID or date and time, use the search function.

To filter columns, select the option in the column header. To undo the filter effect, click the option again and clear your selection.

Editing:

To edit the properties of an item, select it and make your edits in the property grid.

Where ever the option is available a separate tool for editing can be invoked.

☞ Multi-Edit is possible for points and observations. For further information, refer to the following topics:
Observation Properties
Multi-Editing of Point Code Information

Recovering Points/Observations:

Points and/or observations deleted in the field can be restored by selecting **Recover** from the context menu.

Selecting Columns/Changing Column Order:

Right-click onto a column header or any item in the inspector view and select **Select columns...** from the context menu.

Select the columns to be visible and change the column order if desired.

You can also change the column order by dragging and dropping the column header.

Sorting by Column Heading:

Click onto the header of a column to sort its items in an ascending or descending order.

By default, the items in the inspector view are sorted by the first column.

☞ If tilt compensation was applied to a measurement using the Leica AP20 sensor, tilt compensated measurement values are shown in the data inspector.

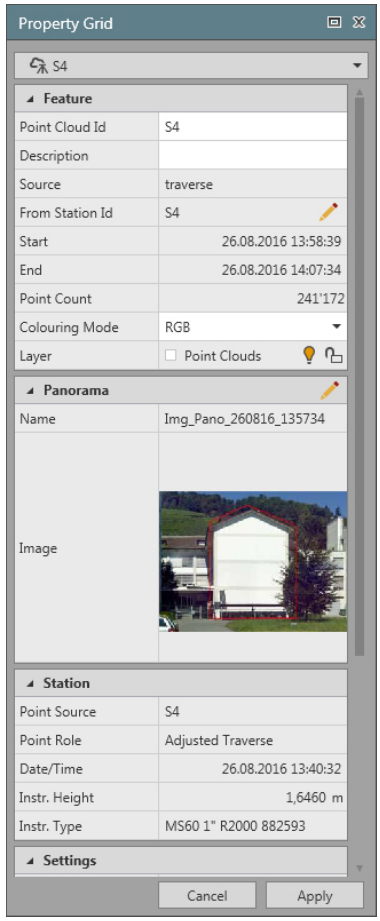
1.3.1.6 Property Grid

Property Grid

The property grid shows the properties of a selected data item. Data items can be selected in the graphical view, in the navigator and in the inspector. By default, the property grid is open and stays open unless you close it. It is filled dynamically depending on the selected data item.

☞ If multiple items are selected, select one from the drop-down list at the top to make its properties be displayed in the property grid.

Functions



All properties are arranged in subgroups that can be collapsed or expanded.

All properties with a white background can be edited.

The icon and the icon indicate that these properties can be edited or recalculated in a fly-out or separate tool. Fly-outs have to be confirmed by **OK** or **Cancel** before you can continue working.

All changes in the properties have to be confirmed by selecting the **Apply** at the bottom of the property grid.

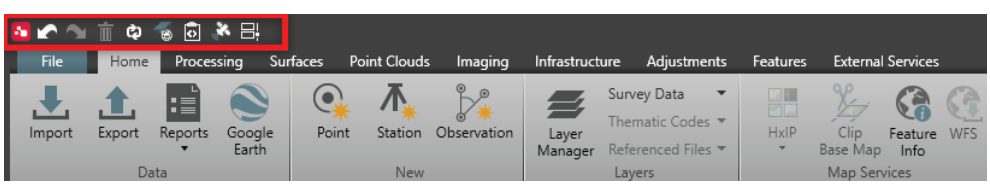
☞ If you have closed the property grid window you can make it reappear by selecting the **Properties** option in the status bar.

☞ If tilt compensation was applied to a measurement using the Leica AP20 sensor, tilt compensated measurement values are shown in the Property Grid.

1.3.1.7 Quick Access Bar

Quick Access Bar

The quick access bar is at the top left corner of the Infinity window and is active with an open project. It enables quick access to some important functions:



Icon	Description
↶	**Undo** an operation. It is possible to undo multiple times.
↷	**Redo** an operation. It is possible to redo multiple times.

Icon	Description
	**Delete** any selected item from the navigator, the data inspector or the graphical view.
	**Refresh** to refresh all windows and reset the graphical view.
	**Open** the coordinate system manager.
	**Open** the code table manager.
	**Open** the GNSS manager.
	**Open** the task manager to monitor the image-processing tasks.

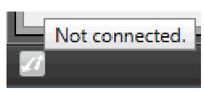
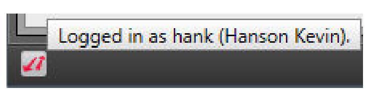
1.3.1.8 Status Bar

Status Bar

The status bar at the bottom of the Infinity workspace, indicates current settings.

Connectivity Status

You can see information about services that Infinity is connected with (Leica Exchange or Leica ConX).

Icon	Description
	Not logged in. It is not possible to send or receive data using Leica Exchange. You are not notified of incoming messages.
	Indicates being logged in. It is possible to import and export data from the open project.
	New messages are available for download.

Selection Status

The status bar indicates the number of selected items.

Window Status

The status bar indicates whether the navigator, the inspector and/or the property grid are visible or hidden.

Select the  to open/close them.

Tools

Open the **Layer Manager** or the **BIM Explorer** or the **Flythrough Creator** from here.

Unit Settings

The status bar indicates the units you have chosen for distances and angles.

To change the units and the available decimal places in the current project open the drop-down lists: Meter ▾ Gons ▾.

☞ You can also change the unit settings through **File** > **Info & Settings** > **Coordinates & Units**.

Master Coordinate System Settings

The status bar indicates the coordinate system which is currently used for displaying the project data. The selected coordinate system is called the master.

To select another coordinate system open the drop-down list:
Sample RT 1200 ▾

☞ You can also select a different master coordinate system through **File** > **Info & Settings** > **Coordinates & Units**.

Conditions:

- If you import raw data coming with a coordinate system then this coordinate system is automatically stored to the project and is made available in the drop-down list.
  To make it be used, select it from the list.

☞ If you want a coordinate system from the global coordinate system management to become available for selection, then go to **File** > **Tools** > **Coordinate Systems** and copy the required coordinate system to the current project.

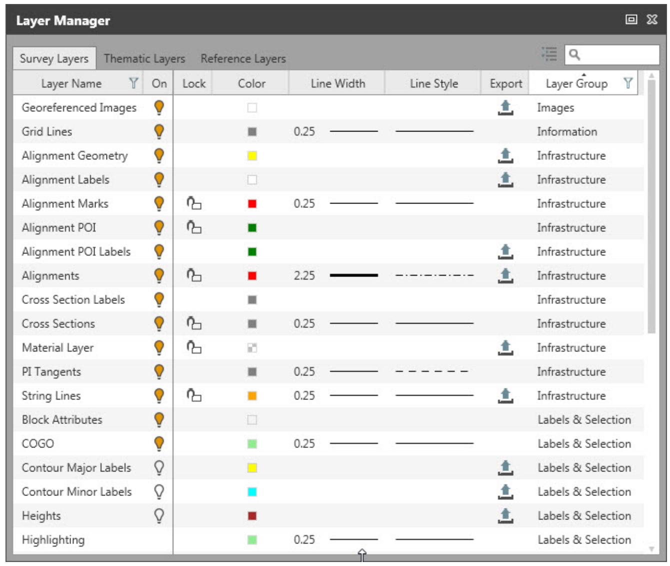
- If you select **None** then the data of each job is displayed using its own coordinate system.
- If you export project data to DBX, LandXML or HeXML, always the master coordinate system are exported with your data.
  If none is used then your data is exported without any coordinate system information.

☞ If you want to export all coordinate systems from the current project choose to export coordinate systems in the data export dialog.
All coordinate systems are saved to the same TRFSET.DAT file.

1.3.1.9 Layer Manager

Layer Manager

When you import data into an Infinity project, it is automatically organised into layers. Each layer is given a name in relation to its status and a pre-defined graphical representation. Each layer consists of a pre-defined colour, line style and shading style of the elements that belong to a layer. In the layer manager, you can change the status (on/off, locked/unlocked) and the graphical settings.



Layer Name	On	Lock	Color	Line Width	Line Style	Export	Layer Group
Georeferenced Images							Images
Grid Lines				0.25			Information
Alignment Geometry							Infrastructure
Alignment Labels							Infrastructure
Alignment Marks				0.25			Infrastructure
Alignment POI							Infrastructure
Alignment POI Labels							Infrastructure
Alignments				2.25			Infrastructure
Cross Section Labels							Infrastructure
Cross Sections				0.25			Infrastructure
Material Layer							Infrastructure
PI Tangents				0.25			Infrastructure
String Lines				0.25			Infrastructure
Block Attributes							Labels & Selection
COGO				0.25			Labels & Selection
Contour Major Labels							Labels & Selection
Contour Minor Labels							Labels & Selection
Heights							Labels & Selection
Highlighting				0.25			Labels & Selection

To open the layer manager:

Select the  icon in the home tab or in the Infinity status bar.

☞ The layer manager can stay open while working within Infinity, be shifted to another screen or docked in a preferred window position within the Infinity frame.

Survey data layers:

All surveyed data is organised in layers. On the survey layers, you can find all the Infinity objects that are imported as field data from SmartWorx (DBX) jobs or HeXML files. Survey layers can be exported to *.dxf, *.dwg, *.xml or *.HeXML files.

See also:

Objects, Point Roles and Symbols

Labels and selection layers:

On these layers, elements are grouped so that label survey data entities like point ID and point code ID, line/area ID or position and height labels.

☞ Labels can be switched on and off, but they cannot be locked or unlocked.

Survey data layers:

On these layers, the survey data is grouped by object type.

Thematic layers

All feature codes are organised in layers. On a thematic layer, the thematic information of feature objects like points, lines or areas are grouped. The thematic layers can be exported to *.dxf or *.dwg.

Thematic layers can either be edited directly in the code manager or in the layer manager. You can also create new thematic layers from inside the layer manager. In both cases, the code table is updated accordingly.

To create new layers in the layer manager:

Select the **New Layer** option in the top right corner and define the new layer through inline editing and selecting features from the drop-down list in each column. The layer name can also be changed through inline editing.

To delete selected layers:

Select one or more layers and select the **Delete** option to delete layers. The layers are deleted without any further warning message.

Reference layers

Under **Reference Layers** you can find all the layers as defined in referenced files like imported *.dxf or *.dwg files or *.ifc files. For those layers, you can also change the appearance, switch layers on/off or lock/unlock them, without having to modify the source files themselves.

See also:

Data Import

Settings

Switching layers on or off:

If the data in your project is organised in many different layers you can reduce the view to just the layers you are currently working on by switching others off:

Select the option to switch off a layer, select the option to switch on a layer.

Switched off layers are not visible in the graphical view.

Locking or unlocking layers:

If you want to make sure that elements in a specific layer cannot be selected then you can lock that specific layer.

Select the option to unlock a layer and the option to lock a layer.

The properties of data in locked layers cannot be edited.

☞ The graphical settings can always be changed for all layers, even if a layer is locked.

Defining the graphical settings:

To redefine the graphical appearance of a layer (for example, the colour of the point symbols or the line style of an area boundary) you can modify the line colour, its width and style as well as shading colour and style of area objects.

Excluding layers from export:

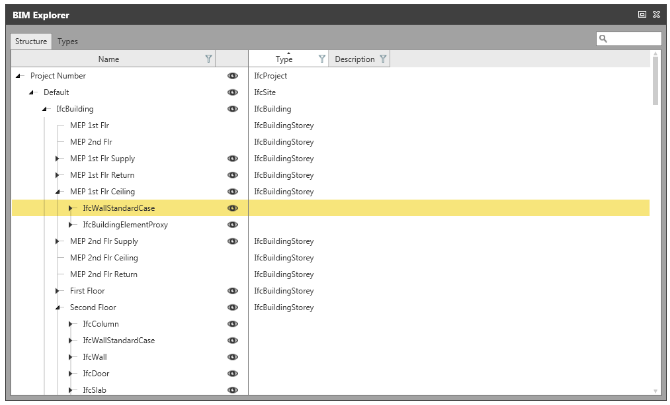
Most of the layers can separately be excluded from export.

In the **Export** column, select the option to exclude a layer from being exported.

To visualise the new status the option is crossed out, .

BIM Explorer

Use the BIM Explorer to inspect the structure of *.ifc files and list its entities by types.



To open the BIM Explorer:

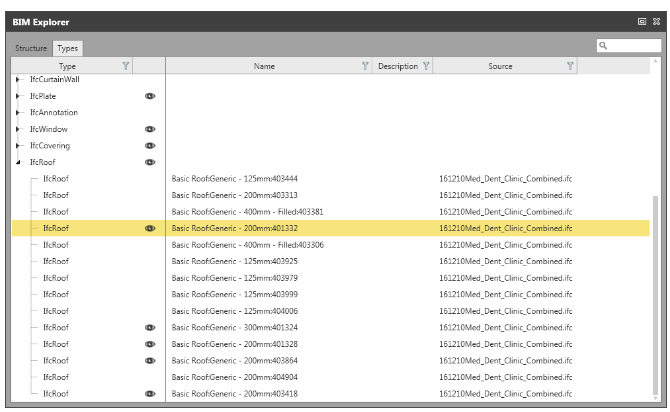
Select **BIM Explorer** from the status bar.

☞ The BIM Explorer can stay open next to working within Infinity, be shifted to another screen or docked in a preferred window position within the Infinity frame.

Available functionality

Switching between Structure view and Types view:

View the elements/entities either by file structure or sorted by type.



Selecting elements/entities:

When you select an element in the BIM Explorer it is simultaneously selected in the graphical view and its properties are displayed in the property grid.

You can select single elements as well as whole entities.

Viewing/Hiding elements/entities:

Select the **eye** to hide an element or a whole entity from view.

Deleting elements:

Selected elements can be deleted.

Searching for elements/entities:

Search for single elements or entities.

The Search functionality filters the elements. All other elements are hidden from view.

1.3.1.11 Flythrough Creator

Flythrough Creator

The flythrough creator allows the creation of video with defined flythrough methods like fly along view points, orbit, look around or fly along feature.

To open the flythrough creator, select the icon in the status bar.

☞ The flythrough creator can stay open while working with Infinity, be shifted to another screen or docked in a preferred window position within the Infinity frame.

Terminology	Definition
View Point:	Defined from the camera and look at point position.
Camera Position:	Is a place in the space from which you look at the data, for example when rotating around an object, you move the camera position. The camera position is changed by modifying the distance and angle between the camera and look at position.
Look at Position:	Defined by the current view (indicated by the rotation pivot point in the centre of the screen), by the current selection (for example a feature like road, line or area) or by a technical point.
Camera Angle:	Is the vertical angle of the camera to the data being viewed. Looking straight on is 90°/100 gon, 0° is up and 180°/200 gon is down.

To store the Flythrough:

1. Enter the name for the flythrough and then select **Save**, the flythrough is now active. Save all changes manually.
2. To create a new flythrough, select **New...** from the **Template** drop-down list, enter the name and select **Save**.
3. To use an existing flythrough, select it from the available list of templates.

Flythrough using Fly Along View Points:

Fly along view points create flythrough based on the view point positions defined by navigating in the 3D graphical view. To set a new view point navigate to the desired location and select **Create a new View Point from current view**.

1. Select the icon in the status bar to open the **Flythrough Creator** and select the **Fly Along View Points** method.
2. Navigate in the 3D view (by panning, zooming and rotating) to set the desired position and orientation.

3. Select create **New View** icon point to store the current position.
   A temporary indication for the camera position and the look at position appears in the graphical view.
4. The camera position and the look at position are modified by selecting a new technical point from the list or by manually changing the current view point coordinates.
5. Repeat steps 2 and 3 until all desired fly along view points are added.
6. Use play and stop to preview the flythrough.
7. Modify the available settings to change the flythrough duration, number of frames or to change the direction.
8. To save the flythrough, define the video format and resolution then select **Create**. In the export window, select the destination of the flythrough and select **Save**.

Flythrough using Orbit:

Orbit rotates the camera position 360° around the look at position at the specified distance and vertical angle. The look at position is a current view, selection or technical point.

1. Select the icon in the status bar to open the **Flythrough Creator** and select the **Orbit** method.
2. Select the **Look at Position** by defining the current view icon, selection icon or choose a technical point from the list.
3. Define the distance between the look at point and camera position and the angle. The vertical angle range is between 0 - 180°. For example, an angle of 30° makes the camera look up and an angle of 170° makes the camera look down.
4. Modify the available settings to change the flythrough duration, number of frames or to change the direction.
5. To save the flythrough, define the video format and resolution then select **Create**. In the export window, select the destination of the flythrough and select **Save**.

Flythrough using Look Around:

Look around fixes the camera position and rotates the view 360° at a specified vertical angle. The camera position is a current view, selection or technical point.

1. Select the icon in the status bar to open the **Flythrough Creator** and select the **Look Around** method.
2. Select the **Camera Position** by defining the current view icon, selection icon or choose a technical point from the list.
3. Define the angle for the look at position. The vertical angle range is between 0 - 180°. For example, an angle of 30° makes the camera look up and an angle of 170° makes the camera look down.
4. Modify the available settings to change the flythrough duration, number of frames or to change the direction.

5. To save the flythrough, define the video format and resolution then select **Create**. In the export window, select the destination of the flythrough and select **Save**.

Flythrough using Fly Along Feature:

The fly along feature creates flythrough by moving along a feature such as a line or alignment. The camera position comes from the offset to the look at position feature or the trajectory along a secondary feature.

1. Select the icon in the status bar to open the **Flythrough Creator** and select the **Fly Along Feature** method.
2. Choose between the **Offset** and **Trajectory** method to create the flythrough.
3. To create a flythrough with the offset method, choose the pen option to select a feature and define the offset for the camera position. The offset for the camera position works similar as the offset for the technical point.
   - The **Length** offset - The negative value (-) moves the position back and the positive value (+) moves the position forward.
   - The **Cross** offset - The negative value (-) moves the position to the left and the positive value (+) moves the position to the right.
   - The **Up** offset - The negative value (-) moves the position down and the positive value (+) moves the position up.
4. To create a flythrough with the trajectory method, select the pen option and select a feature for the look at trajectory and the camera trajectory.
5. Modify the available settings to change the flythrough duration, number of frames or to change the direction.
6. To save the flythrough, define the video format and resolution then select **Create**. In the export window, select the destination of the flythrough and select **Save**.

1.3.2 Local and Global Data Objects

Local and Global Data Objects

In Infinity, we distinguish between objects and settings from the **File** global and within the **Project-Local**.

A project holds all objects that allow it to be opened on a machine, and some of these objects are global. We have global objects so that they can be reused for single projects.

Global Objects:

- All objects which are stored in Infinity under the **Tools** within the **File** tab.
- Use **Copy to Project** to use them inside your active project.

Global Settings:

- All settings that are listed under **Preferences** within the **File** tab.
- Project templates that are used when creating a new project.

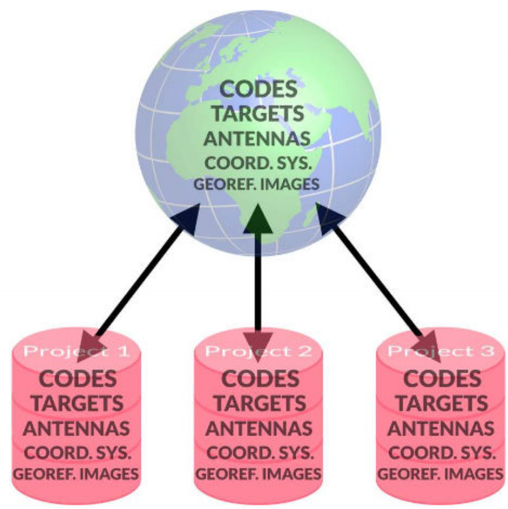
Local Objects:

- The data object is imported or found only in your project.
- Use **Export** to global to copy data from the project to the global objects.

Local Settings:

- Settings that are only valid within a single project.
- Are available only when an individual project is opened.
- Found under Info & Settings.

Working with Local and Global Data Objects:



No.	Name	Description
1.	Code Tables	Feature codes can be imported from Leica field software jobs into a global code table. The global code table can then be made available to any Infinity project.
2.	Targets	TPS target definitions are automatically imported into Infinity projects from Leica field software jobs (along with the survey data they relate to). New TPS target definitions can be manually entered globally and from there they can be copied to projects.
3.	Antennas	GNSS antenna definitions can be imported into the GNSS manager from a dedicated antenna file (not from a Captivate job). You can do it globally, or inside a project. Antennas are automatically mapped during import of GNSS data to the project. When the GNSS antenna is found in the global antenna list, then all antenna calibration definitions are copied into the project automatically.
4.	Coordinate Systems	Coordinate systems are often imported into Infinity projects (along with survey data), in jobs from Leica field software. They can then be saved to the global libraries and from there they can be copied into any other project.

No.	Name	Description
5.	Georeferenced Images	Background map images can be georeferenced backstage (using the points from a project) or inside an Infinity project, then saved to the global library.

☞ Any editing we do in Tools is on the global objects. Changes to these global objects do not affect any existing projects. But any new project that we create uses the updated global objects.

☞ When having to work with different coordinate systems, projects units and code tables use project templates Save as Template.

See also:

Tools

Save as Template

1.3.3 Objects, Point Roles and Symbols

Objects, Point Roles and Symbols

Infinity is an object-oriented software. The objects are the basic entities for performing operations. All objects are intelligent objects which means that they are interrelated with each other.

Symbol	Object	Description
	Points	A point is a named object that represents a location with either local or global latitude and longitude coordinates, and possibly a height or elevation.
	Lines	A line is used to describe feature objects by drawing a line between two or more points.
	Areas	An area is a closed line object.
	Setups	A setup is the source of any measurement observation.
	Observations	An observation is the measurement that defines the source of a point.
	GNSS Tracks	A GNSS track is an object resulting from processing moving or mixed data.
	Traverses	A traverse is a series of intervisible points at which angles are measured and also distances can be measured, to determine the station setups.
	Sets of Angles	A set of angles are a series of observations that are reduced to an angular value. The angular value is then coordinated from the source of the observations.
	Surfaces	A surface is a triangulation network representing points, cloud points, lines and areas.
	Point Cloud Groups	A point cloud group is a defined group of point clouds that represents a single object, used for cleaning or surface creation.

Symbol	Object	Description
	Point Clouds	A point cloud is a measurement object from the data source.
	Image Groups	An image group is a defined group of images that you can measure image points from.
	Images	An image is a picture captured that is represented by its position or object it is assigned to.
	Georeferenced Images	An image that has been ortho-rectified and can be placed in the project coordinate reference frame.
	Alignments	The alignment is a linear object made of multiple segments geometrically defined and joined together. It defines the route of a road construction, generally its centreline or axis.
	Roads	A road is made of several objects including a centreline and a material layer. The material layer describes the stringlines assigned to it.
	Material Layers	Are the layers constituting a road. A material layer groups a set of stringlines that belong to the same level, material or phase of construction.
	Cross Sections	A cross section is a slice or cut at a certain chainage of the road, displaying the position of different linear features connected together in a cross section view.
	Cross Section Assignments	Result from cross sections templates being assigned to a road. Stringlines result from interconnecting cross section assignments.
	Stringlines	Are special line objects that can either be imported from CAD or defined through cross section assignment.
	Scan Setup	A setup belonging to a BLK360, RTC360 or RTC360LT.
	Scan Group	A group of scan setups that have been joined together by links.
	Registration Link	A line shown between two scan setups to symbolise that they have been joined by visual alignment or target matching.
	Used Registration Target	A target that has been used to join setups by target matching.
	Unused Registration Target	A target that has not been used to join setups by target matching.

Interval Icons for GNSS Tracks:

Symbol	Object	Description
	Static Interval	This icon indicates that GNSS raw data is acquired with static mode.
	Moving Interval	This icon indicates that GNSS raw data is acquired with kinematic mode.
	Mixed Interval	This icon indicates that GNSS raw data is acquired with alternating static and kinematic modes.

Point Roles and Point Symbols:

Symbol	Point Role	Description
	Control Point (not fixed in adjustment)	Is a control point that is not considered for adjustment and not fixed.
	1D Control Point (fixed in height)	Is a 1D control point that is considered for adjustment, fixed only in height.
	2D Control Point (fixed in position)	Is a 2D control point that is considered for adjustment, fixed only in position.
	3D Control Point (fixed in position and height)	Is a 3D control point that is considered for adjustment and fixed in position and height.
	Adjusted Measured Point	Is a measured point adjusted by the least squares method or in a traverse computation. The adjusted least squares point role, include the adjustment method 3D, 2D or 1D.
	Station Setup	Is a point on which a station setup exists after import of field data.
	Averaged Point	This point is derived by averaging two or more measured points.
	TPS Reduced Measurement	This point is generated from the reduced observation computed from sets of angles, reduced foresights or a traverse.
	TPS Measured with Reflector	Is a point that has been measured using a reflector.
	TPS Measured Reflectorless	Is a reflectorless measured point.
	TPS Measured with Reflector (compensated)	Is a tilt compensated point that has been measured using a reflector and the Leica AP20 sensor.
	TPS Measured	Is a point that has been measured without instrument EDM information. Typically, such points are imported from XML.
	TPS Measured Setup Point (with reflector)	Is a control point used in a Setup application and a measurement has been taken with reflector EDM.

Symbol	Point Role	Description
	TPS Measured Setup Point (reflectorless)	Is a control point used in a Setup application and a measurement has been taken with reflectorless EDM.
	Fixed Real-Time Kinematic (RTK)/ Fixed PP	Is a GNSS RTK measured or post-processed phase fixed point (most accurate).
	xRTK/Widelane PP	Is GNSS xRTK measured or a widelane post-processed phase fixed point.
	PPP Converged	Is a GNSS point measured with precise point positioning, final position converged.
	Float RTK/Float PP	Is a GNSS RTK measured or post-processed point with float solution (less accurate).
	PPP Converging	Is a GNSS point being measured with precise point positioning, final position not yet converged.
	Code RTK/Code PP	Is a GNSS RTK measured or post-processed point with code solution (least accurate).
	Navigated RTK/ Navigated PP	Is a GNSS RTK measured or post-processed point with lower accuracy. It is measured without using a reference station.
	☞	When the GNSS point roles have got a green background, this indicates a tilted measurement.
	GNSS Measured	Is a measured GNSS point with unknown solution type. Typically, such points are imported from XML or SKI ASCII.
	GNSS Track Post-Processed	Is a post-processed GNSS track (using a reference station).
	GNSS Track Navigated	Is a post-processed GNSS track computed by applying the SPP technique (without using a reference station).
	GNSS Track Averaged	Is a GNSS track derived by averaging two or more post-processed tracks.
	Level Measured	Is a point measured by level staff.
	Offset Point	Is a point for which an offset observation has been measured or entered.
	Auto Point	Is a point measured with the auto points application.

Symbol	Point Role	Description
●	Computed Point	Is a point calculated by an application and not measured with an instrument.
○	User-entered Point	Is a manually entered point.
+	Point (unknown)	Is a point for which no point role can be defined on import. Typically, such points are imported from XML.
🗑	Point (deleted)	Is a point which has been deleted on the instrument and could be recovered.

1.3.4 Shortcut Keys in Infinity

Shortcut Keys in Infinity

Graphical view:

Action	Result
Ctrl+/Ctrl- or move mouse wheel	Zooms in/out.
Alt+Ctrl+/Ctrl- or Alt+move mouse wheel	Locks zooming to the pivot point.
Right, left, up or down arrow or press the mouse wheel and move it	Shifts the view (pan the view).
Click into the view with the left mouse button +dragging the mouse	Rotates the view.
Ctrl+R	Turn on/off look around.

Selecting elements:

Action	Result
Single left mouse click onto an element	Selects the element.
Ctrl +single left mouse clicks onto elements	Selects a series of elements.
Shift (↑) +single left mouse clicks into background	Draws a polygon and selects all elements inside.
Shift (↑)+left mouse button +dragging the mouse	Draws window and selects all elements inside.
Shift (↑)+Ctrl +dragging window/drawing polygon	Selects all elements outside the window/polygon.

Lighting of shaded objects:

Action	Result
I+moving the mouse wheel	Adjusts the overall intensity.
K+moving the mouse wheel	Adjusts the key light intensity.

Action	Result
F+moving the mouse wheel	Adjusts the fill light intensity.
B+moving the mouse wheel	Adjusts the back light intensity.
A+moving the mouse wheel	Adjusts the ambient light intensity.

1.3.5 Window Behaviour

Window Behaviour

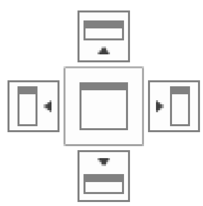
The overall Infinity user interface is designed such that the graphical 2D/3D view is always available in the background. The other three main windows (the navigator, the inspector and the property grid) can individually be made to be:

Floating:

Select the window with a left mouse-click onto its title bar and drag it out of its place. It is undocked from its place and becomes a floating window.

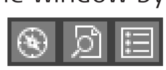
To dock it again, select the floating window with a left mouse-click onto its title bar and drag it to one of the docking places that appear when you start moving the window. Drop it with the mouse pointer onto the desired docking area and the window is docked in the frame of the opaque yellow area.

Illustration of docking places:



☞ To enlarge the available space for the graphical view, floating windows can comfortably be moved to a second monitor.

Hidden:

Hide the window by either selecting  in its title bar or by selecting its option  in the status bar. The option frame disappears to indicate that the window is hidden: 

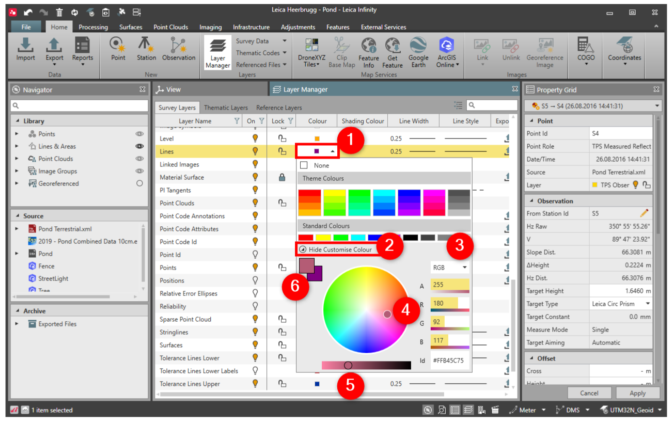
To make the hidden window visible again select the option again in the status bar.

1.3.6 Customising Colours

Customising Colours

Code and layer colours can be customised.

To customise a colour:



1. Select **Home**, then **Layer Manager** and select the object to be customised.
2. From the drop-down menu, select **Show Customise Colour**.
3. You can choose between **RGB** (Red-Green-Blue) colour definitions or **HSB** (Hue-Saturation-Brightness) from the drop-down list.
4. Drag the inner circle to the area of the desired colour spectrum.

   ☞ You can also enter the ID of a specific colour or its RGB/HSB parts.
5. Move the circle in the bar at the bottom to change the brightness.
6. Set the selected colour by selecting the square in the top left corner of the customisation area.

1.3.7 Working with Coordinate Systems

1.3.7.1 Overview

Working with Coordinate Systems

Each Infinity project can support many coordinate systems. These coordinate systems can be imported from data sources such as SmartWorx jobs or copied into the project from the global objects list. Since multiple coordinate systems can exist in the project, the project has settings to define how the coordinate systems are used. The project can have a single coordinate system which is used to project all project data to the same grid positions or each data source can use its own coordinate system to project the data that belongs to that imported data source.

☞ Coordinate systems are global objects.

Master Coordinate System

The master coordinate system is what the project is defined to use.

If a coordinate system is chosen when creating a new project, then this is called the master coordinate system. This master is used to project all data to the common grid reference frame. All WGS84 data is projected to local grid values and all local grid values can be converted to WGS84. You see this when working with the project, from the status bar.

☞ When you are exporting data to various formats, it is important to have a master coordinate system set for your project data so all the data is on the same reference frame.

Data Source Coordinate Systems

For many data sources, it is possible that they include their own coordinate system. Importing a SmartWorx job could have a coordinate system as used on the field instrument. You see this in the data source properties if a coordinate system is included.

It is possible to view each coordinate system as was used on the field sensors by choosing **None** in the status bar. This can be useful to identify potential problems between different data sources.

Using the Coordinate Systems

<table>
<tr><td>Master Coordinate System</td><td>

What you see with project data when a master coordinate system is in use:
WGS84 entered points are convertible to local grid unless they fall outside the projection limits of the selected coordinate system.</td></tr>
<tr><td>Data source Coordinate Systems</td><td>

What you see with project data when no master coordinate system is in use and if no coordinate system is assigned to the data source:
WGS84 entered points are not convertible to local grid. They are not displayed in the graphic view and are indicated with this icon: .
Point averaging results only include the local grid values if available:
When only WGS84 coordinates exist for that point ID, then the average is computed in WGS84.
When one or more local coordinates exist for that point ID, then the average only considers the local grid points.
Exporting data might not include the coordinates as expected.</td></tr>
</table>

See also:

Coordinate System Properties

1.3.7.2 Coordinate Systems inside a Project

Coordinate Systems inside a Project

You can import one or more coordinate systems to your current project either together with SmartWorx DBX jobs or independently of raw data as transformation sets stored in TRFSET.dat files or from LandXML.

All imported coordinate systems are automatically:

- Stored to the project.
- Available for selection in the job properties, the ASCII file properties, the project settings (Info & Settings) and the status bar.
- Listed in the **Coordinate Systems** tab of the Inspector.

Job Coordinate Systems and ASCII/SKI ASCII file Coordinate Systems

In the job property grid you can choose a different coordinate system to be attached to the source data of a job.

In the ASCII/SKI ASCII file property grid, you can choose a coordinate system to be attached to the source data of an imported ASCII/SKI ASCII file.

All coordinate systems stored with the current project are available for selection.

1. In the navigator, go to the **Source** section and select the job/ASCII or SKI ASCII file to which you want to attach a different coordinate system.
2. In the property grid, select a different coordinate system from the drop-down list.

☞ If you select **None** no coordinate system is used with the selected job/ASCII/SKI ASCII file.

☞ Directly after import ASCII/SKI ASCII files always have the coordinate system None. Attach a coordinate system to be able to use individual coordinate systems with each set of imported raw data, be that jobs or ASCII files, in your project.

If you want to make the coordinate system of the selected job globally available under **File** > **Tools** > **Coordinate Systems**:

Select the Export to global option.

Inspecting Coordinate Systems

In the **Coordinate Systems** tab of the inspector all coordinate systems that are stored with the current project are listed.

You can:

- Remove coordinate systems from the project.
  Select one or more than one coordinate system, right-click into the selection and select **Delete** from the context menu.
  Alternatively, select the option in the top left corner of the main window.

☞ You can only delete, for example coordinate systems which are not in use.

☞ Removed coordinate systems are still available in the global coordinate system management under **File** > **Tools** > **Coordinate Systems**.

- Export coordinate systems to the global coordinate system management under **File > Tools > Coordinate Systems**.
  Select one or more than one Coordinate System, right-click into the selection and select **Export to Global** from the context menu.
- Inspect the properties for each coordinate system including geoid and CSCS models.
- See whether a coordinate system is **In Use** and/or in use as a **Master**.
  Coordinate systems are in use when they are attached to one of the jobs that have been imported to the project.
  A Coordinate system is the master when it is selected either in the info & settings or from the status bar to be used with all data in the project.

☞ The master coordinate system may not be in use with one of the jobs.

☞ If no master is selected (for example, **None** is chosen in the status bar) then the data of each job/ASCII file is displayed using its own coordinate system.

Coordinate systems can be modified in the property grid.

Exporting Coordinate Systems:

If you export a project to DBX, LandXML or HeXML always the master coordinate system is exported with your data. If **None** is used then your data is exported without any coordinate system information.

To Export all Coordinate Systems from a Project:

Select to export coordinate systems in the data export dialog to export all coordinate systems that are available with the current project and save them to the same TRFSET.DAT file.

See also:

Data Export

1.3.7.3 Coordinate System Properties

Coordinate System Properties

To view and edit the coordinate system properties:

Go to the **Coordinate Systems** tab of the inspector and select a coordinate system from the list.

In the property grid:

- Select the Export to Global option if you want to make the selected coordinate system be globally available under **File** > **Tools** > **Coordinate Systems**.

☞ The option is inactive if the selected coordinate system is already available under **File** > **Tools** > **Coordinate Systems**.

- Select the icon to get detailed information on the transformation, ellipsoid and projection used with the coordinate system in a fly-out.
- Select the icon to get detailed information on the geoid model or the CSCS model used with the coordinate system. Import a geoid or CSCS model if required or select a different one from the drop down-list.

☞ Only Geoid/CSCS models that are valid for the selected coordinate system are available for selection. Their names can be modified if required.

- Leave the **Property Grid** with **Apply** to take over your changes.

To import Geoid/CSCS Model:

In the Geoid/CSCS Model fly-out:

1.	Select the option to browse for and select a geoid field file (*.gem) or a CSCS model (*.csc). Their names can be modified.
2.	Leave the fly-out with **OK** to add the geoid/CSCS model to the drop-down list.
3.	The newly imported model is selected automatically. Select **None** if you want no geoid or CSCS model to be used with the coordinate system.
4.	Leave the fly-out with **OK** to apply your modifications. The inspector view is updated accordingly.

☞ Only Geoid/CSCS models that are valid for the selected coordinate system can be imported.

2 Basic Modules

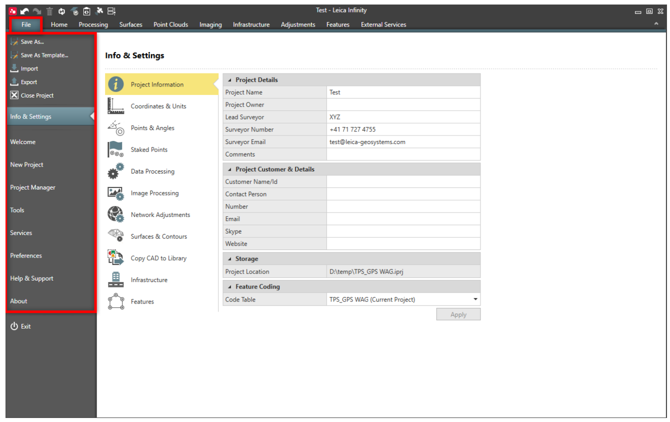
2.1 File

2.1.1 Overview

File

The file tab includes all the backstage functionality.

Here you can do the following:



Name	Description
Save as Template	Save the current project as a template.
Data Import	Import many different data formats.
Data Export	Export all or selected data to many different formats.
Info & Settings	Adapt the settings for the Infinity project.
Welcome	See all your recent projects and read Leica news.
New Project	Create a new Infinity project.
Project Manager	See and manage all your projects in one place.
Tools	Manage or create Infinity global objects.
Services	Access services from Leica, Hexagon and third parties.
Preferences	Under preferences, you can configure global application settings.
Help & Support	Access Infinity training materials as well as the localisation tool.
About	Get Information about your licence and installed Infinity version.

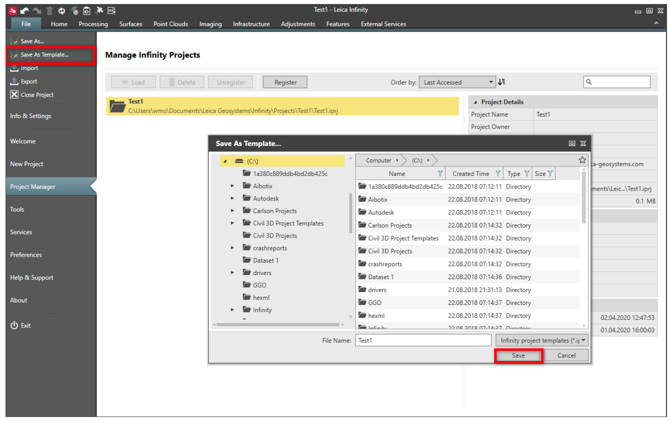
2.1.2 Save as Template

Save as Template

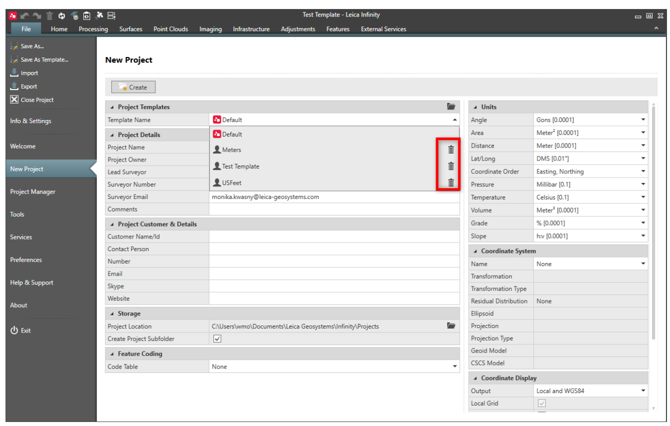
Save the current project as a template. When you save a project as a template, all project settings as well any data in the project are saved in the new template file (.iprjt). The new name is added to the template list, to select when creating a new project.

☞ Good practice is to create a project template when having to work with different coordinate systems, project units and code tables you commonly use.

☞ The default project template is from the user settings which were set using the **Save as Project defaults** option in an older version of Infinity.



Project templates can be deleted from the selection list, when creating a new project.



2.1.3 Info & Settings

Info & Settings

In this section of the file tab, you can find information on the settings for the Infinity project that you are currently working on. You can adapt the settings to your personal requirements.

Project Information

Project Details and Project Customer & Details

View and/or edit the project details and project customer details as set when you created a project under **File** > **New Project**.

See also:

New Project

Feature Coding

Choose a different Code Table to be used with the current project. Select **Apply** to make your changes become effective in the current project.

Coordinates & Units

Units

Decide on your preferred distance, area, volume and angle units and the decimal places to be available.

Decide on your preferred temperature and pressure units and the decimal places to be available.

Choose the coordinate order you prefer. Available are easting/northing or northing/easting.

☞ Your choice is applied to the listing of coordinates in, for example, the wizards or the property grid. It does not apply to the content views, though, where a change in the column order can be achieved by drag and drop of columns.

Coordinate Systems

Choose one of the coordinate systems that are stored with the project to be used as the master. Coordinate systems are either stored to a project automatically by being imported together with raw data or can be copied to a project manually from inside the global coordinate system management under **File > Tools > Coordinate Systems**.

The coordinate system you choose is used for converting between WGS84 and local coordinates.

See also:

Coordinate Systems inside a Project

Coordinate Systems

Choose **None** if you want each job to be converted using its own coordinate system.

☞ If you want a different coordinate system to be attached to a job then change the job properties accordingly in the property grid.

View information on the transformation and residual distribution, on the ellipsoid and projection as well as on the geoid and CSCS model used with the selected coordinate system. A different geoid or CSCS model can be selected in the coordinate system properties.

See also:

Coordinate System Properties

Coordinate Display

Choose local if you want Infinity to hide WGS84 coordinate representations in the inspector, the property grid and the graphical view. Only WGS84 observations which can be converted to local grid shows up in the graphical view.

Choose **Local and WGS84** if you want Infinity to output local grid and WGS84 coordinates. To display local grid coordinates is by default always selected. You can additionally select geodetic and/or cartesian coordinate representations to be output in either local and/or WGS84 if available or convertible. The inspector shows the additional columns. In the property grid switch between representations by selecting a different output.

☞ To be able to convert coordinates from one format to another, a coordinate system must be selected to be used with the project data.

Coordinate Direction

Choose to switch the northing and/or easting.

Points & Angles

Point Averaging Settings

Define the maximum distance allowed between an average and the measurements from which it has been computed.

Define whether weighted averages shall be computed for the current project. The setting is applied to points and GNSS tracks.

☞ For GNSS tracks the average is computed with respect to the solution type.
Meaning that if a phase fixed solution and a code solution shall be averaged automatically, the phase fixed solution is taken into account while the code solution is ignored.

Angle Reduction Parameters

Define the angle residuals and slope distance residuals to be allowed when computing sets of angles.

☞ The tolerance settings can also be changed for single computation runs in the sets of angles wizard or for single targets once a sets of angles computation is completed. To change the tolerances for single targets select the ✏ option in the **Observation** section of the **Sets of Angles Target** properties. The tolerances shown correspond to the last used, but can be changed for single targets if necessary.

Define the maximum face differences to be allowed when computing sets of angles.

Image Point Tolerances

Defines the maximum 3D accuracy allowed for the computed image points. If the 3D accuracy is outside of this value, no point is computed.

You can always reset your changes to the factory defaults.

Staked Points

Define the stakeout tolerances.

☞ Defaults are available.

When you import stakeout data to Infinity then the delta values that exceed the defined tolerances are marked in red bold in the stakeout report.

See also:

Reports

Checked Points

Define the checked tolerances.

Δ Azimuth - Is the difference between the calculated azimuth and the current orientation.

Δ Hz Distance - Is the difference between the calculated and the current horizontal distance.

Height Difference - Is the difference between the calculated and the current height.

☞ Defaults are available.

When you import checked points data to Infinity then the delta values that exceed the defined tolerances are marked in red bold in the checked points report, as well as the TPS inspector.

See also:

Reports

Data Processing: TPS

Default Traverse Adjustment Parameters

Define the default parameters for adjusting traverses like method and balancing.

Decide whether you want to apply scale to observations.

Tolerance Checks in Traverse Wizard

Define which tolerance checks shall be used in the traverse wizard and enter different values for the limits.

Station Difference before and after Adjustment

Define the maximum allowed difference between stations before and after the adjustment computation.

The changes you make here are used as defaults in the traverse wizard. You can always reset your changes to the factory defaults.

☞ For entering different values for tolerance checks, the available decimal places should be set to at least five positions after the decimal point for angles and three for distances.

For detailed information on single parameters, see:

Traverse Processing Parameters

Data Processing: GNSS

The sections data and processing strategy, correspond to dialogs in the GNSS processing module.

See also:

Settings: Data

Settings: Strategy

Settings: Advanced

☞ The defaults used in the advanced settings are suffice in most cases. Only modify them if you are an advanced user and have special data to be processed.

The section time, sets the time format used in the **Inspector** > **Intervals View**.

See also:

GNSS Intervals

The section Computation, steers automated behaviour that follows a computation.

You can also decide whether navigated tracks are processed on import. Only when this setting is active are GNSS tracks automatically recognised as data objects on import.

Data Processing: Level

Level Line

Define the default level line adjustment method - By distance or equally.

Set the default tolerance values to flag the line information before applying adjustment.

Observations

Set the default tolerance values to flag the observation information to help identify data issues.

Point Heights

Set the default tolerance values to flag the height differences to be applied to the points.

Staff Corrections

Set the default staff correction values that could be applied in a level line adjustment.

Registration

Cloud-to-Cloud

Define default parameters for the cloud-to-cloud routine including max. iterations and search radius. The max. points density and normal threshold can also be set in this section.

Targets

Define default parameters for the match targets routine including max. target error and prioritise targets.

Unified Point Cloud

Define the default parameter for the max. number points.

Registration Tolerances

Define the flagging tolerances for a number of registration metrics in the application and scan registration report.

☞ See Point Cloud Registration Settings for descriptions on individual settings.

Network Adjustments: General

General

Set the general parameters for adjustment computations and iterations.

Test Criteria

Define the **Test Criteria** to be used in adjustment computations.

Advanced Terrestrial Parameters

Decide whether reduced observations and/or refraction coefficients and scale factor corrections shall be used.

Define the values for the refraction coefficients and scale factor corrections.

Coordinate System

Decide on the coordinate system and the height mode in which your observations shall be adjusted.

Define rotations about the axis and a scale if necessary.

The changes you make here are used as defaults in the adjustments module of your project. You can always reset your changes to the factory defaults.

☞ For entering different values, the available decimal places should be set to at least five positions after the decimal point for angles and three for distances.

For detailed information on single parameters see:

General Adjustment Settings

Test Criteria

Advanced Terrestrial Parameters

Coordinate System Settings

Network Adjustments: Accuracy

TPS

Define standard deviations and centering and height errors for TPS observations.

GNSS

Define standard deviations and centering and height errors for GNSS observations.

Level

Define standard deviations and centering and height errors relative to the level data.

Surfaces & Contours

Meshing

Define the **Max. Triangle Size** and the **Min. Triangle Size** for the surfaces created with the refined, regular and 2.5D method.

☞ Occasionally, triangles can exceed the maximum and minimum defined size. This may occur when the surface must be continuous, for example while creating a watertight mesh. Like for example on surface boundaries, the triangle size may exceed the defined maximum and minimum size to produce a more reasonable and realistic shape of the surface.

Fill Holes

Fill Interpolation - Define the method. The level of interpolation uses surrounding surface geometry to determine the number of extra triangles to create between hole edges.

Contours

Minor Interval - Define the distance between contour lines that are drawn.

Major Frequency - Define when a major contour is drawn. Usually this setting is set at five or ten.

Min. Length - Define the minimum contour length. Contours shorter than this value are not created.

Smoothing - Define the degree of smoothing applied to the contour lines.

Cut Fill

Cut Definition - Define if the cut positive values are referring to the material **Above Reference** or **Below Reference**.

Cut Fill/Comparison Map

Define the **Type** of cut fill and comparison map. Should only differences in the height be calculated (**Height Difference**) or if a value above and below the reference should be used for calculation and visualisation (**Tolerance**).

☞ If Tolerance is selected for the Type, then define the **Upper Tolerance**, **Lower Tolerance** values and the **Min. Segment Distance**.

Then define the **Colour Mode**, **Colour Ramp** and the **Colour Transition**.

Copy CAD to Library

The copy from CAD settings can be set separately for points, lines, areas, surfaces, alignments and roads.

Define the point role that points copied from CAD shall have. Choose between **User-entered** or **Control**.

Define a naming scheme (a prefix or a suffix) that entities copied from CAD shall have.

Separate prefixes and/or suffixes can be defined for points, lines, areas, surfaces, alignments and roads.

Decide if/how the imported entities shall be coded.

A code table needs to be defined.

☞ Surfaces and roads cannot be coded.

Decide on which layer the imported entities shall be:

- By default the layer structure is taken over from the source file.
- If the source file does not have layer information, then the entities are imported to a default layer.

☞ If a code table is defined and codes are assigned to layers, then layer selection is blocked.

Define how heights shall be managed for points copied from CAD.

Define a default height that 2D objects shall get.

You can also specify a height to be excluded. Entities with that height are copied either as 2D objects (with no height) or they get the specified default.

☞ Defaults can be restored separately for points, lines, areas and surfaces.

Infrastructure

General

Choose the way and with which precision chainages are displayed in the graphical view, inspector and the property grid.

Choose major interval and minor frequency to define how the alignment shall be segmented visually.

Define your own abbreviations for horizontal and vertical points of interest.

Road

Set the tolerance thresholds for the maximum deflection angle that two consecutive elements are allowed to have in a horizontal or vertical alignment.

Exceeding values are flagged in the application and reports.

The stringline chord tolerance defines the spacing used for digitizing stringlines.

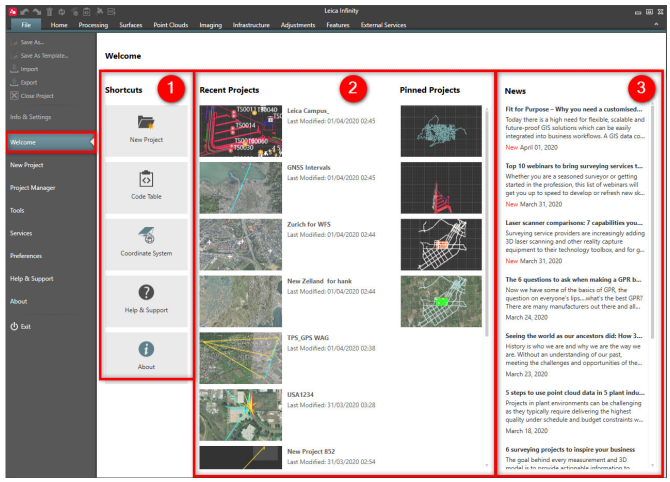
Curves need to be digitised as a sequence of straight segments and the tolerance defined here is used for it.

☞ Angular and metric units can be changed under Coordinates & Units.

2.1.4 Welcome

Welcome

In welcome you see:



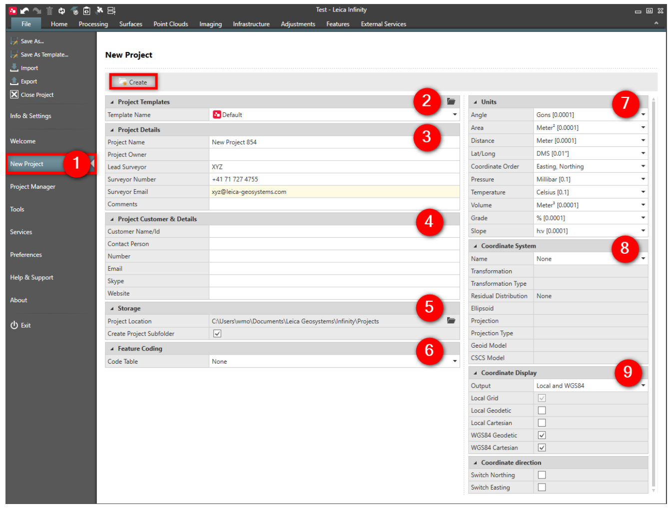
No.	Name	Description
1.	**Shortcuts**	Shortcuts allow you to create, for example, new projects, code tables, coordinate systems or to access help & support or about pages.
2.	**Projects**	Open recent or pinned projects. ☞ There is a preview for a project if it has been opened once.
3.	**News**	Information about various Leica product releases including hardware and software, as well as blog posts, new videos on YouTube and new tutorials.

2.1.5 New Project

New Project

New projects can only be created from inside the file tab.

To create a new project:



1.	Go to the **File** tab and select **New Project** from the menu on the left.
2.	Define the **Project Template**.
☞	The default project template is from the user settings which were set using the save as project defaults option in an older version of Infinity. For more information, refer to Save as Template.
3.	Define the **Project Details**: • Give the project a unique **Project Name**. • Optionally, fill in the details on the **Surveyor**. • Optionally, add **Comments** and **Tags**.
4.	Optionally, fill in **Customer Details**.
5.	Define the **Storage** information, for example where to store the project and if you want to create project subfolders. Project subfolders provide a clear folder structure to organise the following data: • Data sent and received through ConX. • Data sent and received through Leica Exchange. • Exported data from Infinity. • Imported data to Infinity. • Reports generated within Infinity.
6.	Optionally, choose a **Code Table** to be used with the project. To be able to do so, at least one code table has to be globally available from the **File** > **Tools** > **Code Table Management**. It is copied to the project. If there are no global objects yet, leave as **None**.
7.	Define Units for various measurement types.
8.	Optionally, choose a **Coordinate System** to be used with the project. To be able to do so at least one has to be globally available from inside the **File** > **Tools** > **Coordinate Systems**. It is copied to the project. If there are no global objects yet, leave as **None**.
9.	Choose from the **Coordinate Display** which coordinate format you want to see in the project.

10. Set the **Coordinate Direction** which switches the coordinate system axes for the northing, easting or both. For example, used in South Africa.

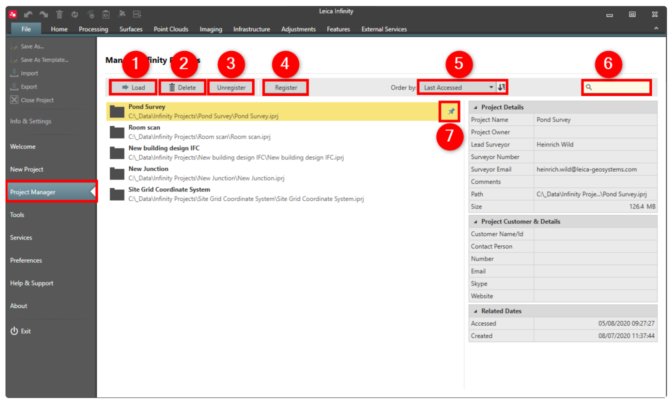
11. Select the **Create** option to create and open the new project in the home module.

2.1.6 Project Manager

Project Manager

The whole data and object management in Infinity is done in working units called projects. Each project uses its own database. The same data and objects may belong to and can be handled in different projects. Start the project manager from the file tab.

In the project manager you can do the following:



No.	Name	Description
1.	**Load**	Open an existing project in Infinity so you can carry on working on it. A single click on a project selects it and a double-click loads it.
2.	**Delete**	Delete the selected project and all the data within it. There is no undo option!
3.	**Unregister**	Remove an existing project from the project manager list. Does not delete the project.
4.	**Register**	Browse to an existing Infinity project file (.iprj) to add it to your list of projects.
5.	**Order by**	The list of Infinity projects can be ordered by: • Project name (alphabetical order). • Date the project was last used. • Date the project was created.
6.	**Search box**	Search for any part of a project name.
7.	**Pin project**	Pin the project to have it always available on top of the project list. Pinned projects also appear on the Welcome page.

See also:

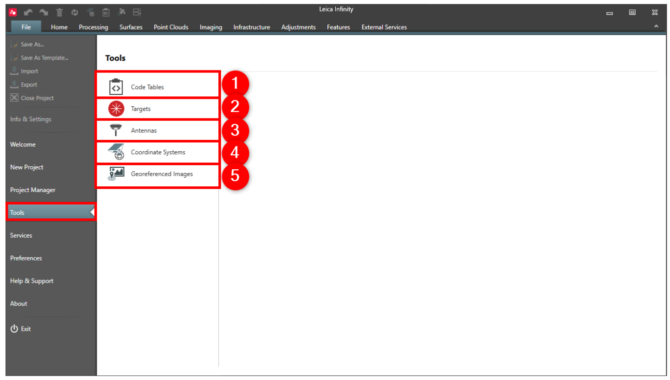
Save as Template

2.1.7 Tools

2.1.7.1 Overview

Tools

Tools are available to manage the following global objects:



No.	Name	Description
1.	Code Tables	Has a collection of codes and attributes. New Infinity code tables can be created here.
2.	Targets	Manage prism constant values.
3.	Antennas	Manage antennas.
4.	Coordinate Systems	Manage coordinate systems.
5.	Georeferenced Images	Manage georeferenced images.

☞ Any edit we do in tools is on the global objects. Changes to these global objects do not affect any existing projects. But any new project that we create uses the updated global objects.

See also:

Local and Global Data Objects

2.1.7.2 Code Tables

Code Tables

The code table is to manage the list of code groups, codes, attributes and styling information including layers, blocks and custom line styles.

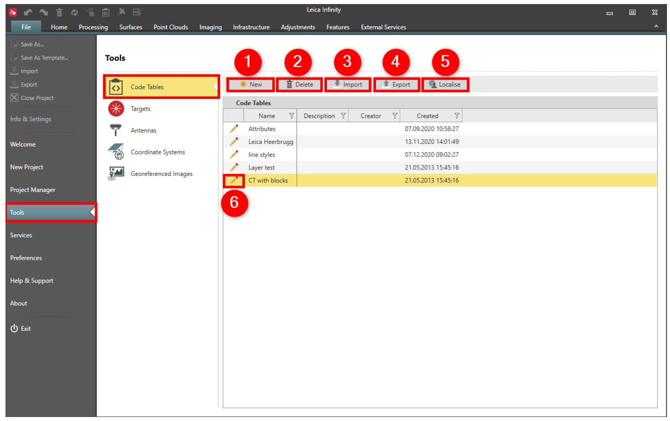
A codelist can be generated in the code table and transferred to the field software. When measurements are made, the details of the features like codes and attributes, are recorded and stored with the measured point. Everything can be considered a feature, a building, a fence, a road, a bench, or even a tree.

The code table applies automatically the correct style to the imported field data.

Use feature coding to help increase productivity both in the field and the office by:

- Having features described.
- Visualising the data in the software.
- Simplified CAD export.

In code tables you can do the following:



No.	Name	Description
1.	**New**	Create a new code table.
2.	**Delete**	Delete an existing code table, including existing codes and attributes. ☞ There is no undo function.
3.	**Import**	Import an Infinity code table from an *.lic file. ☞ To import a codelist from Leica field software, a new Infinity code table must first be created and edited it in the Code Manager.
4.	**Export**	Export an Infinity code table to an *. lic file. ☞ A code table, can be exported as a codelist to Leica field software, from inside the Code Manager.
5.	**Localise**	Open the Localisation Tool and download code tables from the available global or national list.
6.	**Edit**	Open the Code Manager to add/edit codes and its attributes as well as layers, line styles and blocks.

See also:

Code Manager

The video **"Leica Infinity – Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity – Feature Coding Part 2 - How to use blocks and layers"** https://www.youtube.com/watch?v=REYP8JzmaSs

The video **"Leica Infinity – Feature Coding Part 3 - How to use custom line styles"** https://www.youtube.com/watch?v=ze9ganVf6AU

The video **"Leica Infinity – Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.3 Code Manager

2.1.7.3.1 Overview

Code Manager

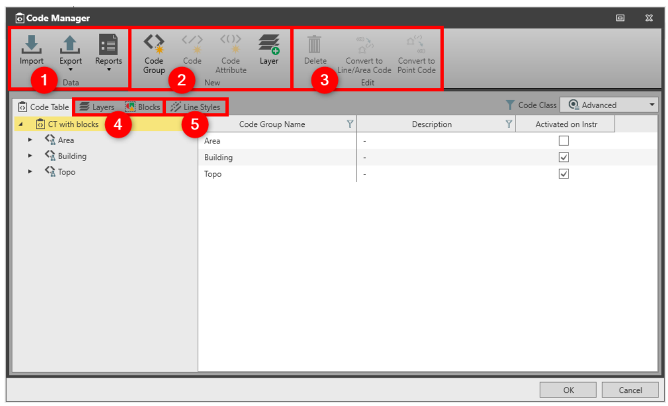
The code manager allows you to manage all feature coding information included in code tables:

- Coding information (code groups, codes and attributes).
- Coding style information (blocks and layers), which are extended code properties used for visualisation.

To open code manager :

1. Go to **File**, then select **Tools** and then Code Tables.
2. Select one or create new and select **Edit**.

In code manager you can do the following:



No.	Group Name	Name	Description
1.	**Data**	Import	Import codes from available format files.
		Export	Export a codelist to various format files or a code table.
		Reports	Open either code report or code table report.
2.	**New**	Code Group	Create a new code group.
		Code	Create a new code.
		Code Attrib-ute	Create a new code attribute.
		Layer	Create a new layer.
3.	**Edit**	Delete	Delete selected objects.
		Convert to Line/ Area Code	Convert a point code with linework to a line/area.
		Convert to Point Code	Convert a line/area code to point code with linework.
4.		Layers and Blocks	Import layers and blocks from DXF/DWG CAD files.
5.		Line Styles	Import custom line styles from LIN files.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity - Feature Coding Part 2 - How to use blocks and layers"** https://www.youtube.com/watch?v=REYP8JzmaSs

The video **"Leica Infinity - Feature Coding Part 3 - How to use custom line styles"** https://www.youtube.com/watch?v=ze9ganVf6AU

The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

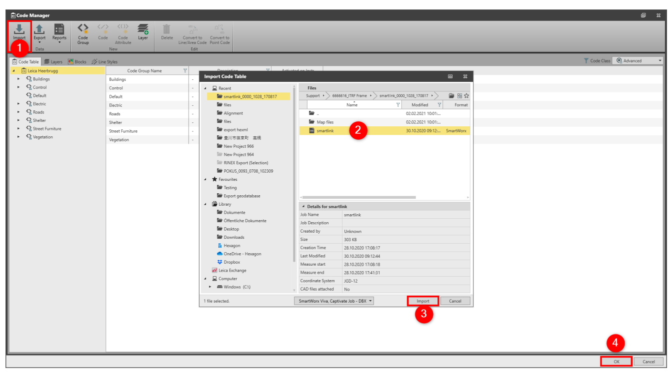
2.1.7.3.2 Import

Import

The code table content can be built by importing:

- Code Tables (*.lic) or (*.xcf).
- SmartWorx Viva, Captivate Job -DBX.
- HeXML/LandXML- XML.
- ESRI Geodatabase.
- Trimble -FXL.
- FlexLine - CLS.

To import codes from existing sources:



1.	Open the Code Table and select **Import**.
2.	Navigate to the file you want to import and select it.
3.	Select **Import**.
4.	Select **OK** to save the changes and to close the Code Manager.

☞ After importing a codelist, you may add coding style properties to the imported codes.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

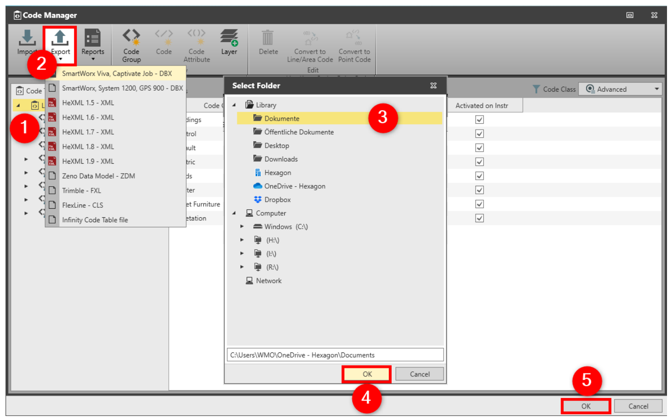
2.1.7.3.3 Export

Export

The code table content can be exported to:

- SmartWorx Viva, Captivate Job - DBX.
- SmartWorx, System1200, GPS 900 - DBX.
- HeXML- XML.
- Zeno Data Model -ZDM.
- Trimble -FXL.
- FlexLine - CLS.
- Infinity Code Table - (*.lict).

To export a code table or codelist:



1.	Open the Code Table that you want to export.
2.	Select **Export** and select the format.
3.	In the Select Folder window, select the location to save the file.
4.	Select **OK**.
5.	Select **OK** to save the changes and to close the Code Manager.

See also:

Code Manager

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

2.1.7.3.4 Code Group

Code Group

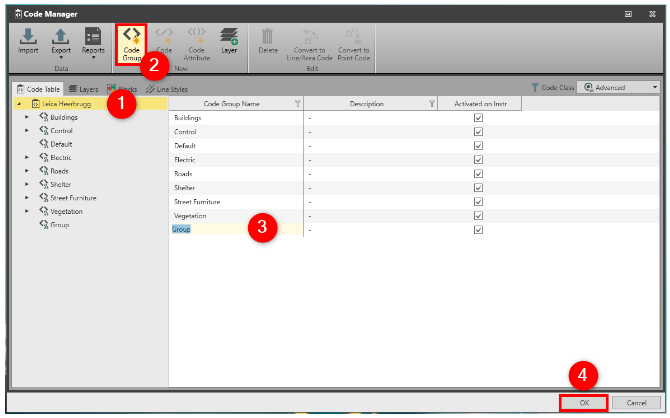
Code groups are the primary building blocks of a code table and describe groups of objects, which have a common theme. For example, utilities, vegetation, buildings or roads. Grouping helps to manage codes by activating or deactivating groups on board the instrument.

A code table may have as many code groups as you define. Each code group then has subcomponents known as codes and attributes.

Code group names may be up to 16 characters long and may consist of alphanumeric characters.

You can add new code groups, modify existing code groups and Delete code groups.

To add a new code group:



1. Select the Code Table in the Code Manager.
2. Select **Code Group**.
3. In the content area, a new group is created with default settings that you can modify.
   - ☞ A Code Group Name must be unique and may be up to 16 characters long.
   - ☞ A description is optional.
4. Select **OK** to save the changes and to close the Code Manager

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.3.5 Code

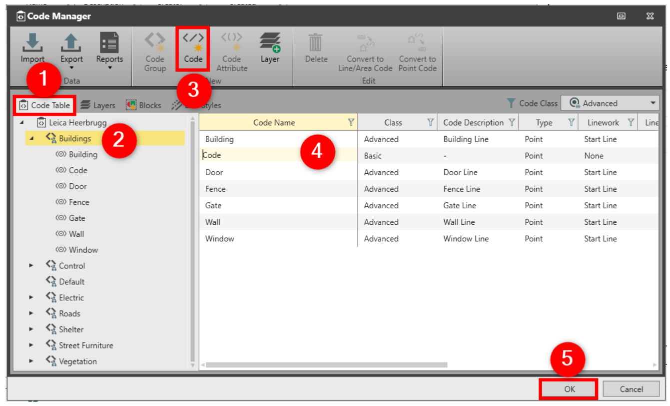
Code

Codes are parts of code groups and are used to describe objects. A set of codes with a common theme is grouped into a code group. For example, the codes tree, hedge and grass, may be attached to a code group entitled vegetation.

There are two methods for feature coding:

Name	Description
Thematical codes	Can be assigned to points, lines and areas.
Free codes	Purely time-related information recorded in-between measurements in the field. A timestamp is recorded with each free code allowing a chronological order of export points and free codes to be used for third-party mapping software.

To add a new code to the code group:



1.	In the Code Manager, select the **Code Table** tab.	
2.	Select the code group to which you want to add a new code in the navigator view.	
3.	Select **Code** from the ribbon bar	
4.	In the content area, a new code line is created with default settings that you can modify.	
	☞	In the Attributes, you can see how many attributes the code has.
5.	Select **OK** to save the changes and to close the Code Manager.	

A code consists of:

Group Name	Name	Description
Coding Information	Code Name	May be up to 16 characters long and may consist of numbers or alphanumeric characters (for example tree).
	Code Description	May be up to 16 characters long and may consist of alphanumerical characters (for example outstanding tree). ☞ The description of a code is optional. ☞ For Leica Captivate V2.30 and higher, the allowed length is 48 characters when ASCII characters are used. If other than ASCII characters are used, then it depends on the character set whether longer names are truncated on export. Codelists and jobs with names longer than 16 characters cannot be imported to any Infinity version lower than 2.4.
	Type	Can either be point, line, area or free.

Group Name	Name	Description
	Quick Code	Must be unique within a code table and up to three alphanumeric characters are allowed. ☞ To define a quick code is optional.
	Linework	Is only available for codes type point and it can be either none, start line or start area.
Coding Style Information	Layer	Enables grouping of codes by theme. For example, the codes kerb, canter line and pavement can be grouped in the same layer called road.
	Style Option	Can be either by layer or by code. This option allows you to set if the code uses its own style in the project or uses the style from the layer. For example, the codes kerb and pavement (type line) can use different colours. Both can be assigned to the same layer called road. When you select the by code option, they are presented in the project with their own colour. When you select the by layer option they are presented with the same colour defined for layer road. ☞ This option is not available for free codes. ☞ By code is not available for the point code if linework is set to none.
	Block	Geometry objects used to represent features. For example, a CAD block represents a tree or a signpost.
	Line Colour	Is available for code types point/linework (start line/start area), line, area.
	Line Width	Is available for code types point/linework (start line/start area), line, area.
	Line Style	Is available for code types point/linework (start line/start area), line, area.
	Shading Colour	Is available for code types point/linework (start area), area layer.

There is no limitation for the number of codes. You can add new codes, modify existing codes and delete codes within a code group.

Each code may have attributes attached to it. Attributes prompt you to enter further information about the code.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.3.6

Code Attribute

Code Attribute

Attributes are the tertiary building block of a code table. Attributes prompt you to enter information describing a code. You can add new attributes, modify or delete existing attributes of a code.

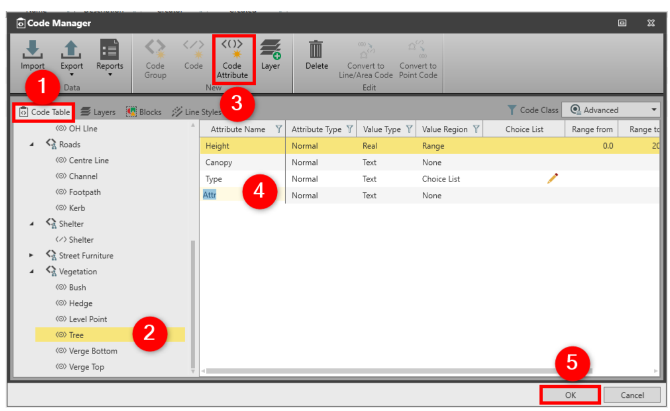
For example, attributes for the code tree could be species, diameter and remark.

Each code may have one or more attributes attached to it. Attributes are the tertiary building block of a code table. Attributes prompt you to enter information describing a code.

For example, the code tree could have the attributes diameter, species, height, and remark attached to it. You may then define an attribute value for an attribute. It may be chosen from a predefined choice list or a predefined range. For example, possible values for the attribute diameter could be from a range from 1 to 25 metres and the attribute species from a choice list that has the values pine, fir and oak.

☞ You do not have to define an attribute value within code management. If no value is defined for an attribute you may enter a value or description in the field.

To add new attribute:



1. In the Code Manager, select the **Code Table** tab.
2. Select the code to which you want to add a new attribute in the navigator.
3. Select **Code Attribute** from the ribbon bar.
4. In the content area, a new attribute is created with default settings that you can modify.
   ☞ To copy attributes between codes use Ctrl C and Ctrl V.
5. Select **OK** to save the changes and to close the Code Manager.

An attribute consists of:

Name	Type	Description
Attribute Name		The Attribute Name may consist of alphanumeric characters and may be maximum 13 characters long. ☞ For Leica Captivate V2.30 and higher, the allowed length is 48 characters when ASCII characters are used. If other than ASCII characters are used, then it depends on the character set whether longer names are truncated on export. Codelists and jobs with names longer than 16 characters cannot be imported to any Infinity version lower than 2.4.
Attribute Type	Normal	The attribute value can be edited in the field.
	Mandatory	The attribute value can be edited in the field.
	Fixed	No value is shown on the instrument and the default value is automatically attached to the attribute.
Value Type	Text	Possible values may only be entered in a choice list. For example, for the attribute species choose text as the value type.
	Real	You have the possibility of entering a choice list or a range of possible values.
	Integer	You have the possibility of entering a choice list or a range of possible values. For example, for the attribute diameter choose integer as the value type.
Value Region	None	For example, for the remark attribute choose none.
	Range	Enter or modify the interval (range from/to). For example, for the attribute diameter a range from 1 to 5 metres could be defined.
	Choice List	Enter or modify a list of possible attribute values. For example oak, pine and fir could be possible choice list entries for the attribute species. • To add new values to the choice list select **Add**. • To remove values from the choice list select **Delete**. • To modify the sequence of the values in the choice list select **Move Up/Move Down**.
Default Value		Optional.

Name	Type	Description
		Enter default value, for value region choice list or range.
		If the attribute type is fixed, enter a default value otherwise no attribute value is set.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

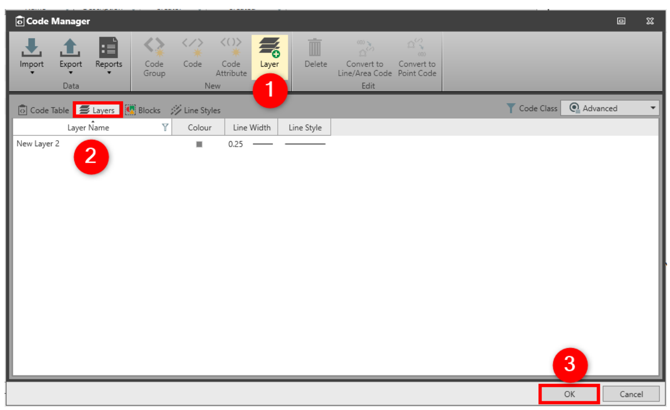
2.1.7.3.7 **Layer**

Layer

A layer is used to group thematic information within a project. The layer includes style information that can apply to all the coded features that are using this layer.

Use the style option to decide if the feature is presented with its individual by code style or with the common by layer style.

To add a new layer:



1.	In the Code Manager, select **Layer** from the ribbon bar.
2.	The view is switched to the **Layers** tab and in the content area a new layer is created with default settings that can be modified.
3.	Select **OK** to save the changes and to close the Code Manager.

See also:

Code Tables

The video **"Leica Infinity – Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

The video **"Leica Infinity – Feature Coding Part 2 - How to use blocks and layers"** https://www.youtube.com/watch?v=REYP8JzmaSs

---

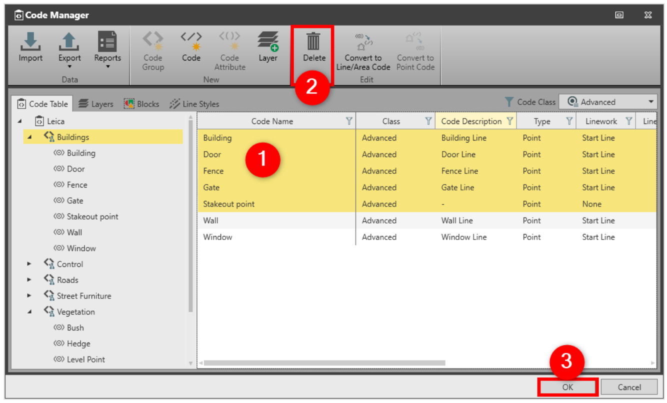
2.1.7.3.8 **Delete**

Delete

In the Code Manager It is possible to delete the following objects:

- Code Groups
- Codes
- Attributes
- Layers
- Blocks
- Line Styles

To delete an object from code manager:



1. In the Code Manager, select the objects from either the **Code Table**, **Layers**, **Blocks** or the **Line Styles** tabs
2. Select **Delete**.
3. Select **OK** to save the changes and to close the Code Manager.

☞ If you delete a code group, all codes and attributes in that code group are deleted.

☞ If you delete a code, all attributes of that code are deleted.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

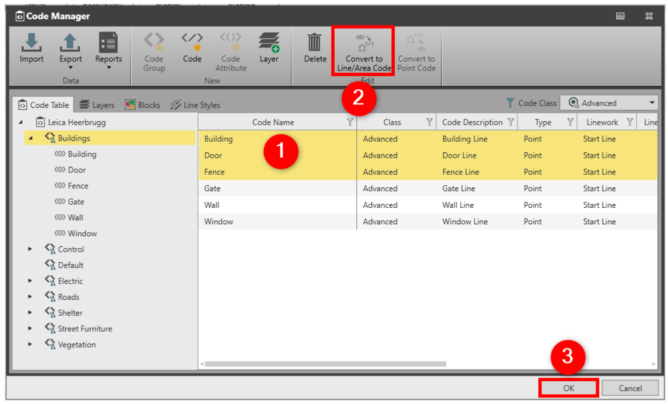
The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.3.9 Convert to Line/Area Code

Convert to Line/Area Code

The convert to line/area code option is useful when data has to be exported to GIS, CAD or VIVA where the point linework is not supported.

To convert to line/area code:



1. In the Code Manager, select one or more point codes with linework.
   ☞ You can also select the whole code group.
2. Select **Convert to Line/Area Code**.
   ☞ Points with linework, start line convert to lines and the start area convert to area.
3. Select **OK** to save the changes and to close the Code Manager.

See also:

Code Tables

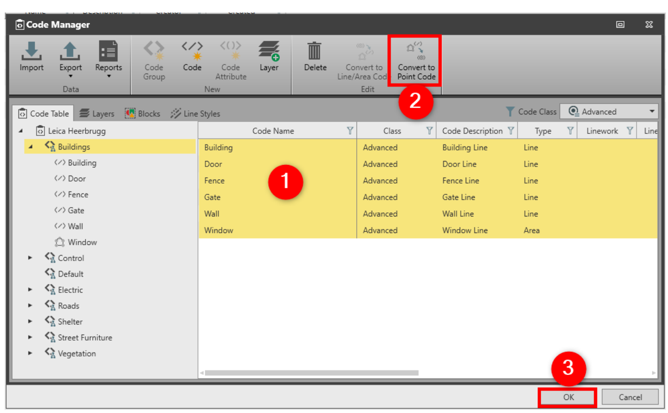
The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

2.1.7.3.10 Convert to Point Code

Convert to Point Code

The convert to point code option is useful when you want to use your existing line/area codes in Captivate where only the point linework is supported.

To convert to point code:



1. In the Code Manager, select one or more codes of type line or area.
   ☞ You can also select the whole code group.

2. Select **Convert to Point Code**.

☞ All selected codes are converted to the type point with linework start line or start area.

3. Select **OK** to save the changes and to close the Code Manager.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 1 - How to create a Code Table & Codelist"** https://www.youtube.com/watch?v=9Mf8E0trTXA

2.1.7.3.11 **Layers and Blocks**

Layers and Blocks

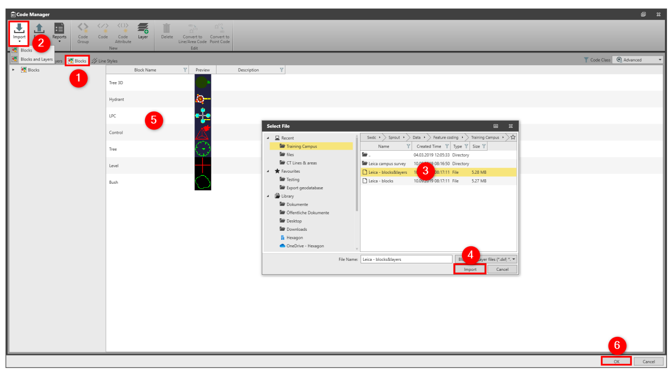
Blocks are the collection of geometry objects that act as a single object and can be used in a drawing repetitively to represent features. For example, a block can be used to represents a tree or a signpost.

Blocks can have attributes and can only be used for the codes, type point.

Layers are used to define a common style.

Blocks, block attributes and layers can be imported into the code table from DXF/DWG CAD files.

To import blocks and layers:



1. In the Code Manager, select the **Layers** or the **Blocks** tab
2. Select **Import** and then **Blocks** or **Blocks & Layers** from the ribbon bar.
3. Navigate to the file you want to import and select it.
4. Select **Import**.
5. In the content area, the newly imported blocks are seen.

☞ The block preview is only shown if it is imported from the CAD file. Blocks preview can be created in AutoCAD with the 'BLOCKICON' command.

6. Select **OK** to store changes and to close the Code Manager.

☞ To see the imported block attributes, select a block in the blocks tab.

☞ To see imported layers, go to the layers tab.

See also:

Code Tables

The video **"Leica Infinity - Feature Coding Part 2 - How to use blocks and layers"** https://www.youtube.com/watch?v=REYP8JzmaSs

The video **"Leica Infinity - Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.3.12 **Line Styles**

Line Styles

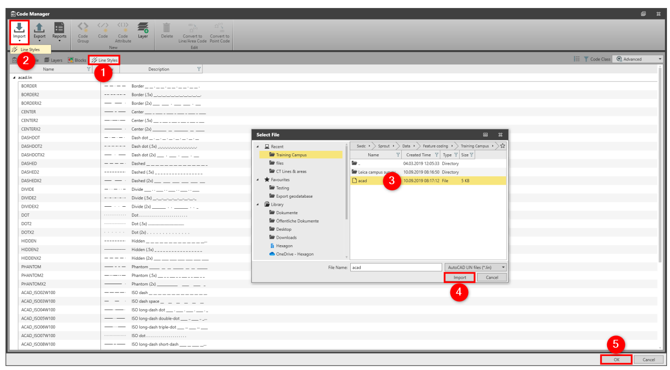
Line style is a visual property that can be assigned to geometric objects. Line styles can be for example a pattern of dashes, dots or text.

Line styles are defined in a line style definition LIN file. You can import the line styles from the LIN file and use them with your codes.

The line style functionality is not supported in SmartWorx Viva or Captivate. Therefore, line styles are not exported to the codelist. Collect your data as normal and assign the code table to the project for the correct style representation.

☞ Line style names are case-sensitive. During an import, line styles with the same name are overwritten.

To import line styles from a LIN file to a code table:



1.	In the Code Manager, select the **Line Styles** tab.
2.	Select **Import** and select **Line styles** from the drop-down menu.
3.	Select the LIN file to import.
4.	Select **Import**.
5.	Select **OK** to save the changes and to close the Code Manager.

☞ Line styles combo box is only available for the code type line or point with linework set to start line or start area.

See also:

Code Manager

The video **"Leica Infinity – Feature Coding Part 3 - How to use custom line styles"** https://www.youtube.com/watch?v=ze9ganVf6AU

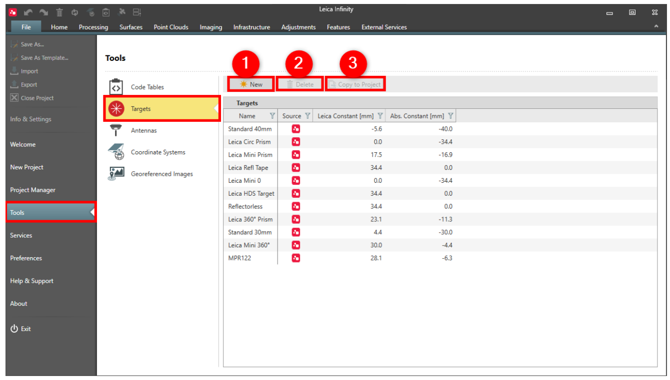
The video **"Leica Infinity – Feature Coding Part 4 - How to process features and export to CAD"** https://www.youtube.com/watch?v=ASLoVHHAYGQ

2.1.7.4 Targets

Targets

Prism constants for TPS targets are defined here. Infinity provides a default list of targets.

In targets you can do the following:



No.	Name	Description
1.	**New**	Create a new user-defined target and enter the prism constant values. ☞ If a user-defined target was created in the field software, it is imported with the TPS data and therefore is not necessary to be created again.
2.	**Delete**	Delete selected target. Only user-defined targets can be deleted. ☞ There is no undo function.
3.	**Copy to Project**	Copy user-defined targets to the currently open project. The project must already be open before this option is available.

See also:

Local and Global Data Objects

2.1.7.5 Antennas

2.1.7.5.1 Overview

Antennas

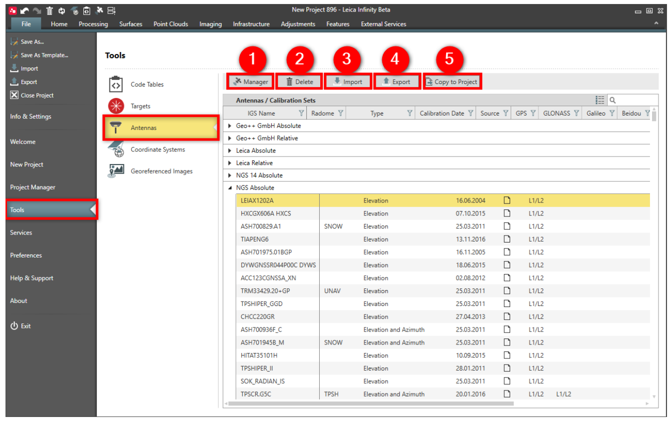
Antennas can be managed in the antennas list. Infinity provides a default list of antennas and calibration sets.

By default, the following calibration sets are available:

- **GEO++ GmbH Absolute** and **GEO++ GmbH Relative**: Include elevation and azimuth calibration values for Leica antennas.
- **Leica Absolute** and **Leica Relative**: Include elevation and azimuth calibration values for Leica antennas.
- **NGS 14 Absolute** and **NGS Absolute**: Include elevation and azimuth calibration values for Leica and third-party antennas.

☞ NGS calibration set, provides the antenna calibration values published by NGS at the following link: https://www.ngs.noaa.gov/ANTCAL/.

In antennas you can do the following:



No.	Name	Description
1.	GNSS Manager	Manage antenna calibration sets, RINEX data from permanent reference stations, precise ephemeris, ionospheric models and predict the satellite availability.
2.	Delete	Delete user defined or imported antennas.
3.	Import	Import antennas.
4.	Export	Export antennas.
5.	Copy to Project	Copy antennas from global list into open project.

See also:

Local and Global Data Objects

Antennas/Calibration Sets

The tutorial **"How to Manage an Antenna Calibration Set"** https://leica-geosystems.com/-/media/a633447d1f3f4afc8c7abf4a1685be6a.ashx

☞ The tutorial can be downloaded in the Localisation Tool.

2.1.7.5.2

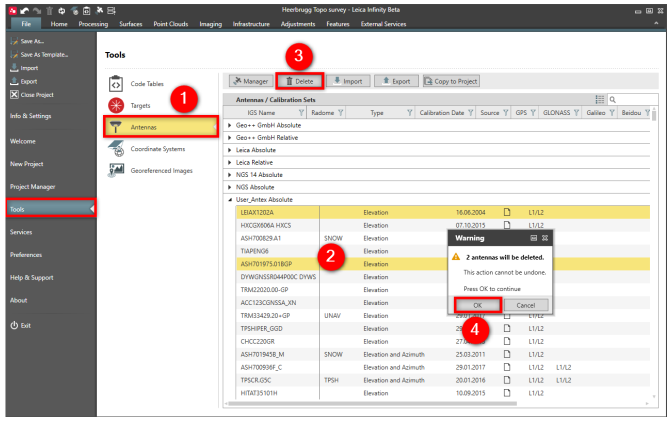
Delete

Delete

User-defined or imported antennas/calibration sets can be deleted.

☞ The following actions can also be done from within the GNSS Manager.

To delete antennas:



1. Select **File**, then **Tools** and then **Antennas** from the menu.
2. Select the antenna file you want to delete.
3. Select **Delete**.
4. Select **OK**.

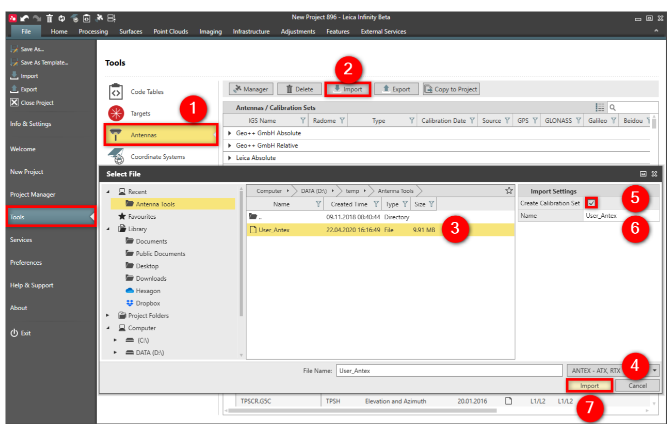
2.1.7.5.3 Import

Import

Except for the predefined antennas, it is possible to import new antennas and add them to the list.

☞ The following actions can also be done from within the GNSS Manager.

To import a new antenna:



1. Select **File**, then **Tools** and then **Antennas** from the menu.
2. Select **Import**.
3. Select the antenna file you want to import.
4. Select the file type from the drop-down menu.
5. Select the checkbox **Create Calibration Set** in the import settings, to add all the antennas in the file, under a new calibration set.
6. Specify the name of the new calibration set.

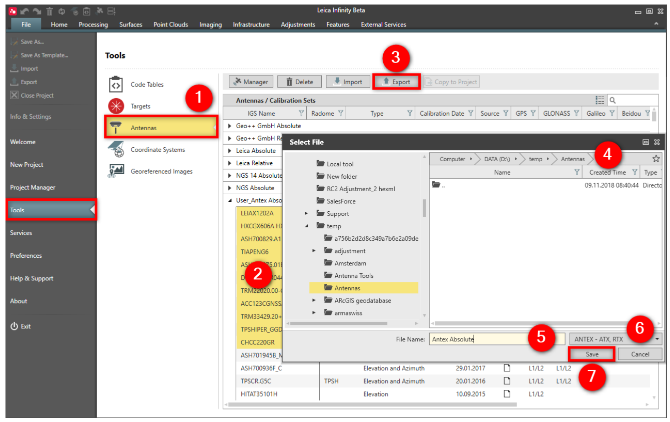
7. Select **Import**.

2.1.7.5.4 Export

Export

☞ The following actions can also be done from within the GNSS Manager.

To export an antenna:



1. Select **File**, then **Tools** and then **Antennas** from the menu.
2. Select the antennas/calibration sets you want to export.
3. Select **Export**.
4. Select the folder location to export the antennas to.
5. Specify the name.
6. Select the file type from the drop-down menu.
7. Select **Save**.

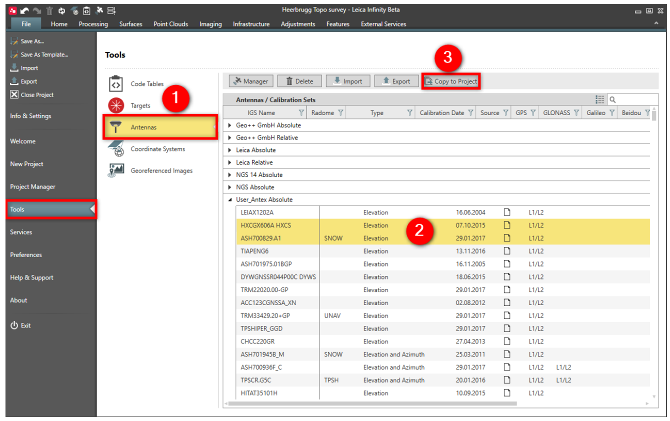
2.1.7.5.5 Copy to Project

Copy to Project

Antennas are automatically mapped during import of GNSS data to the project. When the GNSS antenna is found in the global tools antenna list, then all antenna calibration definitions are copied into the project automatically.

Antennas can be copied from the global tools into single projects to make them available in calibration sets for processing.

To copy antennas to a project:



1.	Select **File** , then **Tools** and then **Antennas** from the menu.
2.	Select the antennas you want to copy.
3.	Select **Copy to Project**.

See also:

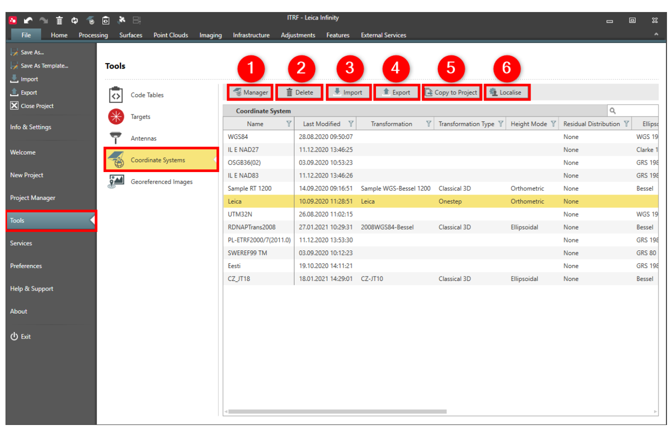
Local and Global Data Objects

2.1.7.6 Coordinate Systems

Coordinate Systems

The globally available coordinate systems are listed by name together with their source and information on the transformation and residual distribution, the ellipsoid and projection and the geoid and CSCS model used with each coordinate system.

Here you can do the following:



No.	Name	Description
1.	**Manager**	Opens the Coordinate System Manager, where geoid field files, CACS field files and the whole coordinate systems can be imported, created and managed.

No.	Name	Description
2.	**Delete**	Permanently delete the selected coordinate system. ☞ There is no undo function.
3.	**Import**	Import existing coordinate systems from: • SmartWorx DBX (*.xcf). • *.dat files. • iCON/SBG Geo *.lok. • Infinity Coordinate System file*.CSYS. • HeXML/LandXML. • LGO CSys *.dbd. • Trimble *.DC/*.CAL/*.JXL.
4.	**Export**	Export coordinates systems as Infinity coordinate system files in: • *.csys. • Global Transformation Sets TRFSET.dat files. • iCON/SBG Geo *.lok. • Trimble *.DC/*.CAL/*.JXL.
5.	**Copy to Project**	Copy selected coordinate system to the currently open project. ☞ You can only copy coordinate systems to the project which are not attached already.
6.	**Localise**	Open the Localisation Tool and download coordinate systems from the available global or national list.

See also:

Coordinate System Manager

Local and Global Data Objects

2.1.7.7 Coordinate System Manager

2.1.7.7.1 Overview

Coordinate System Manager

In the coordinate system manager, you can create and manage all the components that constitute a coordinate system, for example Transformations, Ellipsoids and Projections as well as Geoid Models and CSCS Models.

To open the coordinate system manager:

1. Select **File** > **Tools** > **Coordinate Systems**.
2. Select the **Manager** option to open the Coordinate System Manager.

The ribbon bar gives access to the following actions:

Data	**Import** existing coordinate systems from: • SmartWorx DBX (*.xcf). • *.dat files. • HeXML/LandXML. • LGO CSYS. • Trimble *.DC/*.CAL/*.JXL. • iCON/SBG Geo *.lok. • Infinity Coordinate System file*.csys.
	Export coordinates systems as: • Infinity Coordinate System files in *.csys. • Global Transformation Sets TRFSET.dat files. • iCON/SBG Geo *.lok.
	Copy to Project, to copy selected coordinate systems to the current project.
	Report, to get a coordinate system report.
	Create Geoid Field File, to create a new geoid field file.
	Create CSCS Field File, to create a new CSCS field file.
New	**Coordinate System**, to create a new coordinate system.
	Transformation, to create a new transformation.
	Ellipsoid, to create a new ellipsoid.
	Projection, to create a new projection.
	Geoid Model, to create a new geoid model.
	CSCS Model, to create a new CSCS model.
	Determine Transformation, to determine a new transformation.
Edit	**Delete** coordinate systems from the list.

☞ To be able to copy a coordinate system to a project, the project has to be open. You can only copy coordinate systems to the project which are not attached already.

In the Properties window you are shown the properties of any selected item, be that a coordinate system, a transformation, an ellipsoid, a projection, a geoid model or a CSCS model.

Some properties are editable. Confirm any changes with **Apply**.

In the status bar at the bottom:

- Select the option to hide or show the Properties window.
- Change the linear and angular units and the available decimal places if necessary.
- Change the way latitude and longitude are displayed if necessary.

The functionality is similar to the status bar in the main frame.

See also:

Status Bar

☞ If you have long lists use the **Search** functionality to find an item.

2.1.7.7.2

Create Geoid Field File

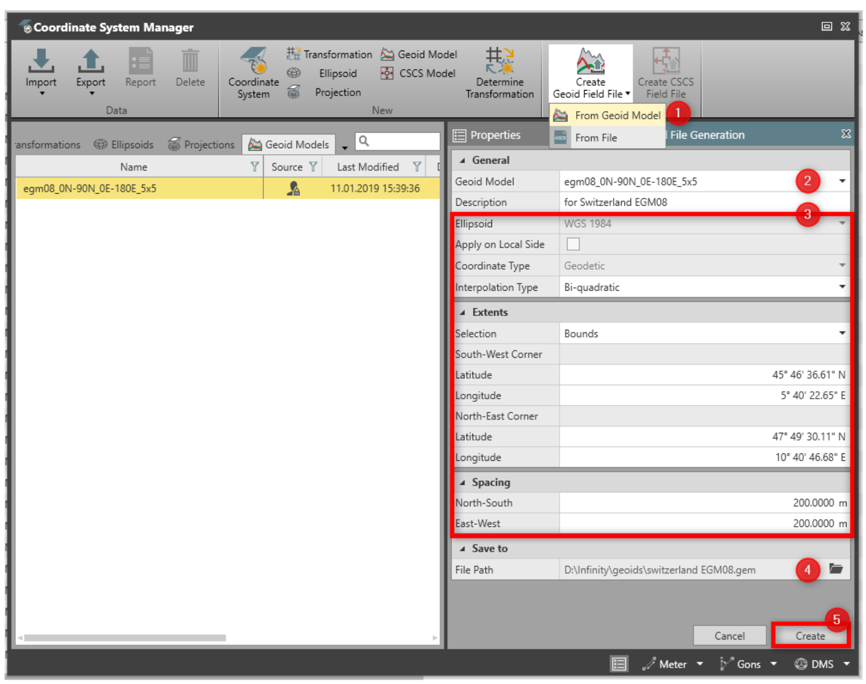
Create Geoid Field File

Geoid models may also be used on the receiver in the field. This command enables you to create a geoid field file.

Geoid models usually consist of a geoid height grid where a geoid separation is defined for each grid point. Depending on the extent and the grid spacing of the geoid model, it may require considerable disk space. In order to use the geoid model on a GPS sensor the disk space has to be reduced and a special field file has to be created which allows the field system to interpolate geoid separations.

From geoid model:

This command enables you to extract a geoid height grid from an existing geoid model for a particular area. The area boundary can be defined by a rectangle or circle and a grid spacing in metres can be selected. The file can then be uploaded to the receiver.



1. From the **Coordinate System Manager**, select **Create Geoid Field File/From File**.

2. Select an existing geoid model from the list.
If **Select source file** is selected from the list, browse to the existing .exe file.

☞ If the tool has been started from a project, the list with all geoids found in the project are shown.

☞ If the tool has been started from the File tab, all available geoids are shown.

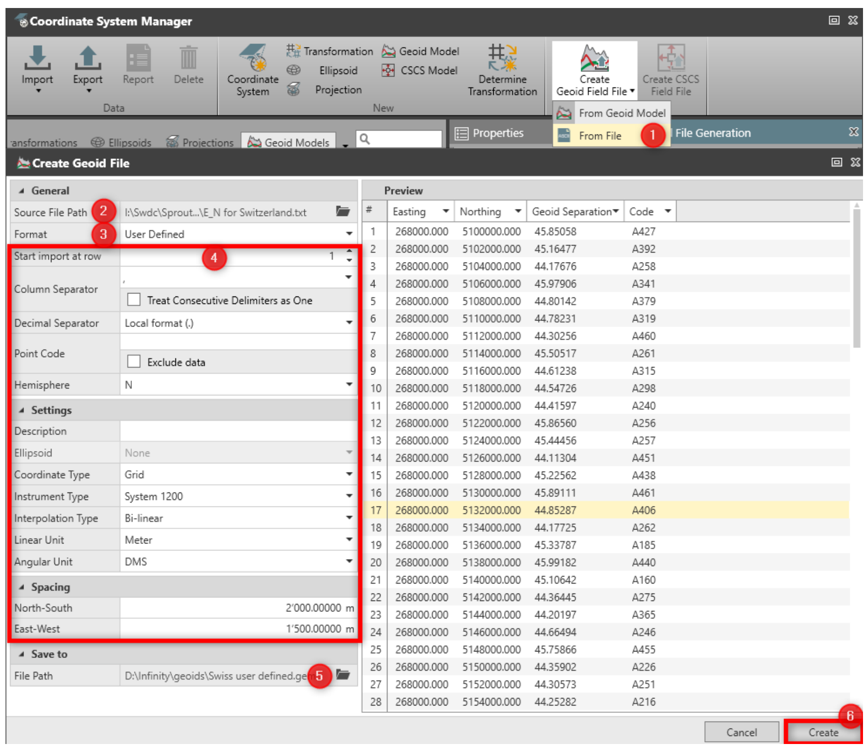
3. Define all needed settings.

4. From the file browser, select the path where the file shall be created (enter a file name without extension).

5. Select **Create** to finish the process.

From File:

You can create a geoid GEM file from locally provided geoid model data usually in ASCII format.



1. From the **Coordinate System Manager**, select **Create Geoid Field File/From File**.

2. Browse for an ASCII grid file.

3. Select the Format.
Select between **User Defined** or some **Predefined formats**.

4. Define all needed settings.

☞ When a predefined format is selected, the settings are filled in automatically.

5. From the file browser, select the path where the file shall be created.
Enter a file name without extension.

6. In the preview, identify the data type in each column by selecting a type from each drop-down menu.
Select **Create** to finish the process.

☞ Once created, the geoid model is automatically added to the list of geoids in the Coordinate System Manager.

2.1.7.7.3 Create CSCS Field File

Create CSCS Field File

CSCS models may also be used on the receiver in the field. This command enables you to create a CSCS model field file.

1. From the **Coordinate System Manager**, select **Create CSCS Field File**.
2. Select a CSCS model from the list and add a new CSCS model.
3. Select the Interpolation Type, which is used when interpolating in the CSCS field file. Select between **Bi-quadratic**, **Bi-linear** and **Spline** interpolation methods.
4. Select the method to define the limits of the CSCS model field file. Select between **Centre & radius** and **Extents**.
5. Enter the coordinates of the centre point and the radius or enter the coordinates of the South-west and North-east corner.
6. From the file browser, select the path where the file shall be created.
7. Enter a File name without extension. (Extension "csc" is added automatically).
8. Select the **Create** option to confirm.

☞ Depending on the file size, this may take a while.

☞ Check the file size. If you want to use the file on the system RAM, it must not exceed a certain size.
The maximum file size may vary depending on the free memory in the receivers system RAM. Refer to the technical reference manual on how to free system RAM of the receiver.

2.1.7.7.4 Coordinate Systems

Coordinate Systems

A coordinate system provides the information necessary to convert coordinates between different representations (cartesian, geodetic, grid) and to transform coordinates between the WGS84 and a local system.

Coordinate systems can be imported, exported and copied to project.

See also:

Coordinate System Manager

☞ To be able to copy a coordinate system to a project, the project has to be open. You can only copy coordinate systems to the project which are not attached already.

☞ Coordinate systems can always be deleted from the list of globally available coordinate systems.

Coordinate System Properties

The properties window is filled dynamically depending on the selected coordinate system. It shows you information on all the components which constitute a coordinate system.

Different constituting components can be chosen. Basically all transformations, ellipsoids, projections, geoid models and CSCS models that are available from within the global coordinate system management are available for selection, too. But which ones are available for a specific coordinate system, depends on validity.

The dependencies are as follows:

- When you select a coordinate system of with a projection of type customised, then the connected ellipsoid is predefined and cannot be edited. You cannot choose another. Customised projections are country-specific, predefined projections.
- If you select a projection then an ellipsoid must be selected too.
- When you select a coordinate system with a transformation that does not have common points stored with it, then there are no residuals to be distributed. This is typically the case with manually entered classic 3D transformations. Residual Distribution are set to *None* and cannot be edited.
- When you select a coordinate system with a transformation of type onestep, then Ellipsoid and Projection are set to *None* and cannot be edited because knowledge of the local map projection and the local ellipsoid is not needed with onestep transformations.
- When you select a coordinate system with a transformation of type twostep, then Ellipsoid and Projection must be read from the transformation and cannot be edited.

Geoid and CSCS models can be selected or imported.

- To import a geoid or a CSCS model select the option and then the **Select Field File** option from within the fly-out.
- Only valid models are available for selection. For further information on validity, see topics:
  Geoid Models
  CSCS Models

☞ The properties of a coordinate system can only be edited from within the properties window.

2.1.7.7.5 **Geoid Models**

Geoid Models

A Geoid Model can be defined for geodetic or grid coordinates and refers to a particular ellipsoid. With a Geoid Model attached to a coordinate system, geoid separations can be computed for the points in your project.

If geoid separations are available you can switch between viewing ellipsoidal and orthometric heights.

☞ Geoid Models are always an approximation of the actual geoid. In terms of accuracy, they may vary considerably and in particular global models should be used with care. If the accuracy of the Geoid Model is not known it might be safer to use local control points with orthometric heights and apply a transformation to approximate the local geoid. The classic 3D transformation can be used in areas where the geoid has a regular shape.

To import a Geoid Model:

1.	In the **Coordinate System Manager**, select the **Geoid Models** tab and then **Import** from the ribbon bar.
2.	Browse for the *.gem file to be imported and open it.

Create a new Geoid Model:

1.	In the **Coordinate System Manager**, select the **Geoid Models** tab and then **New** from the ribbon bar.
2.	In the Properties window, give the Geoid Model a unique Name.
3.	Select the option to browse for and select a Geoid Model Field file (*.gem).
4.	Select **Create** to create the new Geoid Model or **Cancel** to exit the function.

See also:

Coordinate System Manager

Types of Geoid Models

WGS84 Geodetic Models

Geoid Models of this type are of Coordinate Type Geodetic based on the WGS84 ellipsoid. To obtain local orthometric heights the geoid separations are applied to the ellipsoidal heights on the WGS84 coordinate side, that is, models of this type are never flagged to be applied on local side and they are only valid and can only be used with coordinate systems that either have transformation none or a transformation with height mode ellipsoidal, that is a transformation which flags the local heights as ellipsoidal.

Local Geodetic Models

Geoid Models of this type are of Coordinate Type Geodetic based on either a local ellipsoid or the WGS84 ellipsoid but flagged to be applied on local side. To obtain local orthometric heights the geoid separations are applied to the ellipsoidal heights on the local coordinate side. Models of this type are valid and can be used with coordinate systems that either have transformation none or classic 3D. The ellipsoid of the coordinate system must be the same as the ellipsoid used with the Geoid Model.

Local Grid Models

Geoid Models of this type are of Coordinate Type Grid based on Ellipsoid None. To obtain local orthometric heights the geoid separations are applied to the ellipsoidal heights on the local coordinate side. Models of this type are valid and can be used with coordinate systems which either have a projection defined or which are of transformation type onestep or twostep.

☞ If a local grid model is imported that is based on an ellipsoid other than none then the ellipsoid of the coordinate system with which the model shall be used must match the ellipsoid of the Geoid Model.

Geoid Model Properties

The Properties window is filled dynamically depending on the selected Geoid Model. It shows you the Path to the imported *.gem file, the Ellipsoid upon which the model is based, its Extents and Spacing.

For geodetic Geoid Models which are based on the WGS84 ellipsoid you can see if they are flagged to be Applied on Local Side. Valid WGS84 geodetic models never have this flag set. They are always applied on the WGS84 coordinate side. Valid local geodetic models must have the flag set if they are based on the WGS84 ellipsoid. Local grid models are always applied on the local side.

The Name of a Geoid Model can always be modified even if it is in use with a coordinate system.

☞ To be able to delete a Geoid Model it must not be in use with any coordinate system.

2.1.7.7.6

CSCS Models

CSCS Models

Several countries have produced tables of conversion factors to directly convert between GPS measured coordinates given in WGS84 and the corresponding local mapping coordinates, taking the distortions of the mapping system into account. Using these tables it is possible to directly convert into the local grid system without having to calculate your own transformation parameters. A Country Specific Coordinate System Model (CSCS Model) is an addition to an already defined coordinate system, which interpolates corrections in a grid file and applies the interpolated corrections. The extra step of applying these corrections can be made at different points in the coordinate conversion process. Therefore, different methods of CSCS Models are supported.

To import a CSCS Model:

1. In the **Coordinate System Manager**, select the **CSCS Models** tab and then **Import** from the ribbon bar.
2. Browse for the *.csc file to be imported and open it.

Create a new CSCS Model:

1. In the **Coordinate System Manager**, select the **CSCS Models** tab and then **New** from the ribbon bar.
2. In the Properties window give the geoid model unique Name.
3. Select the option to browse for and select a CSCS Model Field file (*.csc).
4. Select **Create** to create the new CSCS Model or **Cancel** to exit the function.

See also:

Coordinate System Manager

Conversion Methods

Grid conversion method (grid shifts):

The grid file is usually based upon local grid coordinates and can only be applied to coordinate systems for which a projection is defined.

1.	Step: Applying the specified transformation, map projection and ellipsoid to get preliminary grid coordinates.
2.	Step: Interpolation of a shift in Easting and Northing in the grid file of the CSCS Model to get the final local Eastings and Northings.

Cartesian conversion method (cartesian shifts):

The grid file can be based upon WGS84 geodetic or upon local geodetic coordinates and can only be applied to coordinate systems for which either a classic 3D transformation is defined or for which the transformation is none.

1.	Step: Applying the specified transformation.
2.	Step: Interpolation of a 3D shift in the grid file of the CSCS Model resulting in local cartesian coordinates.
3.	Step: Applying the specified local ellipsoid and map projection to get the final local Eastings and Northings.

Geodetic conversion method (geodetic shifts):

The grid file can be based upon WGS84 geodetic or upon local geodetic coordinates and can only be applied to coordinate systems for which the transformation is none.

1.	Step: Interpolation of a shift in geodetic latitude and longitude in the grid file of the CSCS Model resulting in final local geodetic coordinates.
2.	Step: Applying the map projection to get the final local Eastings and Northings.

Ellipsoidal conversion method (ellipsoidal shifts):

The grid file can be based upon WGS84 geodetic or upon local geodetic coordinates and can only be applied to coordinate systems for which an ellipsoid is defined.

1.	Step: Applying the specified transformation and the ellipsoid to get preliminary local geodetic coordinates.
2.	Step: Interpolation of a shift in geodetic latitude and longitude in the grid file of the CSCS Model resulting in final local geodetic coordinates.
3.	Step: Applying the map projection to get the final local Eastings and Northings.

CSCS Model Properties:

The Properties window is filled dynamically depending on the selected CSCS Model. It shows you the Path to the imported *.csc file, the Kind of conversion method it uses, its Geodetic Datum Kind (either WGS84 or Local) and its Coordinate Type (either Grid or Geodetic) as well as the Extents and Spacing of the model.

The Interpolation Type can be either Bi-linear, Bi-quadratic or Spline and is pre-defined by the imported grid file, too.

The Name of a CSCS Model can always be modified even if it is in use with a coordinate system.

☞ To be able to delete a CSCS model it must not be in use with any coordinate system.

2.1.7.7.7 Determine Transformation

2.1.7.7.7.1 Overview

Determine Transformation

In many surveying jobs it is necessary to transform the WGS84 coordinates into a local coordinate system or vice versa.

Using the Determine Transformation wizard, you can determine transformation parameters necessary to perform datum transformations between two sets of coordinates.

To determine a transformation it is necessary to have common control points whose positions are known in both WGS 1984 coordinates and local coordinates. The more points that are common between datum, the more accurately the transformation parameters can be calculated. Depending on the type of transformation used, details about the map projection, the local ellipsoid and a local geoid model can also be needed.

With one common control point, it is still possible to calculate a transformation.

The one point localisation method, allows you to fix specific parameters of the transformation. It requires only one point for the calculation. The method is valid for classic 3D, onestep and twostep transformations.

To use the one point localisation method, select the **Use one point localisation method** option in the Settings page of the Determine Transformation wizard.

Create a new transformation:

1. Select the **Coordinate System Manager** (either from the File tab or inside the project) and then **Determine Transformation** from the ribbon bar.
2. The Determine Transformation wizard consists of three steps:
   1. Determine Transformation Wizard: Settings.
   2. Determine Transformation Wizard: Match Points.
   3. Determine Transformation Wizard: Results.

Calculated transformations can be accessed and managed using the Coordinate System Manager.

Depending on the purpose of determining transformation parameters, there are four different transformation types supported in Infinity.

Quick Ground allows the transformation from local grid to ground coordinates, based on the selected coordinate system and the base point. The coordinates of the base point can be kept fixed during the transformation. More shifts in Northing and Easting can be applied, if necessary.

See also:

Classic 3D

Onestep

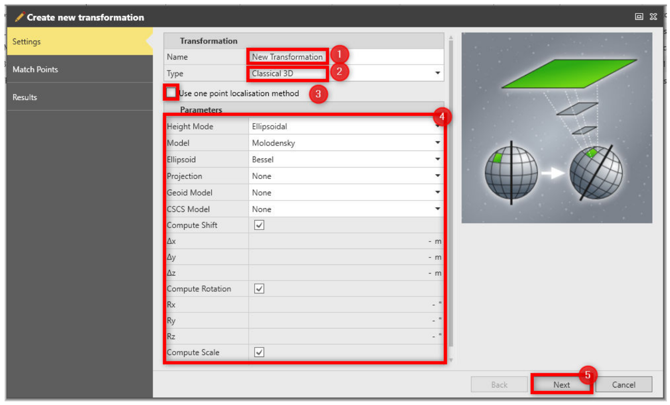
[Twostep](#)
[Which Approach to Use](#)
[Minimum Requirements for Coordinates](#)

2.1.7.7.7.2

Determine Transformation: Settings

Determine Transformation: Settings

To define the settings:



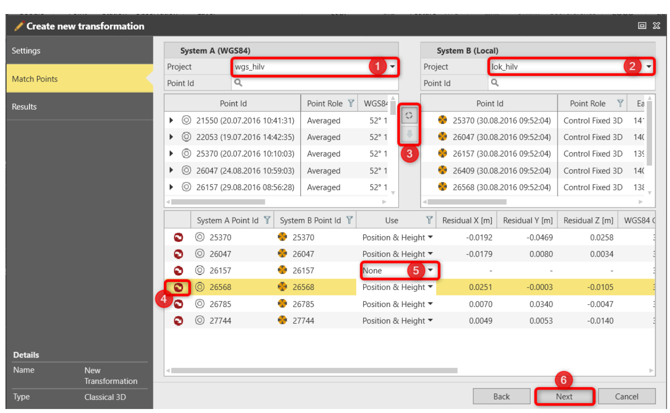
1.	Enter the transformation Name.	
2.	Select the transformation Type from the drop-down list: Classic 3D, Onestep or Twostep.	
3.	If you want to use a one point localisation, check the option **Use one point localisation method**.	
4.	Depending on the selected Type, choose the Parameters.	
	☞	You can enter the parameters for shift, rotation and scale manually or allow Infinity to compute them using common points.
5.	Select **Next** to proceed with Step 2: [Determine Transformation: Match Points](#).	

2.1.7.7.7.3

Determine Transformation: Match Points

Determine Transformation: Match Points

To match common points:



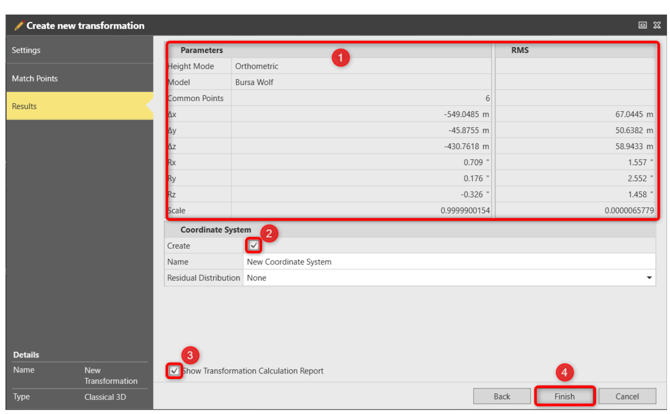
1.	Select a project to load the list of points for System A (WGS84).	
	☞	Only points stored in the project as WGS84 are shown.

2.	Select a project to load the list of points for System B (Local).	
	☞	Only points stored in the project as local grid are shown.
	☞	Points can be loaded into the list for System A (WGS84) and System B (Local) separately by selecting an Infinity project. These can be two different projects, but you can also select the same project for A and B.
3.	Match points:	
	Match the points manually: Available only for the single selection. Select a point in System A (WGS84) then select a point in System B (Local). Select the option to make a pair.	
	Match the points automatically: Available if nothing is selected or if at least one point which has a match is selected. The match looks for the same point ID from selected points and makes pairs automatically. Pairs always use the highest point roles.	
	To unselect a point press Ctrl+click.	
4.	: When selected, the pair is removed from the list. The calculation is updated based on remaining pairs.	
5.	Select the **Use**: Position & Height, Position, Height or None.	
	☞	None removes matched common points from the transformation calculation but does not delete them from the list. This option can be used to help improve residuals.
	☞	For classic 3D, only Position & Height or None are available.
6.	Select **Next** to proceed with Step 3: Determine Transformation: Results.	

2.1.7.7.7.4 Determine Transformation: Results

Determine Transformation: Results

To finalise the calculation:



1.	Review the calculated transformation Parameters.

2. Select the **Create** Coordinate System checkbox to create a new coordinate system with the newly calculated transformation. The new coordinate system is available for further usage in the coordinate system manager.
Decide on a Residual Distribution.

3. Deselect the **Show Transformation Calculation Report** checkbox to hide the report.

   ☞ A report is created by default and opens when you select Finish.

4. Select **Finish** to perform the operation.

2.1.7.7.7.5 **Classic 3D**

Classic 3D

The classic 3D transformation approach creates transformation parameters using a rigorous classic 3D method.

Basically, the method works by taking the cartesian coordinates of the GNSS measured points (WGS84 ellipsoid) and comparing them with the cartesian coordinates of the local coordinates. Shifts, rotations and a scale factor are calculated, to transform from one system to another.

The classic 3D transformation approach allows you to determine a maximum of seven transformation parameters (three shifts, three rotations, and one scale factor). However you can select the parameters to be determined.

The classic 3D transformation allows the choice of two different transformation models: Bursa-Wolf or Molodensky-Badekas.

For the classic 3D transformation method, we recommend that you have at least three points for which the coordinates are known in the local system and in WGS84. It is possible to compute transformation parameters using only three common points but using four produces more redundancy and allows for residuals to be calculated.

The advantage:

- The advantages of this method of calculating transformation parameters, are that it maintains the accuracy of the GNSS measurements and may be used over virtually any area as long as the local coordinates (including height) are accurate.

The disadvantage:

- The disadvantage is, that if local grid coordinates are desired, the local ellipsoid and map projections must be known. In addition, if the local coordinates are not themselves accurate, any new points measured using GNSS, may not fit into this existing local system once transformed.
- In order to obtain accurate ellipsoidal heights, the geoid separation at the measured points must be known. This may be determined from a geoid model, but many countries do not have access to an accurate local geoid model. See also Geoid Models.

Other transformation approaches:

Onestep

Twostep

Which Approach to Use

2.1.7.7.7.6 Onestep

Onestep

This transformation approach works by treating the height and position transformations separately. For the position transformation, the WGS84 coordinates are projected onto a temporary transverse mercator projection and then the shifts, rotation and scale from the temporary projection to the real projection are calculated.

The height transformation is a single dimension height approximation.

Because of how the position transformation approach works it is possible to define a transformation without any knowledge of the local map projection or local ellipsoid.

The height and position transformations are separate and therefore errors in height do not propagate into errors in position. Additionally, if knowledge of local heights is not good or non-existent you can still create a transformation for position only. Also, the height points and position points do not have to be the same points.

Because of how the transformation works it is possible to compute transformation parameters with just one point in the local and WGS84 system.

The combinations of the number of points in position and the position transformation parameters that can be calculated from them are as follows:

No. of position points	Transformation parameters computed
1	Classic 2D with shift in X and Y only.
2	Classic 2D with shift in X and Y, rotation about Z and scale.
More than 2	Classic 2D with shift in X and Y, rotation about Z, scale and residuals.

The number of points with height included in the transformation directly affects the type of height transformation produced.

No. of height points	Height transformation based on
0	No height transformation.
1	Constant height transformation.
2	Average constant between the two height points.
3	Plane through the three height points.
More than 3	Average plane.

The advantage:

- The advantages of this method are that transformation parameters may be computed using little information. No knowledge is needed of the local ellipsoid and map projection and parameters may be computed with the minimum of points. Care should be taken however when computing parameters using just one or two local points as the parameters calculated are only valid in the vicinity of the points used for the transformation.

The disadvantage:

- The area of the transformation is restricted to about 10 km^2 (using four common points).

Other transformation approaches:

Classic 3D

Twostep

Which Approach to Use

2.1.7.7.7.7

Twostep

Twostep

This transformation approach works by treating the height and position transformation separately. For the position transformation, the WGS 84 coordinates are first transformed using a classic 3D pre-transformation to obtain preliminary local cartesian coordinates. These are projected onto a preliminary grid using the specified ellipsoid and map projection. Then the two shifts, the rotation and the scale factor of a classic 2D transformation are calculated to transform the preliminary to the "real" local coordinates.

The position transformation requires knowledge of the local map projection and the local ellipsoid. However, as the distortions of the map projection are taken into account, twostep transformations can be used for larger areas than onestep transformations.

The height transformation is a single dimension height approximation.

The height and position transformations are separate and, therefore, errors in height do not propagate into errors in position. Additionally, if knowledge of local heights is not good or non-existent you can still create a transformation for position only. Also, the height points and position points do not have to be the same points.

Because of how the transformation works it is possible to compute transformation parameters with just one point in the local and WGS84 system.

The combinations of the number of points in position and the position transformation parameters that can be calculated from them are as follows:

No. of position points	Transformation parameters computed
1	Classic 2D with shift in X and Y only.
2	Classic 2D with shift in X and Y, rotation about Z and scale.
More than 2	Classic 2D with shift in X and Y, rotation about Z, scale and residuals.

The number of points with height included in the transformation directly affects the type of height transformation produced.

No. of height points	Height transformation based on
0	No height transformation.
1	Constant height transformation.
2	Average constant between the two height points.
3	Plane through the three height points.
More than 3	Average plane.

The advantage:

- Errors in local heights do not affect the position transformation.
- The points used for determining the position and height transformation do not necessarily have to be the same points.
- The distortions of the map projection are taken into account which enables you to use the transformation for larger areas.

The disadvantage:

- Knowledge of the local projection and local ellipsoid are required.

Other transformation approaches:

Classic 3D

Onestep

Which Approach to Use

Minimum Requirements for Coordinates

2.1.7.7.7.8

Which Approach to Use

Which Approach to Use

This question is almost impossible to answer since the approach used depends totally on local conditions and information.

If you want to keep the GNSS measurements totally homogenous and the information about the local map projection is available, the Classic 3D approach would be the most suitable.

For cases where there is no information regarding the ellipsoid and/or map projection and/or you want to force the GNSS measurements to tie in with local existing control, then the Onestep approach may be the most suitable.

The Twostep approach treats position and height information separately which allows for position only control points to be used as well. Compared to the onestep approach, information regarding the ellipsoid and map projection has to be known. The advantage is that this approach can be used for larger areas than the onestep.

2.1.7.7.7.9

Minimum Requirements for Coordinates

Minimum Requirements for Coordinates

The following list provides you with the minimum requirements for coordinate system A and B, necessary to calculate transformation parameters using the different transformation types. The coordinates either have to comply with the minimum requirements or the coordinate system attached must allow the conversion of the coordinates to the type required. For example, if coordinates are required in cartesian format but are available in geodetic format only, an ellipsoid must be defined allowing the system to convert to the appropriate format.

System	Classic 3D	Onestep	Twostep
System A	Cartesian + Ellipsoid	Cartesian	Cartesian
System B	Cartesian	Grid	Grid + Ellipsoid + Projection

☞ Cartesian + Ellipsoid means that the coordinates have to be available in cartesian or geodetic format plus an ellipsoid must be defined in the attached coordinate system.

The onestep and the twostep transformations require that the geoid model is of type grid. This means that geoids of type geodetic or geoids created from WGS84 or another ellipsoid cannot be used.

Additionally, when a transformation is determined it is important to state whether ellipsoidal or orthometric heights are intended to be used in the target system B. With this information being stored as part of the transformation definition (Height mode), the system knows in which direction the geoid separations have to be applied.

Type	Minimum requirements
Classic 3D 7 parameters	3 points with position + height
Classic 3D 3 shifts	1 point with position + height
Classic 3D 3 shifts + scale factor	2 points with position + height
Classic 3D 3 shifts + rotation about Z	2 points with position + height
Classic 3D 3 shifts + scale factor + rotation about Z	2 points with position + height
Classic 3D Other combinations	If 3 unknowns or less 1 point with position + height If 4,5 or 6 unknowns 2 points with position + height
Onestep	1 point with position only 1 point with height is required for the height part of the transformation to be determined.
Twostep	1 point with position only 1 point with height is required for the height part of the transformation to be determined.

Only if these requirements are met it is possible to see the points loaded in the lists and to move to the Result page with the Next option of the Match Points.

☞ Since a onestep transformation may be calculated without any given height information on the local side, the system takes the WGS84 ellipsoidal height as the local height in that case. This height is also displayed as ellipsoidal then.

☞ If a twostep transformation is calculated without any given height information on the local side, the system displays the height after applying only the pre-transformation as a local height.

2.1.7.8 Georeferenced Images

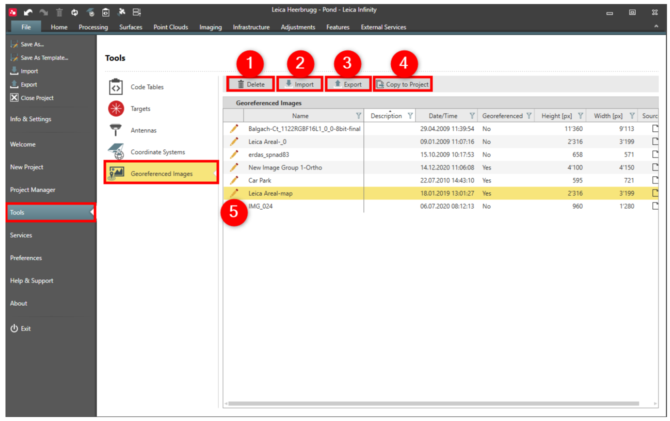
2.1.7.8.1 Overview

Georeferenced Images

Infinity supports the display of georeferenced images as a background. The use of these images is a nice way to help visualise, reference and relate your project data.

Georeferenced images are shown as objects in the library and can be set to visible or not visible. It is possible to use many georeferenced images. An existing georeferenced Image can be edited and updated with a new transformation.

In georeferenced images you can do the following:



No.	Name	Description
1.	Delete	Delete a georeferenced image from the global library.
2.	Import	Import georeferenced images or images you want to georeference.
3.	Export	Export georeferenced images.
4.	Copy to Project	Copy georeferenced images to be used in the project.
5.	Georeference Images	Open georeference image wizard to edit an existing or to calculate a new transformation.

See also:

Local and Global Data Objects

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

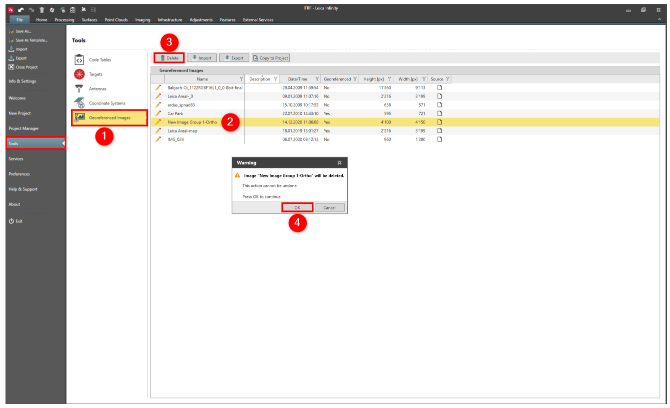
2.1.7.8.2

Delete

Delete

User-defined or imported georeferenced images can be deleted.

To delete images:



1. Select **File**, then **Tools** and then **Georeferenced Images** from the menu.
2. Select the image you want to delete.
3. Select **Delete**.
4. Select **OK**.

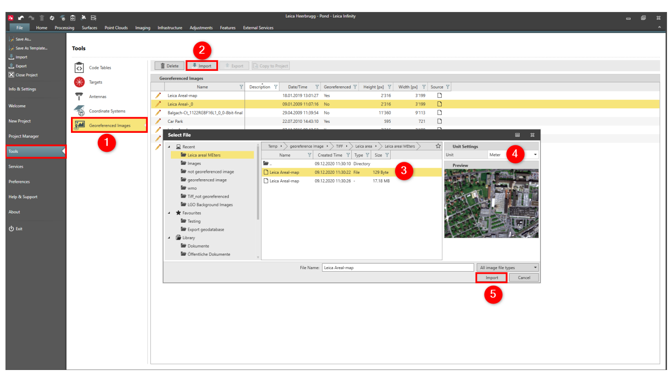
2.1.7.8.3 Import

Import

Images can be imported from PNG, JPG, TIFF or PDF.

When the images include a matching world file TIFF+TFW, PNG+PNW or JPG+JPW, then they are imported and translated as a georeferenced image. Otherwise it is possible to reference these images using the Georeference Images.

To import images:



1. Select **File**, then **Tools** and then **Georeferenced Images** from the menu.
2. Select **Import**.
3. Select the image you want to import.
4. Select the unit settings.
   ☞ You now see the image preview.
5. Select **Import**.

See also:

Local and Global Data Objects

Georeference Images

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

2.1.7.8.4 Export

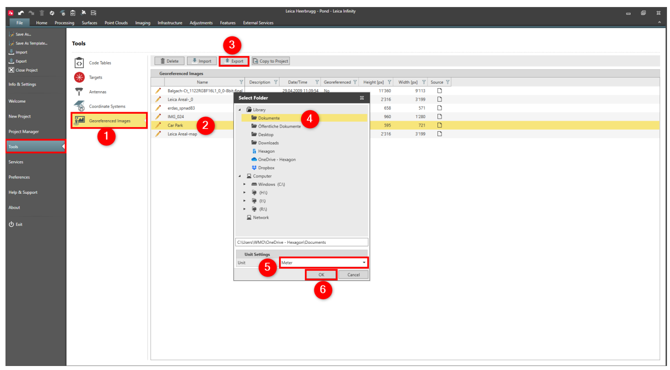
Export

Georeferenced images can be exported in JPG format with the matching JPW world file.

Requirements:

- The image must be georeferenced.

To export an image:



1.	Select **File**, then **Tools** and then **Georeferenced Images** from the menu.
2.	Select the image you want to export.
3.	Select **Export**.
4.	Select the file location.
5.	Select the unit settings.
6.	Select **OK**.

☞ To export a georeferenced image to Captivate simply attach it to a project and export the project to Captivate.

See also:

Local and Global Data Objects

Georeference Images

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

2.1.7.8.5 Copy to Project

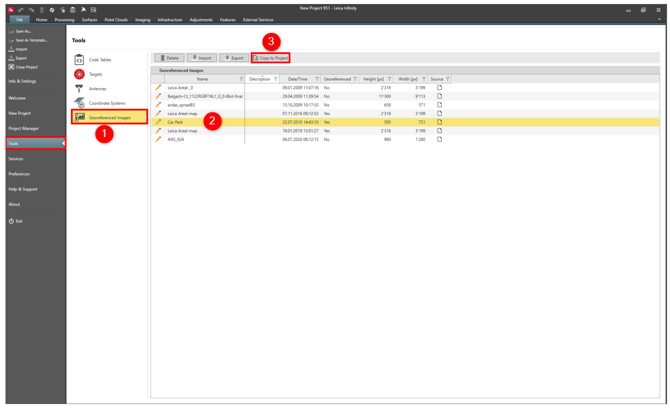
Copy to Project

Georeferenced images can be copied to the currently open project.

Requirements:

- The image must be georeferenced.
- The project must be opened before this option is available.

To copy to a project:



1.	Select **File**, then **Tools** and then **Georeferenced Images** from the menu.
2.	Select the image you want to copy.
3.	Select **Copy to Project**.

See also:

Local and Global Data Objects

Georeference Images

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

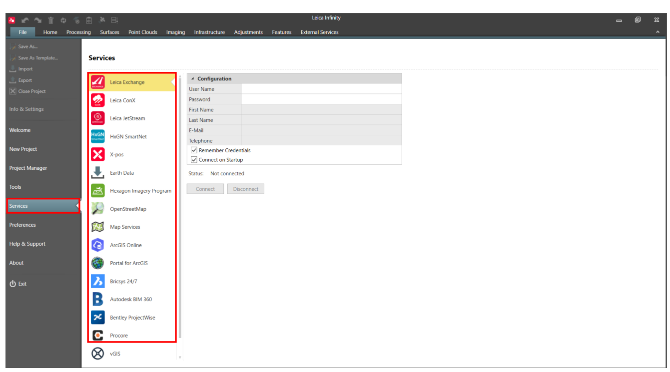
2.1.8 Services

2.1.8.1 Overview

Services

With data services by Leica, Hexagon and third parties are integrated with Infinity, access to data is seamless and effective.

Under services, there is a collection of external services which Infinity can communicate to and from.



Service	Description
Leica Exchange	Transfer data easily between the field and the office.
Leica ConX	Manage and share construction and survey data when working with iCON.

Service	Description
Leica JetStream	Allows you to publish point clouds to an existing storage location.
HxGN SmartNet	Access HxGN SmartNet to download reference station data for GNSS post-processing within Infinity.
X-pos	Access X-pos to download reference station data for GNSS post-processing within Infinity.
Earth Data	Allows you to log in to the data provider used by Infinity to download precise ephemeris and elevation data.
Hexagon Imagery Program	Access and use high-quality aerial imagery information for base maps.
OpenStreetMap	Access and use the open source map service.
Map Services	Define map services (WMS; WMTS; XYZ; Arc GIS Map Server) to use as base maps and feature services (WFS; Arc GIS Feature Server) to get data.
ArcGIS Online	Access feature servers to get data and upload data as web maps.
Portal for ArcGIS	Access feature servers to get data.
Bricsys 24/7	Share project data connecting to the Bricsys 24/7 service.
AUTODESK BIM 360	Access, download and upload field data.
Bentley ProjectWise	Transfer data between Infinity and the Bentley ProjectWise server.
Procore	Transfer data between Infinity and the Procore service.
vGIS	Transfer digital surface model data to the vGIS service.

☞ The Hexagon Imagery Program and OpenStreetMap services are only available within Infinity, if the computer has a valid maintenance end date/current Customer Care Package (CCP). All other services require subscriptions to be able to access them.

2.1.8.2 Leica Exchange

Leica Exchange

Infinity supports the Leica Exchange service.

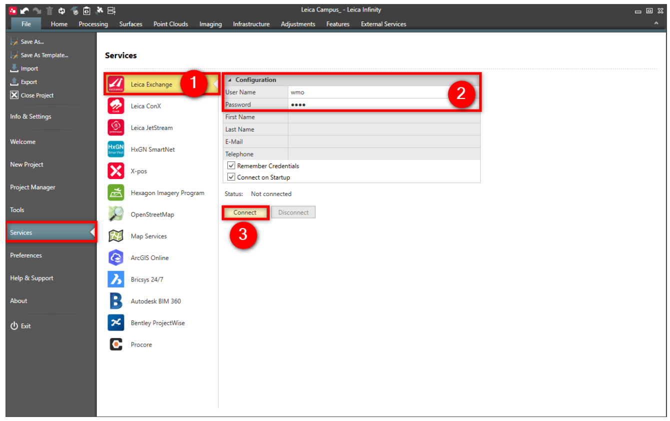
Transfer objects easily between the field and the office with Leica Exchange.

Requirements:

- Valid subscription.

☞ The yearly Leica Exchange service allows an unlimited number of exchanges to an unlimited number of users.

To connect to the Leica Exchange service:

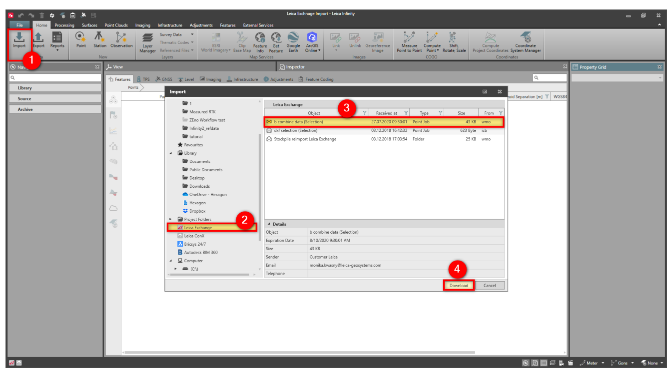


1. Select **File**, then **Services** and then **Leica Exchange** from the menu.
2. Enter your **User Name** and **Password**.
3. Select **Connect**.

To make it as easy as possible:

- Select remember credentials.
- Connect on start-up.

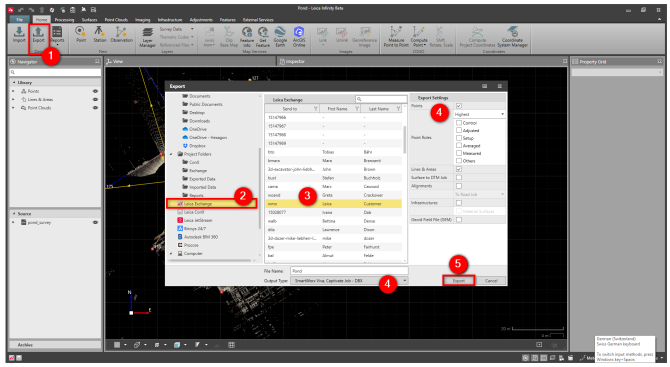
To import data:



1. Select **Import** from the Home tab.
2. In the Import window, select **Leica Exchange**. View the available files from the Bricsys 24/7 window.
3. Highlight the file to import to the project.
4. Select **Download**.

   ☞ The selected file is imported to Infinity and downloaded to the Leica Exchange directory, in the current Infinity project directory.
5. Select the data and if necessary specify import settings.
6. Select **Import**.

To export data:



1.	Select **Export** from the Home tab.
2.	In the Export window, select **Leica Exchange**.
3.	In the Leica Exchange window, select the user or users to send data to.
4.	Select the output type and export settings.
5.	Select **Export**.

☞ The Leica Exchange data is always copied to your computer. From the archive, you can always access the location from the computer the data is stored on.

See also:

https://leica-geosystems.com/services-and-support/workflow-services/leica-exchange

The tutorial **"How to send and receive the data to and from the field - Working with Leica Exchange"** https://leica-geosystems.com/-/media/279c6f19b6ce49479aa9942529c85994.ashx

☞ The tutorial can be downloaded in the Localisation Tool.

2.1.8.3 Leica ConX

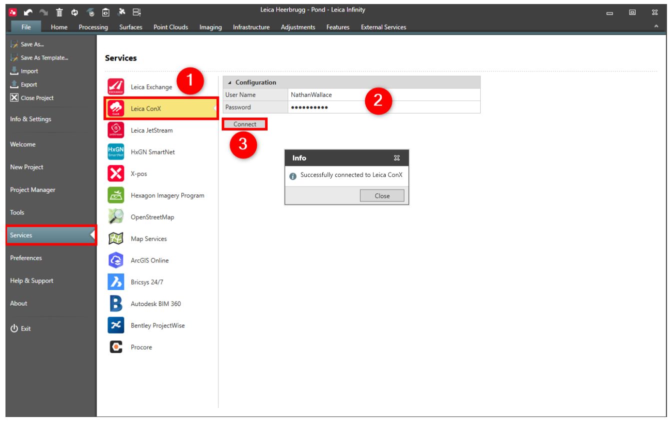
Leica ConX

Infinity supports the Leica ConX service. With the Leica ConX service, you can log in to iCON Build/iCON Site projects and assign data to the project or directly to the units, that are the machines working on building sites.

Requirements:

- Valid subscription.

To connect to the Leica ConX service:



1. Select **File**, then **Services** and then **Leica ConX** from the menu.

2. Enter your **User Name** and **Password**.

   ☞ You can enter a default user name and password. If you do so, the user name and the password are automatically populated when you select login from within the external services module.

3. Select **Connect**.

   ☞ You are connected until you log out from within the external services module.

See also:

Leica ConX

Export to Leica ConX

2.1.8.4 Leica JetStream

Leica JetStream

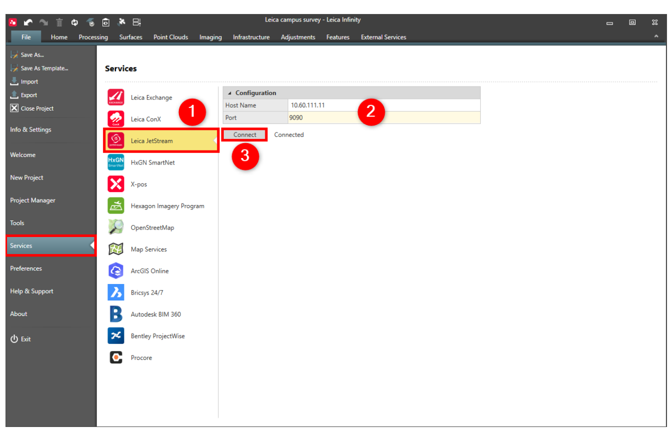
Infinity supports the Leica JetStream service.

You can publish point clouds to existing storage locations.

Requirements:

- Cyclon Publisher or Cyclon Publisher Pro licence (from Infinity 3.4.2).

To connect to the Leica JetStream service:



1.	Select **File**, then **Services** and then **Leica JetStream** from the menu.
2.	Define the host name and the port.
3.	Select **Connect**.

See also:

Publish to Leica JetStream

https://leica-geosystems.com/products/laser-scanners/software/leica-jet-stream

2.1.8.5 HxGN SmartNet

HxGN SmartNet

Infinity supports the HxGN SmartNet service.

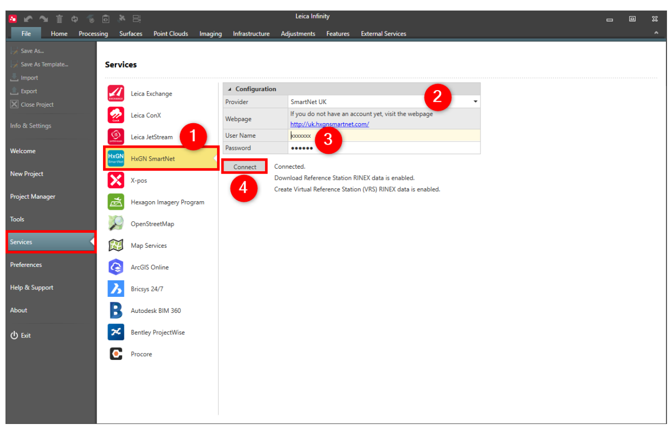
The HxGN SmartNet service allows:

- The download of post-processing data from GNSS reference station service providers directly to a project.
- The creation of Virtual Reference Stations (VRS) directly in a project.
- Valid subscription.

Requirements:

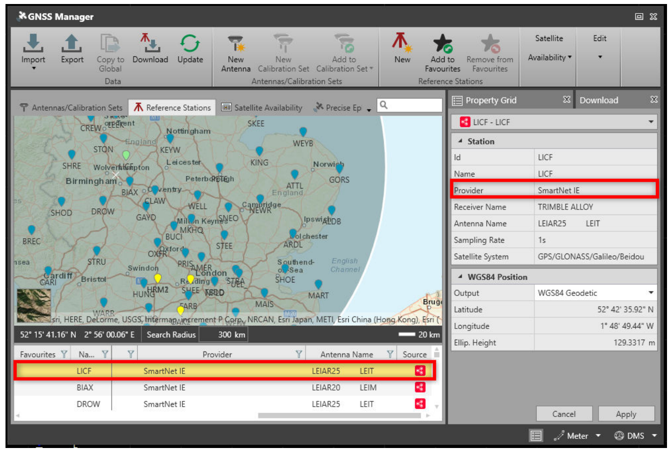
- Valid subscription.

To connect to the HxGN SmartNet service:



1.	Select **File**, then **Services** and then **Leica ConX** from the menu.
2.	Select a **Provider** in your region from the drop-down menu. The list is updated regularly online. Or select to enter the host manually and type the providers **Address** in the following field. ☞ If there is no HxGN SmartNet service provider listed for your region, contact your local Leica Geosystems sales representative to find out more.
3.	Log in with your **User Name** and **Password**. ☞ If you do not have a subscription, select the hyperlink to go to the providers webpage.
4.	Select **Connect**.

When logged in, you see all HxGN SmartNet reference stations in the **GNSS Manager > Reference Stations** tab. HxGN SmartNet reference stations are indicated with blue station icons in the map view and have got HxGN SmartNet listed as the provider in the Inspector view. Download works in the same way as for all other stations.



With the HxGN SmartNet licence, you can create VRS from the GNSS ribbon bar by selecting **Create VRS** after having highlighted a GNSS interval. The VRS is created automatically and imported into the project, starting from the coordinates of the highlighted GNSS interval.

See also:

GNSS Manager

Create a Virtual Reference Station (VRS)

The tutorial **"How to download reference data"** https://leica-geosystems.com/-/media/817c1f0ca2634823b2da45a5359cefec.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

2.1.8.6 X-pos

X-pos

Infinity supports the X-pos service.

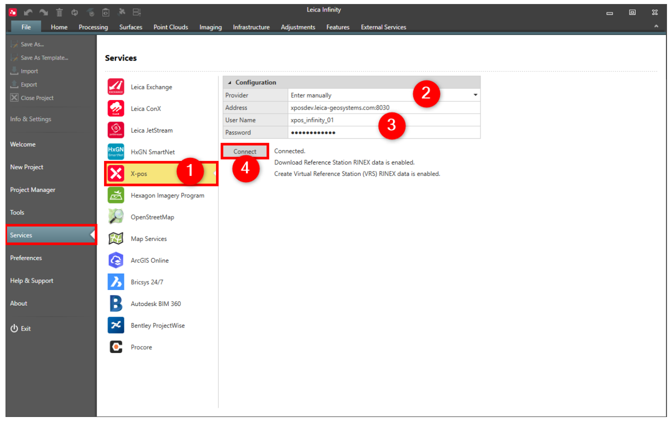
The X-pos service allows:

- The download of post-processing data from GNSS reference station service providers directly to a project.
- The creation of Virtual Reference Stations (VRS) directly in a project.
- Valid subscription.

Requirements:

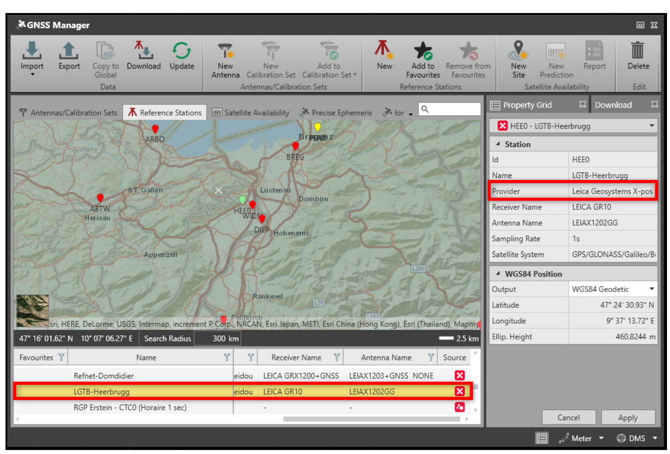
- Valid subscription.

To connect to the X-pos service:



1. Select **File**, then **Services** and then **X-pos** from the menu.
2. Select a **Provider** in your region from the drop-down menu. The list is updated regularly online.
   Or select to enter the host manually and type the providers address in the following field.
   ☞ If there is no X-pos service provider listed for your region, contact your local Leica Geosystems sales representative to find out more.
3. Log in with your **User Name** and **Password**.
   ☞ If you do not have a subscription, select the hyperlink to go to the providers webpage.
4. Select **Connect**.

When logged in, you see all X-pos reference stations in the **GNSS Manager** > **Reference Stations** tab. X-pos reference stations are indicated with red station icons in the map view and have got X-pos listed as the provider in the Inspector view. Download works in the same way as for all other stations.



With the X-pos licence, you can create VRS from the GNSS ribbon bar by selecting **Create VRS** after having highlighted a GNSS interval. The VRS is created automatically and imported into the project, starting from the coordinates of the highlighted GNSS interval.

See also:

GNSS Manager

Create a Virtual Reference Station (VRS)

The tutorial **"How to download reference data"** https://leica-geosystems.com/-/media/817c1f0ca2634823b2da45a5359cefec.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

2.1.8.7 Earth Data

Earth Data

Infinity supports the download of several GNSS data, such as reference stations, precise ephemeris or navigation files.

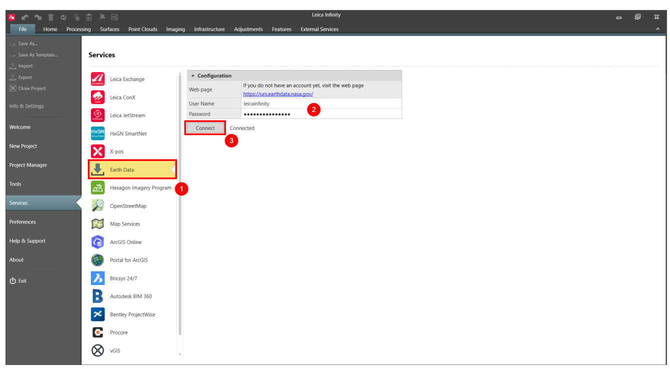
☞ To access the data, some providers require you to register and log in. Under **File** > **Services**, you can log in to the Earth Data provider.

Log in to Earth Data

Earth Data is used by Infinity to download precise and rapid ephemeris from the Crustal Dynamics Data Information System (CDDIS) archive https://cddis.nasa.gov/archive/gps/products/mgex/

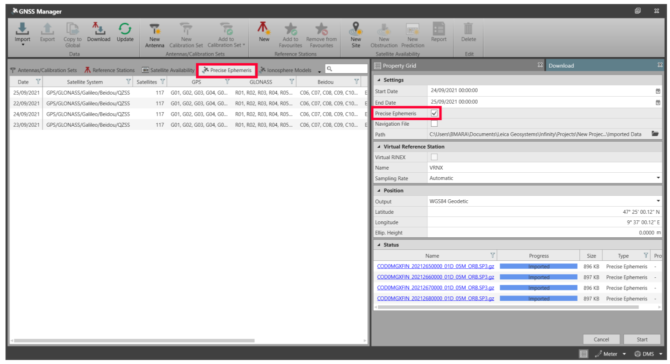
If you do not have an account yet, visit the web page Earth Data Login https://urs.earthdata.nasa.gov/

To log in with your credentials:



1. Select **File**, then **Services** and then **Earth Data** from the menu.
2. Enter your **User Name** and **Password**.
3. Select **Connect**.

When logged in you can download precise ephemeris from the **GNSS Manager** > **Precise Ephemeris** tab by enabling the **Precise Ephemeris** checkbox.



See also:

4.

2.1.8.8 Hexagon Imagery Program

Hexagon Imagery Program

Infinity supports the Hexagon Imagery Program service. Also referred to as HxIP.

Requirements:

- Valid Infinity maintenance Customer Care Package (CCP).

This service is a cached basemap imagery service consisting of up to 30 cm of true colour imagery that is updated on an ongoing basis. This service provides a backdrop to your Infinity projects which is a nice way to help visualise, reference and relate your project data. It can be loaded and displayed in a project and with the Clip Base Map function, downloaded as a georeferenced image and available for export to the field software. This service is provided free of charge for Infinity customers having an active CCP. The software reads from the Infinity Electronic Identification (EID) licence, the information about the maintenance period. When the maintenance period has expired, no access is available to the imagery.

See also:

Base Map

https://hxgncontent.com/imagery

The video **"Leica Infinity - Home Module - Using Base Maps"** https://www.youtube.com/watch?v=ZPXJSI3WWVM

2.1.8.9 OpenStreetMap

OpenStreetMap

Infinity supports the OpenStreetMap service. Also referred to as OSM.

Requirements:

- Valid Infinity maintenance Customer Care Package (CCP).

This service provides a backdrop to your Infinity projects which is a nice way to help visualise, reference and relate your project data. It provides open source maps that can be loaded, displayed in a project and with the Clip Base Map function, downloaded as a georeferenced image and available for export to the field software. This service is provided free of charge for Infinity customers having an active CCP. The software reads from the Infinity Electronic Identification (EID) licence the information about the maintenance period. When the maintenance period has expired, no access is available to the OpenStreetMap service.

See also:

Base Map

https://www.openstreetmap.org/copyright

The video **"Leica Infinity - Home Module - Using Base Maps"** https://www.youtube.com/watch?v=ZPXJSI3WWVM

2.1.8.10 Map Services

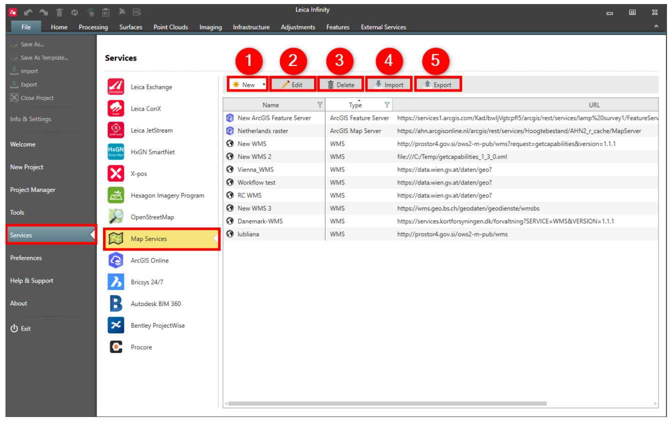
2.1.8.10.1 Overview

Map Services

Infinity supports the following map services:

Map Service	Description
WMS (Web Map Service)	A WMS delivers georeferenced map images. A WMS service provides various layers that can be turned on and off. WMS focuses on rendering custom maps and is an ideal solution for dynamic data or custom styled maps.
WMTS (Web Map Tile Service)	A WMTS delivers pre-generated georeferenced map images. A WMTS service can have one or more styles, dimensions, or tiling schemes to specify how the WMTS layer is displayed. WMTS trades the flexibility of custom map rendering for the scalability possible by serving of static data where the bounding box and scales have been constrained to discrete tiles.
WFS (Web Feature Service)	A WFS serves, queries, and updates feature geometry and attributes using a Geography Markup Language (GML) profile.
XYZ Tiles (Tile Map Service)	Known as TMS. This service provides tiled images. X, Y, Z are tile numbers corresponding to a particular area.
ArcGIS Map Server	Map service (tile layer, map image layer) hosted by ArcGIS Online or ArcGIS Server, can be used as a Base Map.
ArcGIS Feature Server	A feature service (feature layer) hosted by ArcGIS Online or ArcGIS Server. Used in Get Feature to download feature geometry and attributes.

In map services you can do the following:



No.	Name	Description
1.	New	Add a new map service.
2.	Edit	Edit an existing map service.
3.	Delete	Delete selected map service.
4.	Import	Import a list of configured WMS/WMTS/WFS services from the Captivate web services list format.
5.	Export	Export selected services to Captivate web services list format.

See also:

Get Feature

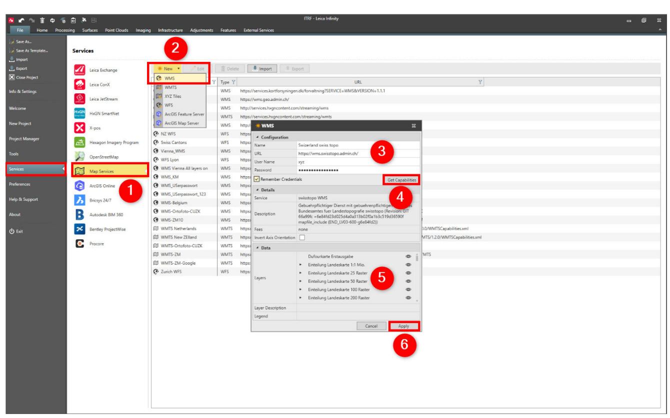
Base Map

The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

2.1.8.10.2 New

New

To add a new map service:



1.	Select **File**, then **Services** and then **Map Services** from the menu.
2.	Select **New** and choose the service type.

3.	Insert all necessary login information in the Configuration table. • **Name**: The name of map service. • **URL**: The web address of map service. • **User Name**: Your user name. • **Password**: A valid password.	
4.	Select **Get Capabilities** to connect to the map service. ☞ Information about the map service appears in the Detail table after connection. ☞ Activate the checkbox **Remember Credentials** to save the login information.	
5.	**For WMTS only:** In the Data table, you can select various Layers and Styles. Click the arrow in the table field to open a drop-down menu of options. In the drop-down menu, select the layer/style to define a default layer/style for your project.	
	For WMS only: In the Data table, various layers are displayed. Select the eye icon, to turn the layers on and off. ☞ To switch all the layers on/off in one go, press shift and select the first and last layer.	
6.	Select **Apply**.	

See also:

Get Feature

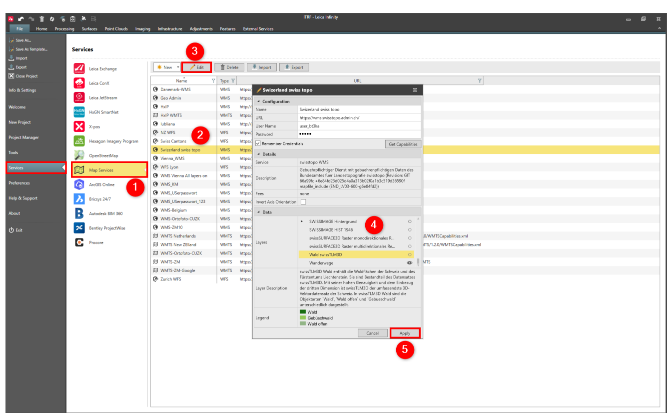
Base Map

The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

2.1.8.10.3 **Edit**

Edit

To edit the map service:



1.	Select **File**, then **Services** and then **Map Services** from the menu.
2.	Select the map service you want to modify from the list.
3.	Select **Edit** from the ribbon bar or the context menu.
4.	Do the changes.

5. Select **Apply**.

See also:

Get Feature

Base Map

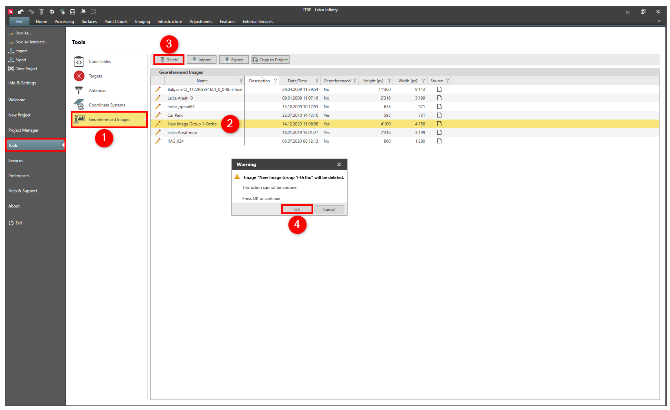
The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

---

2.1.8.10.4 Delete

Delete

To delete a map service:



1. Select **File**, then **Services** and then **Map Services** from the menu.
2. Select the map service you want to delete from the list.
3. Select **Delete** from the ribbon bar or context menu.
4. Select **OK**.

See also:

Get Feature

Base Map

The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

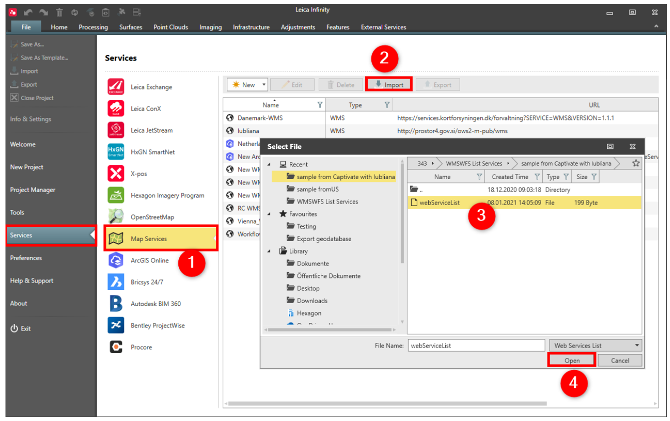
---

2.1.8.10.5 Import

Import

The entire list of configured WMS/WMTS/WFS services (including connection name, URL, user name) can be imported from Captivate or from another Infinity user using the webServiceList.fil format.

To import web service list:



1. Select **File**, then **Services** and then **Map Services** from the menu.
2. Select **Import**.
3. Navigate to the location.
4. Select **Open**.

☞ To save the web service list in Captivate use transfer user objects.

☞ Passwords are not exported, add them manually in Captivate or another Infinity instance.

See also:

Get Feature

Base Map

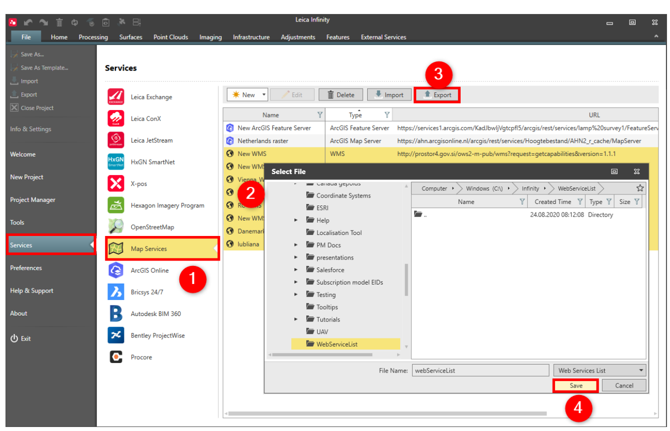
The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

2.1.8.10.6 Export

Export

Existing WMS/WMTS/WFS services (including connection name, URL, user name) can be exported to webServiceList.fil format and then used in Captivate or by another Infinity instance.

To export selected WMS/WMTS/WFS to web service list:



1. Select **File**, then **Services** and then **Map Services** from the menu.

2. Select the WMS//WMTS/WFS services you want to export.
3. Select **Export**.
4. Select the folder location to export to.
5. Select **Save**.

☞ To load the web service list in Captivate use transfer user objects.

☞ Passwords are not exported, add them manually in Captivate or another Infinity instance.

See also:

Get Feature

Base Map

The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

2.1.8.11 ArcGIS Online

ArcGIS Online

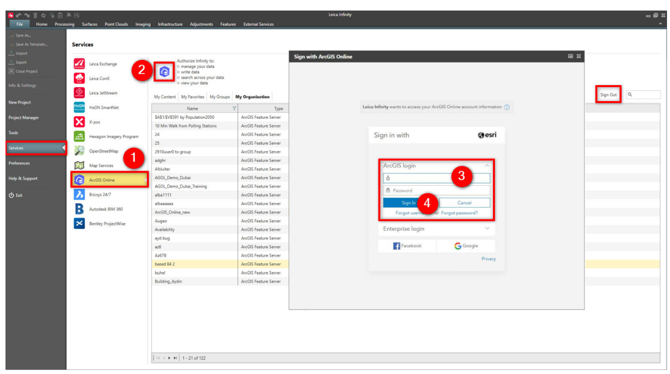
Infinity supports the ArcGIS Online service. ArcGIS Online is a cloud-based mapping and analysis solution from ESRI.

You can get access to the feature servers to get data as well as upload existing projects as web maps.

Requirements:

- Existing account and valid subscription.

To connect to the ArcGIS Online service:



1. Select **File**, then **Services** and then **ArcGIS Online** from the menu.
2. Select **ArcGIS Online**.
3. Enter your **User Name** and **Password**.
4. Select **Sign In**.

   ☞ You are connected until you select to **Sign Out**.

When logged in, you can see all supported feature servers grouped as in ArcGIS Online into: **My Content** > **My Favourites** > **My Groups** > **My Organisation**.

☞ Living Atlas from ArcGIS Online is not seen, however selected data can easily be copied to **My Favourites** or **My Groups**.

See also:

Get Feature

Save as an ArcGIS Online Web Map

Map Services

The video **"Leica Infinity - Services - How to use the ArcGIS Online service"** https://www.youtube.com/watch?v=QgFY17R990o

2.1.8.12 Portal for ArcGIS

Portal for ArcGIS

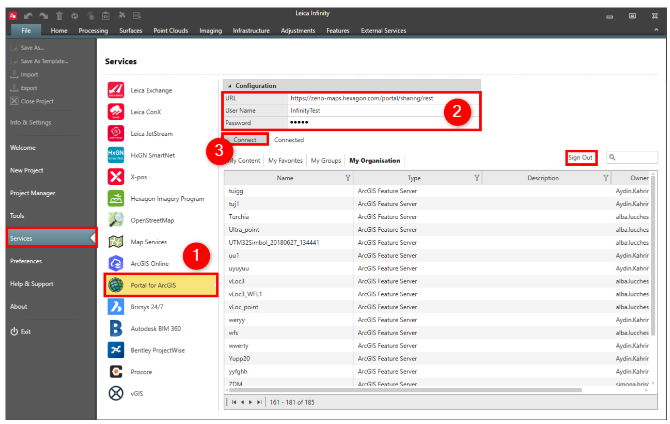
Infinity supports the Portal for ArcGIS, user-defined ArcGIS on premises server.

You can get access to the feature servers to get data.

Requirements:

- ArcGIS Online account hosted on the customers own premises.

To connect to the Portal for ArcGIS service:



1. Select **File**, then **Services** and then **Portal for ArcGIS** from the menu.
2. Enter the **URL**, **User Name** and **Password**.
3. Select **Connect**.

   ☞ You are connected until you select to **Sign Out**.

When connected, you can see all supported feature servers grouped as in Portal for ArcGIS: **My Content** > **My Favourites** > **My Groups** > **My Organisation**.

See also:

Get Feature

Save as Portal for ArcGIS Web Map

https://enterprise.arcgis.com/en/portal/10.4/use/what-is-portal-for-arcgis-.htm

2.1.8.13 Bricsys 24/7

Bricsys 24/7

Infinity supports the Bricsys 24/7 cloud service for sharing project data.

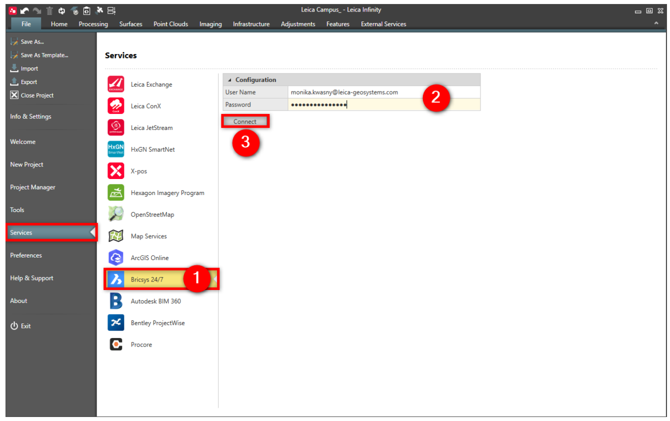
It gives users easy access to their projects and to the data required to prepare for field campaigns.

Being connected also simplifies providing processed data and data deliverables to the project by uploading directly from Infinity.

Requirements:

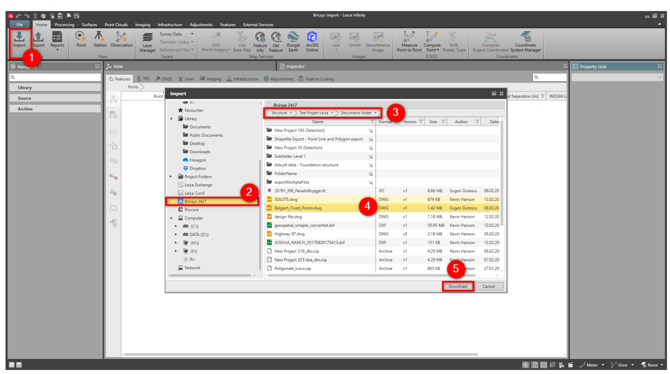
- Valid subscription.

To connect to the Bricsys 24/7 service:



1. Select **File**, then **Services** and then **Bricsys 24/7** from the menu.
2. Enter your **User Name** and **Password**.
3. Select **Connect**.

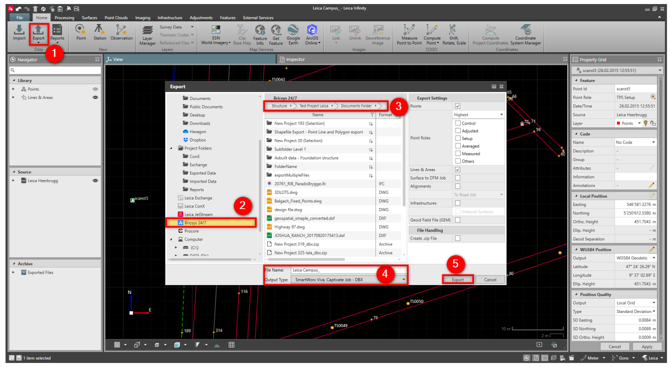
To import data:



1. Select **Import** from the Home tab.
2. In the Import window, select **Bricsys 24/7**.
3. View the available files in the Bricsys 24/7 window.
4. Highlight the file to import to the project.
5. Select **Download**.

   ☞ The selected file is imported to Infinity and downloaded to the Bricsys 24/7 directory, in the current Infinity project directory.
6. Select the data and if necessary specify import settings.
7. Select **Import**.

To export data:



1.	Select **Export** from the Home tab.
2.	In the Export window, select **Bricsys 24/7**.
3.	From the Bricsys 24/7 window, select the directory to export the data to.
4.	Select the data format and export settings.
5.	Select **Export**.

☞ The Bricsys 24/7 data is always copied to the current Infinity project directory.

2.1.8.14 AUTODESK BIM 360

AUTODESK BIM 360

Infinity supports the AUTODESK BIM 360 service, for an easy and direct collaboration with construction projects.

Log in directly to your project services to access, download and upload field data.

Requirements:

- AUTODESK BIM 360 account and subscription.

You are guided through the steps to log in and authorise Infinity to connect to the AUTODESK BIM 360 service.

See also:

The video **"Leica Infinity with Autodesk BIM 360 service"** https://www.youtube.com/watch?v=QMMPzShF2UA&t=0s&list=PL0t-d7rOVk_IV_al3ziSKuAYA1VVu6W0rM&index=2

2.1.8.15 Bentley ProjectWise

Bentley ProjectWise

Infinity supports the Bentley ProjectWise service.

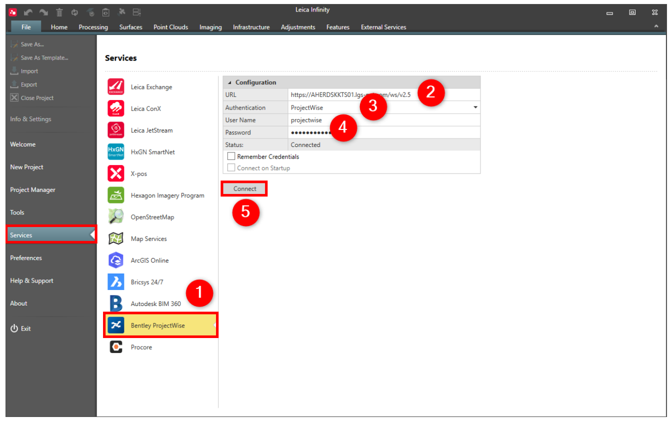
Transfer objects easily between Infinity and the ProjectWise server.

Infinity currently supports version 2.0 and above of the ProjectWise Application Program Interface (API).

Requirements:

- Valid subscription.

To connect to the Bentley ProjectWise service:



1. Select **File**, then **Services** and then **Bentley ProjectWise** from the menu.

2. Enter the **URL**.
   To use the Bentley Projectwise service, enter the full URL, of the Web Service Gateway instance.
   - For example: https://myserver.com/ws/v2.5 (where "v2.5" represents the Projectwise API version).
   - Contact your organisation Projectwise administrator to obtain the URL.

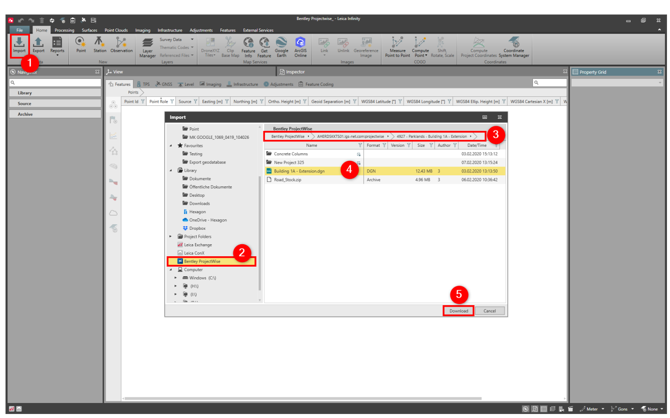
3. Select the Authentication method from the drop-down list.
   Choose to authenticate through:
   - **Bentley IMS** (Federated Identity) - Choose this option when the Bentley Connection Client is required for the ProjectWise connection.
   - **ProjectWise** (Non-Federated Identity) - Choose this option when the Bentley Connection Client is not required for the ProjectWise connection.

4. Enter the **User Name** and the **Password**.

5. Select **Connect**.

To make it as easy as possible:
- Select remember credentials.
- Connect on start-up.

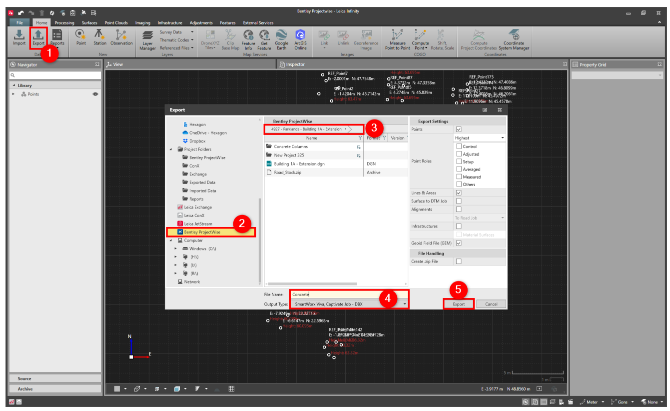
To import data:



1. Select **Import** from the Home tab.
2. In the Import window, select **Bentley ProjectWise** from the drive list.
3. View the available files in the Bentley ProjectWise window.
4. Highlight the file to import to the project and if necessary specify the import settings.

   ☞ The selected file is imported to Infinity and downloaded to the Bentley ProjectWise directory, in the current Infinity project directory.
5. Select **Download**.

To export data:



1. Select **Export** from the Home tab.
2. In the Export window, select **Bentley ProjectWise** from the drive list.
3. In the Bentley ProjectWise window, select the location to export the data to.
4. Enter a **File Name** and select the **Output Type**.
5. Select **Export**.

☞ The Bentley ProjectWise data is always copied to the current Infinity project directory.

See also:

The video **"Leica Infinity - Services - How to use Bentley ProjectWise"**
https://www.youtube.com/watch?v=0DEvgPXL4Mg

2.1.8.16 Procore

Procore

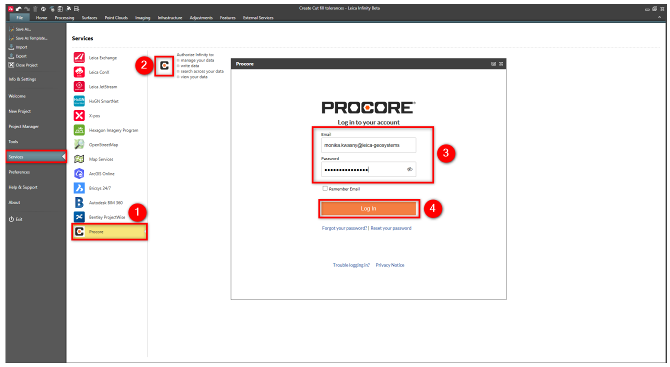
Infinity supports the Procore service.

Easily transfer data between Infinity and the Procore service.

Requirements:

- Valid subscription.

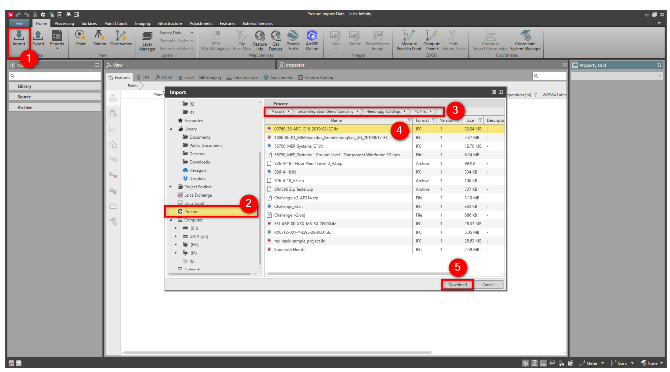
To connect to the Procore service:



1. Select **File**, then **Services** and then **Procore** from the menu.
2. Select **Procore**.
3. Enter your **User Name** and **Password**.
4. Select **Log In**.

   ☞ Login is controlled with Internet Explorer autocomplete settings. Selecting the Procore icon logs the current user out of the Procore service.

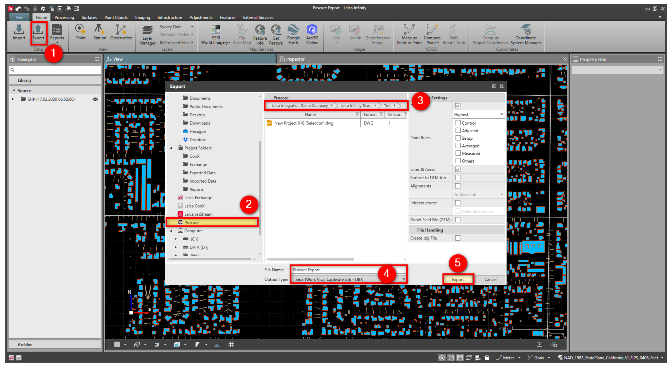
To import data:



1. Select **Import** from the Home tab.
2. In the Import window, select **Procore**.
3. View the available files in the Procore window.
4. Highlight the file to import to the project.
5. Select **Download**.

   ☞ The selected file is imported to Infinity and downloaded to the Procore directory, in the current Infinity project directory.
6. Select the data and if necessary specify import settings.
7. Select **Import**.

To export data:



1. Select **Export** from the Home tab.
2. In the Export window, select **Procore**.
3. From the Procore window, select the directory to export the data to.
4. Select the data format and export settings.
5. Select **Export**.

☞ Depending on the project settings, Procore data is either copied to the current Infinity project directory or the downloads folder.

☞ If the file name already exists, it is uploaded as a new version in the Procore directory.

☞ Folders cannot be updated if the folder name already exists in the Procore directory.

See also:

The video **"Leica Infinity - Services - How to use the Procore service"** https://www.youtube.com/watch?v=ZtmIA7XlQ0M

2.1.8.17 vGIS

vGIS

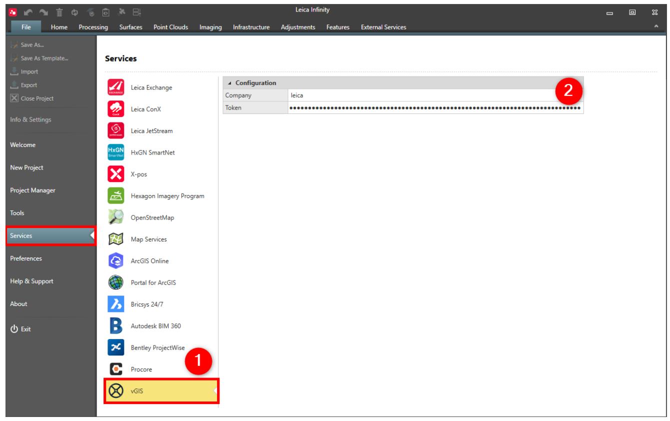
Infinity supports sending data to vGIS. vGIS is a leading extended reality (XR) visualisation platform that transforms traditional BIM, GIS, reality capture and other types of data, into practical augmented reality overlays.

Creating digital surface models from images acquired by UAV or GS18 I, you can share and visualise the models in the vGIS augmented reality field application.

Requirements:

- A token provided by vGIS or your customer is required to access the required service location.

To connect to the vGIS service:



1.	Select **File**, then **Services** and then **vGIS** from the menu.
2.	Enter the **Company**, then the **Token** and then press enter.

See also:

Publish to vGIS

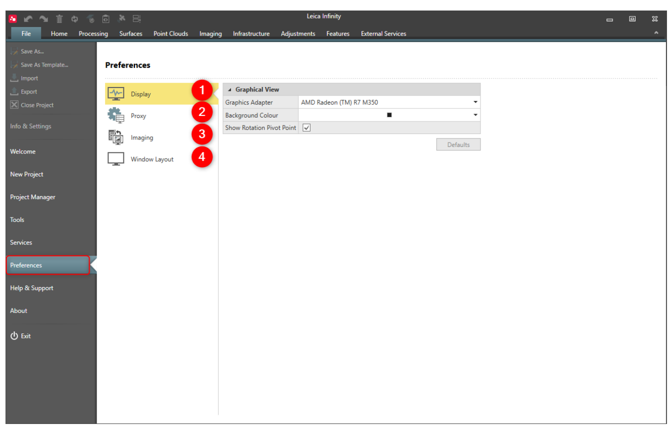
https://www.vgis.io/

2.1.9 Preferences

2.1.9.1 Overview

Preferences

Under preferences you can configure global application settings which apply to all projects.

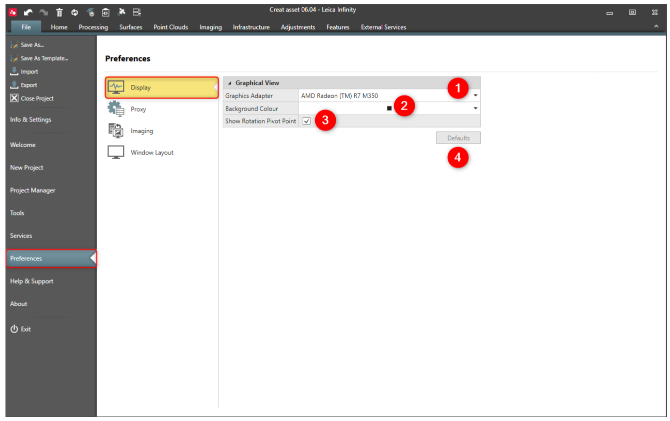


No.	Name	Description
1.	Display	Configure the graphical view.
2.	Proxy	Define the proxy server.
3.	Imaging	Define the host for imaging tasks.
4.	**Window Layout**	Reset window layout to default position.

2.1.9.2 Display

Display

Under display, you can configure the graphical view. The settings you make here apply to all projects.



No.	Name	Description
1.	**Graphics Adapter**	Change the graphics adapter if automatic selection is not suitable for you.
2.	**Background Colour**	Choose a different background colour for the graphical view.
3.	**Show Rotation Pivot-Point**	Show or hide the pivot-point in the centre of the graphical view.
4.	**Defaults**	Reset all the settings to the defaults values.

2.1.9.3 Proxy

Proxy

Under Proxy you can configure a Proxy server. Only needed if your company establishes Internet connections through a Proxy.

By default, this option is disabled.

2.1.9.4 Imaging

Imaging

Under imaging, you can configure the preferences for image rendering, image processing assets and host configuration.

The settings you configure here apply to all projects.

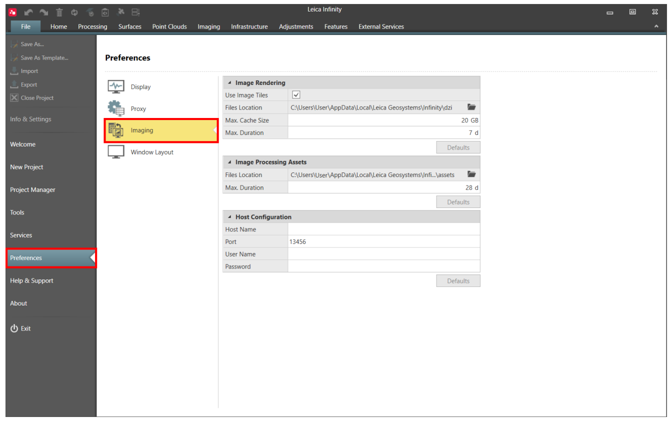


Image Rendering

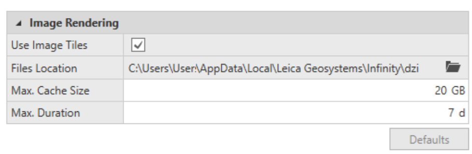
Image import creates a directory to store temporary image-related data objects for large georeferenced images.

The default path for this folder is: C:\Users\[USER]\AppData\Local\Leica Geosystems\Infinity\dzi\.

The default Max. Cache Size for this folder is 20 GB.

The default life length for this folder is seven days.

In **File** > **Preferences** > **Imaging** > **Image Rendering**, these defaults can be modified.



When Use Image Tiles is checked, temporary image-related data are created when images are imported or created (for example, when an orthophoto is created during image processing). The use of image tiles improves image rendering in the graphical view for large georeferenced images.

The Files Location is the location of the folder that has the image tiles.

Using the folder icon, a different folder can be selected. The selected folder contains the automatically generated dzi folder. Inside the dzi folder, a subfolder for each project is generated.

Consider the free disk space, when setting the Files Location.

☞ To improve the processing time, use a fast read-write disk to store the folder containing the image tiles.

The Max. Cache Size defines the max. size of the folder that has the image tiles. Older data is deleted once the Max. Cache Size is reached. This process reduces the size of files on the computer used for image rendering. If deleted, image tiles are created again when images are requested to be shown in the graphical view. Increase this number if there is enough free space in the image tiles location.

The Max. Duration is the life length of the folder that has the image tiles. When the defined threshold is passed, the image tiles are deleted. This process reduces the size of the files on the computer used for image rendering. If deleted, image tiles are created again when images are requested to be shown in the graphical view. Increase this number if there is enough free space at the image tiles location.

☞ Image tiles are anyway deleted when the project is deleted.

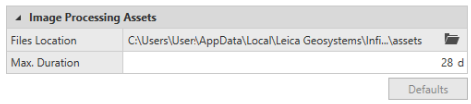
Image Processing Assets

Image-processing tasks create an asset directory to store temporary image-processing related data objects.

The default path for this folder is: C:\Users\[USER]\AppData\Local\Leica Geosystems\Infinity\mvs\assets\.

The default life length for this folder is 28 days.

In **File** > **Preferences** > **Imaging** > **Image Processing Assets**, these defaults can be modified.

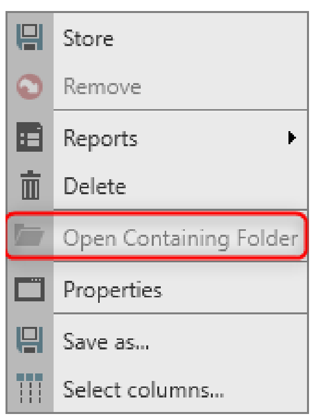


The Files Location is the location for the folder containing the Image Processing Assets. Using the folder icon, a different folder can be selected.
The selected folder has the automatically generated assets folder. Inside the assets folder, a subfolder for each project is generated.

When the path is modified, the change applies to projects created from that moment on, and to old projects where results are not yet generated. If a project already has a result from image processing: a change in the Files Location does not apply to it, even if new processes are started in this project.

For each project, all the asset folders are stored in the same parent folder.

When the assets folder directory for a project differs from the one set in Files Location, an already generated result in that project cannot be stored. The Open Containing Folder is greyed out when opening the context menu for such a result.



Once the same path is set again in the Files Location: the result can be stored again and results can be accessed using Open Containing Folder from context menu.

☞ Another option to make the old results and the old asset folder path available again, is to run a new task for each project.

If Files Location is modified while a task is running, the change is applied to the next task, as soon as the processing engine is idle.

When the local computer is connected to a host computer for processing: the Image Processing Assets is greyed out since the assets folder is created on the host computer. To modify the settings, go to **File** > **Preferences** >

Imaging > **Image Processing Assets** on the host computer and modify the Files Location.

☞ To improve the processing time, use a fast read-write disk to store the assets folder.

The Max. Duration is the life length of the folder containing the Image Processing Assets. When the defined threshold is passed, the assets are deleted.

Time is computed from the most recently created files in the assets folder. When the time passed from the creation of the most recent file is longer than the Max. Duration, the assets are deleted for that project. This process reduces the size of the files on the computer used for image processing.

Objects created like for example dense point clouds or orthophotos are still accessible in the project, but a not stored result, cannot be stored anymore.

Set a bigger value if results should be set to stored after more than 28 days. Set a smaller value if once the process is done and stored there is no need to store other results eventually present in the project.

The Max. Duration applies to all the assets folders on the computer used for processing, so a change in this value also impacts previous generated folders.

If Max. Duration is modified while a task is running, the change is applied as soon as the processing engine is idle.

When the local computer is connected to a host computer for processing: the Image Processing Assets is greyed out since the assets folder is created on the host computer. To modify the settings, go to **File** > **Preferences** > **Imaging** > **Image Processing Assets** on the host computer and modify the Max. Duration.

☞ When you delete a result from the **Inspector** > **Imaging** > **Imaging Results**, the correspondent assets folder is also deleted.

☞ The asset folder is anyway deleted when the project is deleted.

Host Configuration

Imaging tasks can be sent to a different computer (host) for processing, as long as the following requirements are met:

- Both computers have an Infinity licence with the point clouds from images option.
- Both computers have the same version of Infinity.
- The Infinity image-processing worker host is already running on the computer that is used for processing.
- Infinity is running on the computer that sends the processing task.
- The computers are working within the same network.

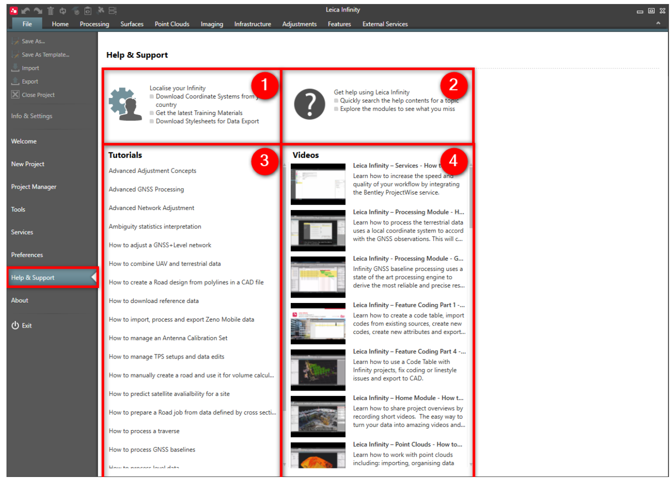
By default, the host for image-processing tasks is the local computer, that is the computer where the project is already running.

2.1.10 Help & Support

2.1.10.1 Overview

Help & Support

In help & support you get access to:



No.	Name	Description
1.	Localisation Tool	Download data predefined for your region.
2.	**Infinity Help**	Open the Infinity Help and search for a topic.
3.	**Tutorials**	Download tutorials to learn about and how to use Infinity.
4.	**Videos**	Watch videos from the Infinity YouTube channel.

See also:

Welcome

About

2.1.10.2 Localisation Tool

Localisation Tool

The localisation tool allows you to download and/or import data that are pre-defined for your region.

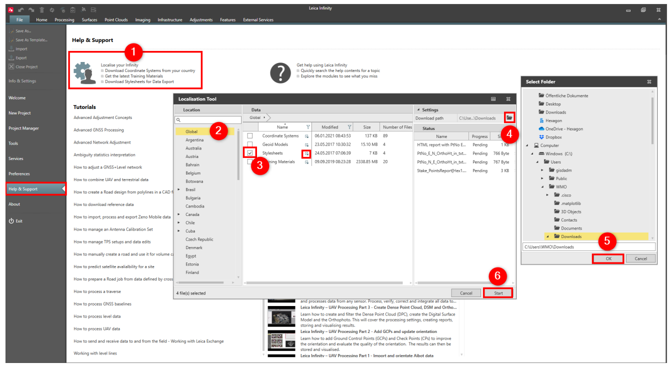
Requirements:

- Latest version of Infinity available.

The following objects are supported:

- Coordinate Systems
- Geoid Models
- Stylesheets
- Training Materials
- Documents
- ASCII Import and Export Templates
- Code Tables

To download/import data:



1.	Select **File**, then **Help & Support** and then **Localise your Infinity**.
2.	Select your Location.
3.	Select the check box of the data category you want to download/import. Drill into the category by clicking the little arrow to select single files.
4.	Select the folder icon to define a download location.
5.	The Select Folder window appears and you can change the default download path.
6.	Select **Start** to start the download/import.

Pre-defined data:

Coordinate Systems	Downloaded coordinate systems are automatically imported and available for use under Coordinate Systems. In the coordinate system manager, select **Copy to Project**, to make the selected coordinate system available for use in the currently active project.
Geoid Models	Geoid models and CSCS models can be downloaded and imported separately. After download, they are automatically available for further use in the coordinate system manager.
Stylesheets	Downloaded stylesheets are automatically imported and available to select in the settings for exporting data using stylesheets.
Training Materials	Downloaded training materials, for example tutorials, are saved in the selected download location.
Documents	Downloaded documents, for example readme files, are saved in the selected download location.
ASCII Import/Export Templates	Downloaded templates are automatically imported and available to select in Import ASCII Data or Export ASCII Data.
Code Tables	Downloaded code tables are automatically imported and available for use under Code Tables.

☞ Only data provided by your local Leica distributor is available for download.

See also:

Coordinate Systems

Training Materials

Import ASCII Data

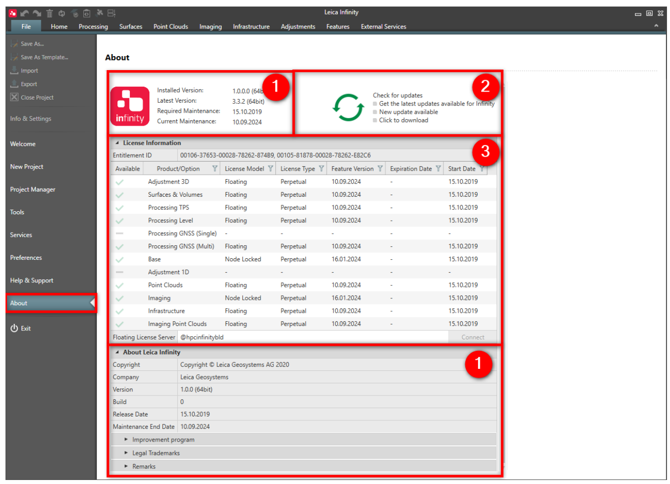
Export ASCII Data

2.1.11 About

2.1.11.1 Overview

About

In about you get information on:



No.	Name	Description
1.	**About Leica Infinity**	Check your Infinity version as well as the maintenance information.
2.	Check for Updates	Check for the latest updates available for Infinity.
3.	Licence Information	The purchased product options are listed together with your licence model.

☞ Under the Improvement program, you are asked to contribute towards the improvement of the quality and performance of Infinity. Leica Geosystems AG collects anonymous information about hardware configuration and application usage.

☞ Under Legal Trademarks, you are informed that and how Leica Infinity is protected by copyright.

☞ Under Remarks, you are informed on the legal and licensing notices on third party software components that Infinity uses.

2.1.11.2 Check for Updates

Check for Updates

Periodically, Infinity updates are made available for download and installation. These updates can include new releases with new modules or new features. Updates can also include maintenance releases with software patches.

You can check for updates from the About menu under the File tab, or have the checks made automatically on start-up. Updates can be downloaded from myWorld@leica-geosystems.com, the Leica Geosystems customer information portal.

To get the most from any new updates for Infinity, it is recommended to have a current Leica Geosystems Customer Care Package (CCP). With an active CCP, access to all software updates including new releases is available.

☞ For users who do not have administrator rights on their computer, the Check for Update message only informs of available updates. The files required for any update must be downloaded and installed by your IT department or a person with administrative rights for that computer.

2.1.11.3 Licence Information

Licence Information

The purchased product options are listed together with your licence model.

You can also see when your licence expires and your maintenance should be renewed.

If you are using a floating licence you can:

- If necessary, connect to a different licence server.
- See whether an option is momentarily available.
- See the borrowed licence options and expiration data.

2.2 Home

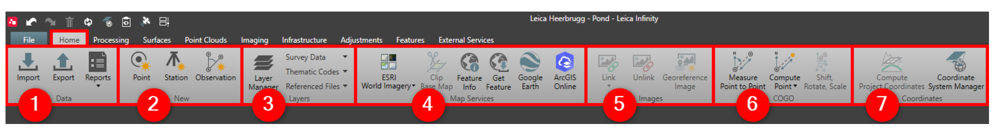
2.2.1 Overview

Home

The home tab is the core data flow working module. You can visualise, quality control, perform basic edits and generate reports from this module.

The home tab bundles some general functionality for data handling and data creation.

In the home tab you can do the following:



No.	Group Name	Name	Description
1.	**Data**	Data Import	Import all supported data formats using import dialog or drag and drop.
		Data Export	Export all or selected data to many different formats.
		Reports	Create detailed reports on your project data.
2.	**New**	New Point	Create new points.
		New Station	Create new stations.
		New Observation	Create new observations.
3.	**Layers**	Layer Manager	Open layer manager and set the colour, style and export settings for the library data.

No.	Group Name	Name	Description
		Survey Data	Explore the imported survey data: Switch them on/off or lock/unlock them.
		Thematic Codes	All feature codes are organised in layers: Switch them on/off or lock/unlock them.
		Referenced Files	All the layers as defined in referenced files: Switch them on/off or lock/unlock them.
4.	**Map Services**	Base Map	Turn on or off one of the available base maps.
		Clip Base Map	Clip a base map as a georeference image stored to the project.
		Feature Info	Get information about objects at the mouse position.
		Get Feature	Download data from a Web Feature Service (WFS).
		Google Earth	Visualise the project data in Google Earth.
5.	**Images**	Link Images	Select an image and link it to a point, line or area.
		Unlink Images	Remove an image linked to a feature.
		Georeference Images	Select an image to ortho rectify.
6.	**COGO**	Measure Point to Point	Measure point to point.
		Compute Point	Compute point.
		Shift, Rotate, Scale	Shift, rotate, scale.
7.	**Coordinates**	Compute Project Coordinates	Compute project coordinates.
		Coordinate System Manager	Open the coordinate system manager.
		Transform Local Grid to Local Grid	Transform local grid coordinates from coordinate system A to another local grid coordinate system B.

2.2.2 Data Import

2.2.2.1 Overview

Data Import

Data can be imported to the current project from different sources. Based on the type of data you can define and apply specific settings and filters. The import functionality is available either from inside the **File** tab or from inside the **Home** tab.

☞ It is only possible to import data into a project if the project is open in the application. Check if the Import menu is active. If not then open a project first either by creating a new project or by opening an existing project from inside the **File** > **Project Manager**.

☞ Imported data files are listed in the **Navigator** under **Source**. Select any file to view its properties in the property grid.

To import data:

1.	Go to the **File** tab and select **Import** from the menu on the left side or go to the **Home** tab and select **Import** from the ribbon bar.
In the **Import** dialog:	
2.	Select the **Type** of the source file to be imported.
3.	Browse to the location where the source file is stored. All source files of the selected file type that are available in the selected location are listed in the content area of the dialog under Files. ☞ Select the **Show subfolders** option to list all files that are of the selected type, but contained in subfolders.
4.	Select the **Source files** to be imported. ☞ Only files of the same type can be multi-selected. ☞ Selecting a file and pressing Ctrl+A, selects all files that are of the same type.
5.	Select the **Import** option.

To import data by drag and drop:

1.	On your computing device: Open the file explorer and browse to the location where the file is stored.
2.	In the Infinity software: Select the **Home** tab in the ribbon bar and activate **View** within an open project.
3.	Go to the file explorer and drag and drop the file into the **View** area to import the file.

Import Formats

ASCII	A **template** must to be defined. See also Import ASCII Data.
SKI ASCII	SKI ASCII files are written in a specific ASCII format to exchange information on baseline vectors. See also Importing SKI ASCII Files.
Captivate/Smart-Worx DBX	Imports all data belonging to a job measured in the field. Points that were deleted in the field are by default also imported. Under **Settings**, de-select this option if you do not want deleted points to be imported. ☞ Points that were deleted in the field can be recovered in the **Navigator** or in the **Inspector**.

GNSS Raw Data	*.MDB only - Imports only the raw GNSS data from Captivate/SmartWorx jobs. RINEX 2.10, 2.11, 3.02 and 3.04 supported. Hatanaka supported (for example, RINEX observations files in a compressed format preferably used when observations are downloaded online from permanent reference station sites).
Level Data	Import *.LEV format level data from the DNA levels.
Coordinate Systems	Coordinate systems can be imported from: *.dat files from Leica. *.lok files from iCON and SBG Geo. *.dc, *.cal and *.jxl coordinate system files from Trimble.
XML	Supported are LandXML and HeXML files.
Scan Data	Point Clouds can be imported from: *.pts or *.ptx, both of which are ASCII based formats. *.xyz. *.sdb, the Leica specific format. *.las/ *.laz. ☞ Importing to *.las/ *.laz: Data has to be in the local grid coordinate system. ☞ Importing to *.las/ *.laz: If the data includes classification, a point cloud group is created. Each classification is written as a point cloud that belongs to the point cloud group.
Images - JPG, PNG, TIFF	Imports images from *.JPEG, *.PNG and *.TIFF.
Georeferenced Images	Imports georeferenced images from *.JPEG, *.PNG and *.TIFF with respective world files in *.JGW,*.PGW,*.TFW.
CAD	Imports CAD data from *.dxf, *.dwg and *.dgn.
ICM	Import the Bentley ICM file with the alignment and surface.
BIM	Import building information modelling data from *.ifc or *.ifczip files. Data structure (structure, types, layers) is preserved.
ESRI Shape files	Imports points, lines and areas information from ESRI Shape files (*.shp). ☞ The attribute information from the (*.dbf) is imported as well and is listed in the Attribute table available from the Data Source context menu.
ESRI Geodatabase	Imports points, lines, areas and codes definition from ESRI file geodatabase (.gdb) and mobile geodatabase (.geodatabase). ☞ Data can be imported from .zip files.
GeoJSON	Imports points, lines and areas information from GeoJSON GIS file format (*.geojson). ☞ The attribute information is imported as well and is listed in the Attribute table available from the Data Source context menu.

GEO Viewer - KML, KMZ	Import points, lines and areas information from *.KML, *.KMZ. ☞ The attribute information is imported as well and listed in properties.
Zeno Mobile	Import Zeno mobile projects as zip files. The zip file has shape files, GNSS observations, linked images and defined coordinate systems.
Observation Data GSI	Import *.GSI format for level data, points and TPS observations. **For TPS observations:** From the **Import Settings** table, you can select how the feature codes shall be processed during the import. In the **Interpret Features** field, you can choose if feature coding is interpreted as **Thematic**, **Free** or **None**. **Thematic**: A point-related code is created. Decide on which point or points the code shall be related to. **Free**: The result is a time stamp code. Time stamp codes are not point-related. **None**: Codes are ignored. Points are created without codes. Thematic codes may belong to either previous or following points. In the**Apply Code to** field you can decide on which points the feature code shall be applied to: **Previous point** or **Following point**. **All previous points until different code** or **All following points until different code** is encountered while reading the raw data file. **All previous and following points** is encountered while reading the raw data file. ☞ Codes registered as WI 71 are imported as thematical codes independent of the coding import setting.
Aibot Data	The Aibot data include image, GNSS and sensor information.
LGO Project/CSYS	Import LGO projects or LGO global coordinate systems. The coordinate systems and georeferenced images attached to a project are imported when the path to the database ODB folder is defined. To import all global coordinate systems navigate to the CSysDb dbd file, default path: C:\ProgramData\Leica Geosystems\LGO\ODB\Fixed\CSysDb\. For more information, see Import LGO Project.
ZIP	Various compressed file formats (*.zip) are supported and show in the import dialog, if import is possible. Infinity attempts to preview the file contents to determine the correct import format.

Navigation features in the Import dialog

Two special folders support better navigation between locations:

- Under **Recent**, the last used locations are listed.
- Under **Favourites**, the locations are listed that you marked as a favourite by selecting the **Set/Reset favourite folder** option.

  Favourites can be removed from the list by selecting **Remove** from the context menu.

To **Refresh** the navigation pane, **Rename** or **Delete** a folder or to create a **New Directory**:

Right-click onto the folder and select the desired function from the context menu.

It is also possible to copy and paste a location. Double-click the breadcrumbs in the files pane to make the field become editable and copy the desired path to it.

2.2.2.2 Import ASCII Data

Import ASCII Data

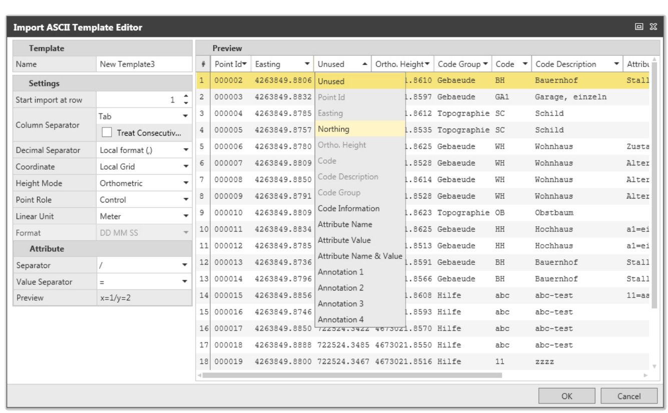
To import ASCII files into your project you have to use the ASCII import. A template must be defined to identify user-specific data structures which are typical of ASCII files.

☞ Without a suitable and correctly defined template, it is not possible to read the data structure in the ASCII file nor to import the data.

To import ASCII data:

1.	Go to the **file** tab and select **Import** from the menu on the left side or go to the **Home** tab and select **Import** from the ribbon bar.
In the **Import** dialog:	
2.	Select **ASCII** as the type of source file to be imported.
3.	Browse to the file to be imported and select the **file**.
4.	Under ASCII Settings select an **existing Template** that fits the file format or select **New...** to define a new one. Confirm a newly defined template with **OK**.
5.	Select the **Import** option.

Defining an ASCII Import template:



Settings:

- Define at which row the import shall be started.
  If the ASCII files have a header in row 1 then start the import at row 2 (for example, at the row that has the first data set).
- Identify the Column Separator.
  Select **Treat Consecutive Delimiters as One** if necessary. See the following for further information.
- Identify the Decimal Separators that are used in the ASCII file.
- Identify the correct Coordinate Format and Height Mode of the data to be imported.
- Select the **Point Role** that shall be assigned to the imported points.
- Identify the Linear Unit.
- If you intend to import geodetic coordinates identify the Coordinate Format.
  For more information, refer to:

Attributes:

- Identify the Attribute and Value Separators that are used in the ASCII file.
- A preview shows you which structure is expected based on your settings.

☞ Preview and ASCII file must correspond to each other.

Preview:

- Identify the data type in each column by selecting a type from the drop-down menus.
  To identify the column point ID is mandatory. All other data types can optionally be set.

☞ Columns of type Unused are ignored during import.

☞ For the height, only the previously selected mode is available for selection.

☞ If a column only has attribute values (without the attribute name) then select **Attribute Value** from the drop-down menu.
If the name is given in the file header then it is taken from there.
I
If the header is empty a default name is assigned.

Treat consecutive delimiters as one

It may happen that your ASCII file looks like this:

```
1  309 → → 549282.26 → 5248890.18 → 413.07
2  440315 → 548314.88 → 5247659.36 → 419.72
3  440402 → 550691.54 → 5247142.54 → 418.67
```

Using two tabs following each other in the first row but only one as separator in the following rows. ASCII Import would interpret this as:

Preview

	Unused	Unused	Unused	Unused	Unused
1	309		549282.26	5248890.18	413.07
2	440315	548314.88	5247659.36	419.72	
3	440402	550691.54	5247142.54	418.67	

Making it impossible to identify the columns.

Select **Treat consecutive delimiters as one** to achieve a correct interpretation of the columns.

The resulting columns in the Preview are inline and you are able to identify each by its header.

Geodetic Coordinate Formats

If you intend to import geodetic coordinates from a text file the coordinates have to be in one of the following formats:

DD.DDDD

```
1        47.156894112 N          9.249561234 W   474.7037
2        47.156894112 S          9.249561234 E   474.7037
3        47.156894112 N          9.249561234 E   474.7037
4        47.156894112 S          9.249561234 W   474.7037
```

Or

```
1        47.156894112            -9.249561234    474.7037
2        -47.156894112           9.249561234     474.7037
3        47.156894112            9.249561234     474.7037
4        -47.156894112           -9.249561234    474.7037
```

With the "-" sign indicating West and South.

DD MM SS

```
1        47 24 31.72146 N        9 37 05.02956 W 474.7037
2        47 24 31.72146 S        9 37 05.02956 E 474.7037
3        47 24 31.72146 N        9 37 05.02956 E 474.7037
4        47 24 31.72146 S        9 37 05.02956 W 474.7037
```

Or

```
1        47 24 31.72146          -9 37 05.02956  474.7037
2        -47 24 31.72146         9 37 05.02956   474.7037
3        47 24 31.72146          9 37 05.02956   474.7037
4        -47 24 31.72146         -9 37 05.02956  474.7037
```

With the "-" sign indicating West and South.

DD.MMSS

```
1        47.243172146 N          9.370502956 W   474.7037
2        47.243172146 S          9.370502956 E   474.7037
3        47.243172146 N          9.370502956 E   474.7037
4        47.243172146 S          9.370502956 W   474.7037
```

Or

1	47.243172146	-9.370502956	474.7037
2	-47.243172146	9.370502956	474.7037
3	47.243172146	9.370502956	474.7037
4	-47.243172146	-9.370502956	474.7037

With the "-" sign indicating West and South.

2.2.2.3 Importing SKI ASCII Files

Importing SKI ASCII Files

When you want to import post-processed GNSS baselines as observations into Infinity, for example as input to an adjustments computation, then you can do so though the SKI ASCII Import.

SKI ASCII files are written in a specific ASCII format to exchange information on baseline vectors.

In the import dialog:

1. Select **SKI ASCII files (*.asc)** as file type.
2. Browse to the location where the SKI ASCII file is stored and select the file to be imported.
3. Select the **Import** option.

In the Infinity project:

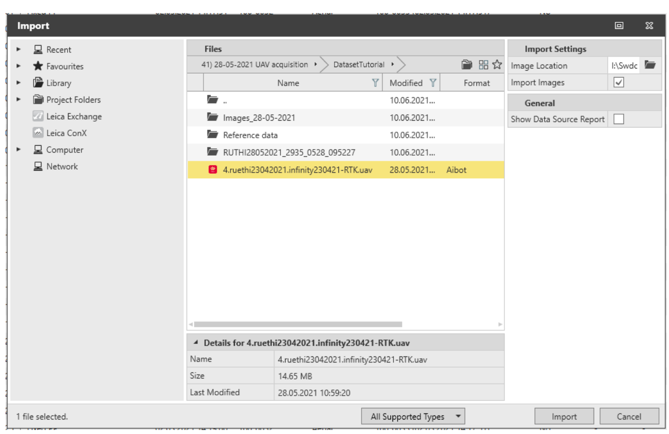
4. Select a **coordinate system** to be used for displaying the imported GNSS observations.

SKI ASCII files that have baseline vector information always hold this information in WGS84. But Leica Infinity only displays data that is either stored as local grid or can be converted to local grid. Thus a coordinate system is required to convert the GNSS observations to local grid and make them visible in the graphical view. On how to assign a coordinate system to a project, see Coordinate Systems.

2.2.2.4 Import Aibot Data

Importing Aibot Data

To import Aibot data:



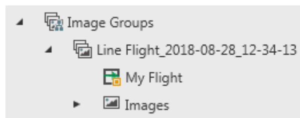
1. Select the .uav file in the import dialog.
2. Specify the exact path where the images can be found at Image Location.

3. Select whether to import images or not. If the **Import Images** checkbox is not selected, images are linked to the project from the import folder.
4. Select the **Import** option.

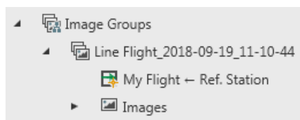
Once the Aibot file is imported, a points group, an image group and a GNSS tracks group are automatically created. The image group holds the flight object and the images acquired during the flight.

Depending on the type of Aibot licence, at import you can have:

- Autonomous flight (no GNSS correction received during the flight).



- RTK flight (GNSS correction are received during the flight to get very accurate image positions).



The default name of the image group is the name of the flight.

☞ The position of the images is updated automatically with respect to the flight track. No additional synchronisation is required.
If the position of the RTK base station is updated either by manually updating the station coordinates or by processing it from another reference station, then the flight track is automatically updated. Therefore also the position of the images is updated.
If GNSS raw data have been logged, they can be post-processed in Infinity using a reference station. Once the processing results are stored, the flight track is automatically updated. Therefore also the position of the images is updated.

Information about the flight can be found in the flight report as well as the data source report.

To create a flight report:

1. Select the image group that contains the Aibot data.
2. Select **Flight Report** from **Reports** in the Home ribbon bar or use the context menu.

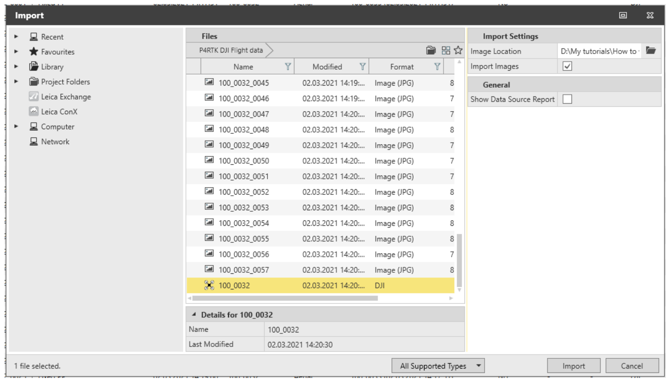
The flight report includes information about:

- The flight: Start and end time and number of events.
- The sensor: Model, serial number and firmware version.
- The GNSS track: Start and end time, length, receiver name and serial number, sampling rate and so on.
- The GNSS intervals and the reference station (if any).
- The image group: Number of images, size, sensor and camera model.
- The interior orientation of the camera.

The image data from Aibot can be processed using the Imaging Point Clouds module.

Import DJI GNSS Flight Data

To import DJI flight data:

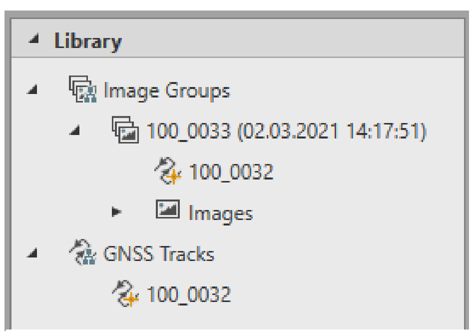


1.	Select the .dji file in the import dialog.
2.	Specify the exact path where the images can be found at Image Location. Default is the current folder.
3.	Select whether to import images or not. If the **Import Images** checkbox is not selected, images are linked to the project from the import folder.
4.	Select **Import**.

Once the DJI file is imported, an image group is created in the library; it has the images and the track.

A GNSS tracks node with the imported track is also created.

The default name of the image group is the name of the flight. It can be modified at any time.



☞ When the images are not located in the same folder as the Flight Data, the image location can be set accordingly using the import dialog.
If multiple DJI flights are copied in the same folder, they can be imported together selecting them in the import dialog at the same time.
If multiple DJI flights are in the same folder, and only one is needed, only the images connected to the selected DJI file are imported.
You can drag and drop data into the application using the events .MRK file.
If the GNSS data is not of interest, only the images are imported by selecting them instead of the DJI file.
The GNSS track can be post-processed in Infinity using a reference station. Once the processing results are stored, the positions of the images are automatically updated, and can be used for a standard image-processing workflow.

To create a flight report:

1. Select the image group that has the DJI data.
2. Select **Flight Report** from **Reports** in the Home ribbon bar or use the context menu.

The flight report includes information about:

- The flight: Start and end time and number of events.
- The sensor: Model, serial number and firmware version.
- The GNSS track: Start and end time, length, receiver name and serial number, sampling rate and so on.
- The GNSS intervals and the reference station (if any).
- The image group: Number of images, size, sensor and camera model.
- The interior orientation of the camera.

The image data from DJI can be processed using the Imaging Point Clouds module.

See also:

The tutorial **"How to work with DJI drones - GNSS and image processing workflow"**

☞ The tutorial can be downloaded in the Localisation Tool.

2.2.2.6 Import LGO Project

Import LGO Project

With this function, it is possible to import data from LGO projects into Infinity.

To import an LGO Project:

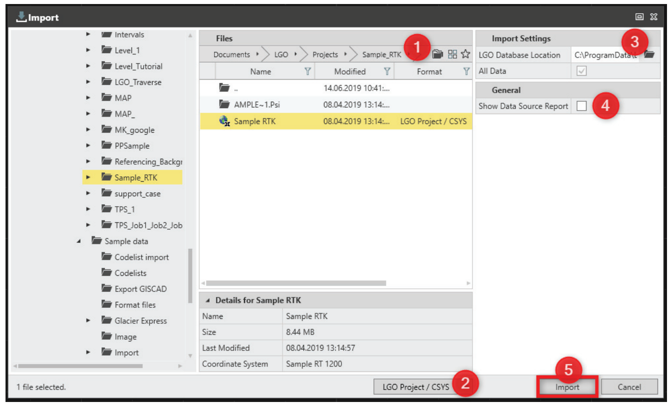
Drag and drop:

1. Navigate to the LGO project folder in Windows File Explorer or similar.
2. Select, then drag the LGO project folder and drop it into Infinity.

☞ Default import settings are used.

Or

Import dialog:



1.	Navigate to LGO project folder and select the project to import data from.
2.	Choose the data format for import to LGO Project/CSYS to filter the import view.
3.	Specify the path of the LGO database (ODB). ☞ If LGO is installed on the computer the path is recognised automatically. If LGO is not installed, copy the ODB directory from another computer and the path to it is defined here.
4.	Select whether to **Show Data Source Report**.
5.	Select **Import**.

The list of supported LGO objects and the data that can be imported is explained in the following table:

Object	What is supported or not supported
Features	**Supported:** Points, lines, areas with codes and assigned Images. **Not supported:** Shading styles for areas.
Coordinate System CSYS	**Supported:** Coordinate systems from the CSYS manager or a project. **Not supported:** If the coordinate system has a geoid.exe file or a stepwise/interpolation transformation, the CSYS is skipped on import.
TPS	**Supported:** Stations, observations, setups, traverse, Sets of Angles. ☞ Set orientation with known azimuth option from LGO is transferred to known backsight method.
GNSS	**Supported:** Stations, observations with QC values, tracks and interval events, antennas. **Not supported:** Raw data is not imported. ☞ Use the RINEX format to export from LGO and import to Infinity if reprocessing is required.
GNSS Hidden Points	**Supported:** Offset information.

Object	What is supported or not supported
Level	**Supported:** Observations and level lines. ☞ To view misclosure and height errors, reprocess the level line.
Codelists	**Supported:** Project codelist imported to Infinity as field codes. **Not supported:** Global codelists.
Images	**Supported:** Images linked to points, lines, areas, TPS stations as well as not linked images.
Georeferenced Images	**Supported:** If LGO is installed on the same computer as Infinity and the path to the database folder (ODB) is defined.
Free Codes	**Supported:** LGO free codes are imported as with the option Apply leap seconds set in LGO.
Surfaces	**Supported:** Surfaces with boundaries and breaklines. ☞ If the surface vertices are stored in WGS84, the created surface is not editable. **Not supported:** Importing contours.
DXF Background Maps	**Not supported:** Imported separately using the DXF/DWG import.

2.2.3 Data Export

2.2.3.1 Overview

Data Export

From your current project either all data can be exported or just a selection of objects or sources. You can choose from different file formats. When you export a selection make sure that the selected object fits to and can be exported in the chosen format.

☞ Exported files are listed in the **Navigator** under **Archive**. Select any file to view its properties in the property grid.

To export data:

1.	Select the **Export** option in the ribbon bar. If elements are selected either **graphically** or in the **Navigator** or in the **Inspector** choose between exporting **All** or **Selection** from the drop-down menu.
2.	In the **Export** dialog: Select the **Destination** where the exported data shall be stored to.
3.	Give the file to be exported a **Name** without extension. The file extension is given automatically according to the selected file type.
4.	Select the **Type** of the file to be exported.
5.	Choose whether a **subdirectory** shall be **created**. If you select the checkbox a subdirectory of the same name as the file is created and the file is written to it.
6.	Select the **Export** option.

Export Formats

ASCII	A **template** must be defined. See also: Export ASCII Data
SmartWorx DBX	You can export points and/or lines and areas and/or surfaces as DTM and/or alignments and/or infrastructures and/or GEM files. For points, select the **Roles** to be exported. Choose between: **Highest:** To export only the highest role for each point. **All:** To export all available roles for each point. **Selection:** To export only the selected roles. For lines/areas, the points that define the line/area are automatically exported, too. For alignments, you can select the export **To Road Job** , **To Rail Job** or **To Tunnel Job**. ☞ If you export a selection, make sure that you select the correct object type under **Settings** to avoid empty files to be written. Select the checkbox to **Create .zip file**. ☞ The suffix _dbx is added to the end of the file name for example " [file name]_dbx.zip". ☞ This option is not available when using Leica Exchange service.
ESRI - SHP	You can export points and/or lines and areas. For points and for lines and areas, you can select various **Additional Attributes**. You can select the **Dimension**: **2D** or **3D**. You can select if the images should be exported as **Individual** or **Combined**. **Combined** image files are exported as a jpg and html. If the images checkbox is selected in Additional Attributes, only one attribute with multiple images is created. **Individual** image files are exported as a jpg. If the Images checkbox is selected in Additional Attributes, a separate attribute is created per image.
ESRI File Geodatabase - GDB	Export project data with thematic information as an ESRI GDB File Geodatabase. points, lines and areas are exported with their assigned code and code attributes. You can select the **Dimension**: **2D** or **3D**. ☞ All data is written with WGS84.
Coordinate Systems	All coordinate systems available with the current project are exported and saved to the same TRFSET.DAT file or separate iCON/SBG Geo LOK, Trimble JXL files. ☞ If you export to DBX, LandXML or HeXML always the master coordinate system is exported with your data. If **None** is used then your data is exported without any coordinate system information.

XML	Supported are HeXML and LandXML. You have the option to use an XSL Stylesheet to format the output. You can use the same stylesheets as available with SmartWorx Viva or define your own.
KML/KMZ	All library objects are exported except for cloud points.
Images	Images are exported in JPG, PNG and TIFF format.
Georeferenced Images	Georeferenced images can be exported in JPG, PNG and TIFF format with the respective world file (JGW, PGW, TFW), and in geoTIFF format. JPG and PNG format can be used by onboard software.
Georeferenced DEM - TIFF, geoTIFF	Surfaces are exported as raster in TIFF (with the respective TFW world file) and geoTIFF format.
GNSS raw data - RINEX	You can select the following **Export Settings**: **RINEX version:** Select whether you want to save files as RINEX files (version 2.11) or as RINEX files (version 3.04). **Apply Interval Windows:** Select the checkbox if you want to apply the interval windows to the exported RINEX files. **Separate files for different tracks:** Select the checkbox if you want to write a separate file for each track. ☞ The files are named according to the point ID, day of year and session number. **Satellite Systems**: Select the GNSS constellation to be exported.
Point Cloud	Point Clouds can be exported to: *.e57. *.las/*.laz. *.ply. *.pts, which is an ASCII based format. *.lgs ☞ When exporting to *.las/*.laz note that Infinity does not support any classification of points. ☞ When exporting to *.lgs, a valid Cyclone Publisher or Cyclone Publisher Pro (from Infinity 3.4 on) licence is required. ☞ When exporting in .lgs format, point cloud includes: MSxx/TLS data - Point cloud arrived from instruments and panoramas. UAV/GS18 I data - Dense point cloud and images which were used for point cloud reconstruction.
AutoCAD	You can export your project data as well as BIM entities to *.dxf or *.dwg. There is no necessity to first create library objects from BIM entities.
Aibotix AiProFlight	Kinematic tracks can be exported for use onboard Aibotix AiProFlight devices. See also: Export UAV Track and Import to AiProFlight Software

GeoMoS Now	For points, select the **Point Roles** to be exported. Choose between: **Highest:** To export only the highest role for each point. **All:** To export all available roles for each point. **Selection:** To export only the selected roles.
Bentley - FWD	Project data can be exported to the Bentley FWD format.

Navigation features in the export dialog

Two special folders support better navigation between locations:

- Under **Recent**, the last used locations are listed.
- Under **Favourites**, the locations are listed that you marked as a favourite by pressing the **Set/Reset favourite folder** option.

  Favourites can be removed from the list by selecting **Remove** from the context menu.

To **Refresh** the navigation pane, **Rename** or **Delete** a folder or to create a **New Directory:**

Right-click onto the folder and select the desired function from the context menu.

It is also possible to copy & paste a location. Double-click onto the bread-crumbs in the files pane to make the field become editable and copy the desired path to it.

2.2.3.2 Export ASCII Data

Export ASCII Data

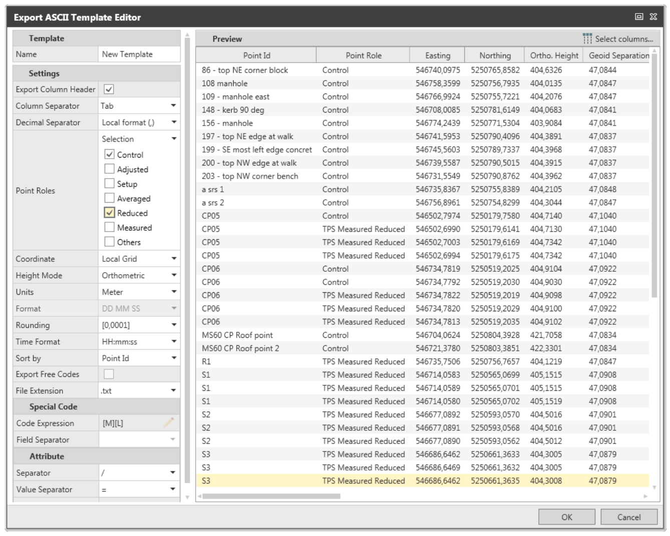
The ASCII export allows you to extract point information from your project to an ASCII file. The export format must be defined in a template. In the template you specify and store information like:

- A template name.
- ASCII specific settings.
- File content settings.

To export data to an ASCII file:

1.	Go to the **File** tab and select **Export** from the menu on the left side or go to the **Home** tab and select **Export** from the ribbon bar.
In the export dialog:	
2.	Select ASCII as the **Type** of source file to be exported.
3.	Browse to a location where the exported file shall be written to (for example, to the drive, folder).
4.	Give the file a **Name**. You can choose a file extension from the drop-down list or give the name any file extension you want.
5.	Choose whether a **subdirectory** shall be **created**. If you tick the checkbox a subdirectory of the same name as the file is created and the file is written to it.
6.	Under **Settings** select an existing **Template** that fits the file format or select **New...** to define a new one. Confirm a newly defined template with **OK**.
7.	Select the **Export** option.

Defining a new ASCII Export template



Settings:

- Select to **Export Column Headers** if desired.
- Select the **Column Separator** to be used in the exported ASCII file.
- Select the **Point Roles** to be exported.
  Choose between **Highest** to export only the highest role for each point, **All** to export all available roles for each point or **Selection** to export only the selected roles.
- Select the type of Coordinates to be exported.

☞ Only available coordinates or coordinates that can be derived from a transformation calculation are exported.

- Select the **Height Mode** to be exported.

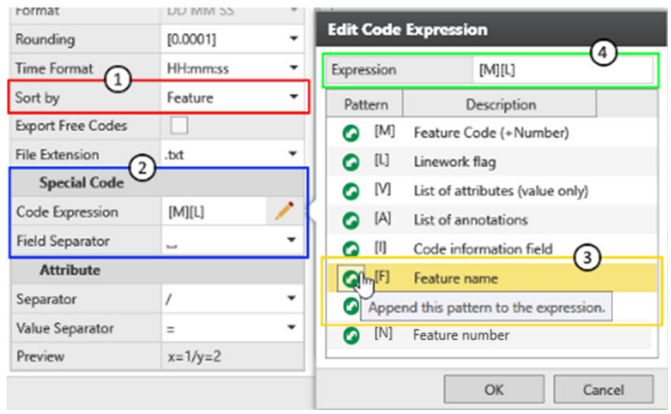
☞ Only the available height mode is exported or the geoid separations must be known.

- Set the **Units** for exported distances.
- If you intend to export geodetic coordinates, identify the coordinate format.
  For more information, refer to:
- Choose the precision for Rounding.
- Select whether the exported data shall be sorted by **Point ID**, **Date/ Time** or **Feature**.
  Sort by Feature: It is possible to export ASCII data sorted by Feature to better support moving field collected data to third party software.
- If you select **Sort by Date/Time**, you can activate **Export Free Codes** by selecting the checkbox.
  Free codes are purely time-related information. They are recorded in between of measurements in the field. A time stamp is recorded with each free code allowing to export points and free codes in a chronological order to be used for third-party mapping software.
- Select the **File Extension** to specify the file type of the export file.

Special Code:

Define the output for the **Code Expression**, that sets how the code, attribute, code information and feature name is written in the ASCII export, including the Field Separator.

To define the code expression, do the following:



1.	In the **Export ASCII Template Editor**, select **Sort by Feature**.
2.	In the **Code Expression** field, select the **Edit** icon to open the **Edit Code Expression** dialog.
3.	In the **Edit Code Expression** dialog, define the **Code Expression** by selecting or entering the patterns to be part of the export.

☞ You can write special characters in between patterns, for example [C]/[N]*[L]*[V1], then they get exported in all cases.
You can write one special character before or after the pattern, then the special character only gets exported, if the pattern itself exists, for example [C][/N][*L][*V1].

Attributes:

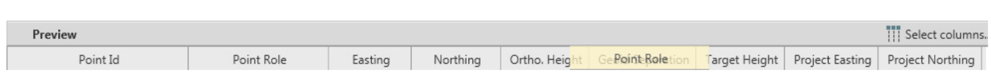
- Select the Attribute and the Value Separator that shall be used in the ASCII file.
- A preview shows you how the attribute columns are structured in the ASCII file based on your settings.

Preview:

- The Preview shows you how the data is written to the ASCII file. Select the columns to be exported. Unchecked columns are not exported.

☞ Only data that is available according to the given ASCII settings are available for selection.

- To change the sequence of columns in the ASCII file, shift them (by clicking and dragging them) in the Preview.



Geodetic Coordinate Formats

If you intend, to export geodetic coordinates to a text file, the coordinates can be exported in one of the following formats:

DD.DDDD.

```
1       47.156894112           -9.249561234    474.7037
2       -47.156894112          9.249561234     474.7037
3       47.156894112           9.249561234     474.7037
4       -47.156894112          -9.249561234    474.7037
```

With the "-" sign indicating West and South.

DD MM SS.

```
1       47 24 31.72146 N        9 37 05.02956 W 474.7037
2       47 24 31.72146 S        9 37 05.02956 E 474.7037
3       47 24 31.72146 N        9 37 05.02956 E 474.7037
4       47 24 31.72146 S        9 37 05.02956 W 474.7037
```

With the "-" sign indicating West and South.

DD.MMSS.

```
1       47.243172146           -9.370502956    474.7037
2       -47.243172146          9.370502956     474.7037
3       47.243172146           9.370502956     474.7037
4       -47.243172146          -9.370502956    474.7037
```

With the "-" sign indicating West and South.

2.2.3.3 Export UAV Track and Import to AiProFlight Software

Export UAV Track and Import to AiProFlight Software

UAV data coming from the Aibot can be processed by Infinity.

Process the track and then export an updated coordinate file that is used to update the geotags of the images in the AiProFlight software.

In Infinity:

1. Select the **track** from the graphical view.
2. Select **Export to Aibotix AiProFlight** from the context menu.

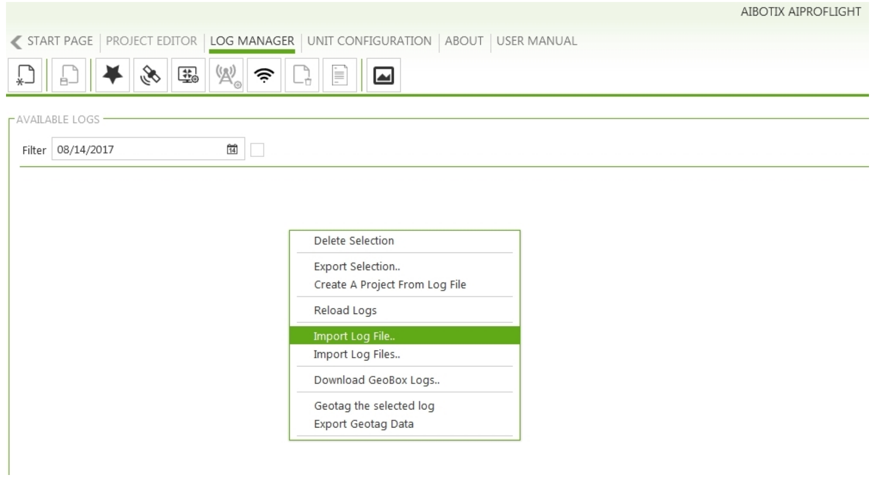
3. In the Export dialog, enter all the necessary information.
4. Select **Export** to create a txt file which can be used in AiProFlight.

In AiProFLight:

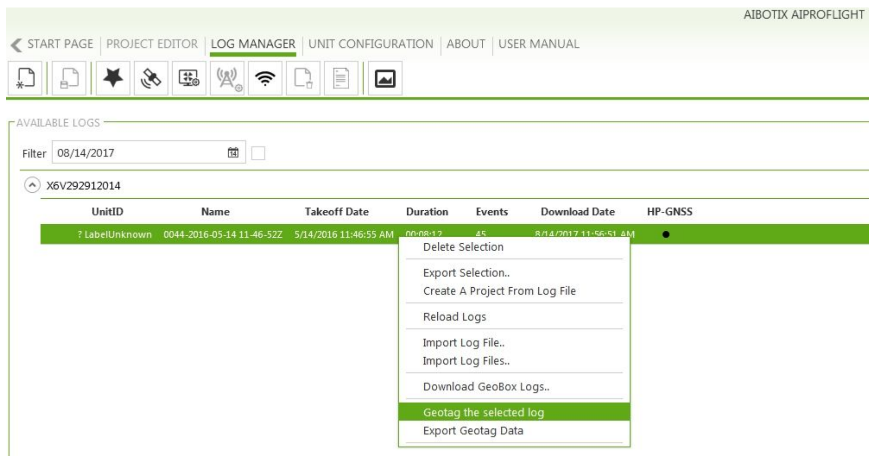
1. Go to the **Log Manager** tab.

2.	Right-click on the white tab and select **Import Log File**.	

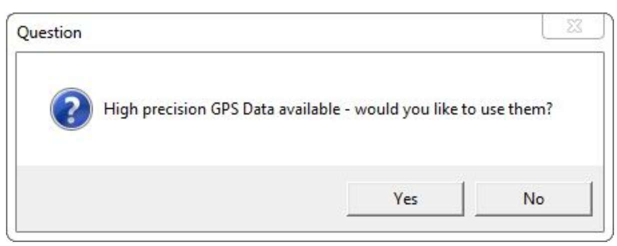
3.	Select your log file. The log file is provided to you together with the flight data and the photos.



4. Select the imported **Log File**. Right-click and select **Geotag the selected log**.

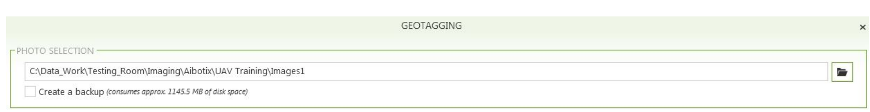


5. A question might pop up in case the log file contains high precision GPS data:

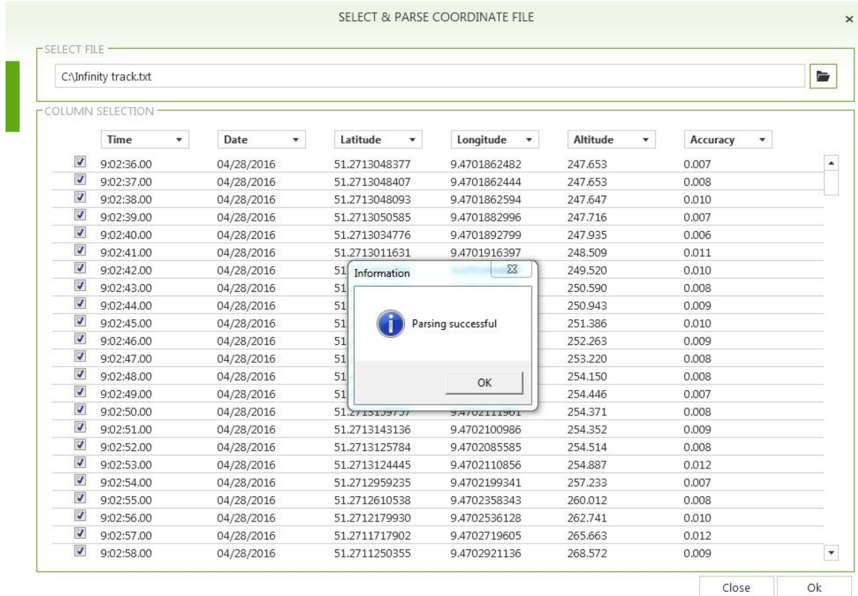


6. Select **Yes** to continue using high precision GPS data.

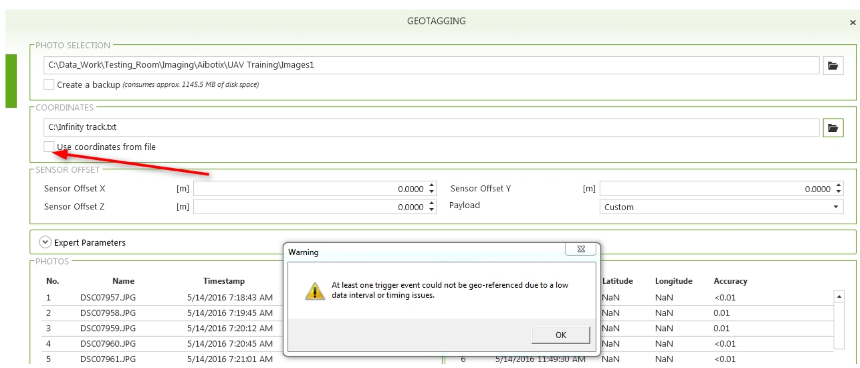
7. In the **Photo Selection** tab, select the folder that contains the photos of the specific flight. Import your photos.



8. In the **Coordinates** tab, import the output file from Infinity.



9. Once the file is imported, activate **Use Coordinates from file**.



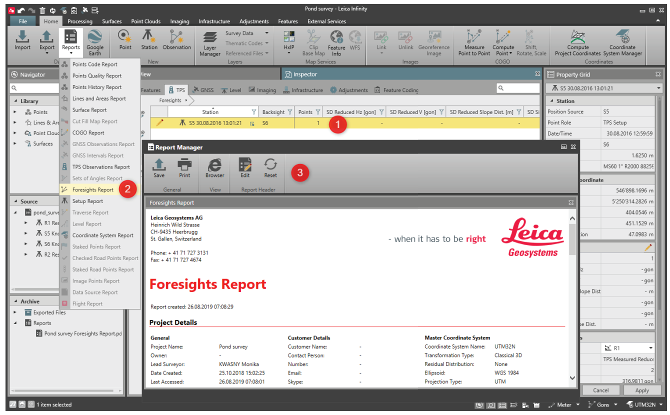
10. On how to update the image geotags and the steps that are necessary to do so, refer to the Aibot Help.

2.2.4 Reports

Reports

An important part of any project is the documentation of the data import, field work and project processing results. Run and store the reports and be aware of all the reports from the navigator archive.

To run a report:



1. Select the objects that you want to have the report of. You can also select the results of processing or adjustment runs or of single volume calculations.

   ☞ In some cases, you can run reports on all the data of the project, such as points, TPS setups, TPS and GNSS observations.

2. Select the **Foresights Report** option in the **Home** tab or select **Report** from the context menu.

3. Report opens in Report Manager.

   ☞ Values exceeding limits are marked in bold red. Go to **File** > **Info & Settings** to set the tolerances as required.

For each report, the following operations are possible:

Icon	Description
	Save the report as a PDF or HTML file.
	Print the report.
	Open the report in an Internet Browser.
	Add an image/HTML file as a Report Header.
	Reset the Report Header to the Leica default.

Saved reports are seen in the **Project Archive** in the **Navigator**.

For a selected report from within the context menu, the following operations are possible:

Icon	Description
	Open the report.
	Open the report in containing folder.
	Remove the report.

Icon	Description
	Open properties.

Reports on Data Sources

After importing a Captivate/SmartWorx DBX job, it is possible to run a report to document the imported state of the data source. In some cases editing of the imported data is necessary to fix various field errors or to shift the data from local to grid coordinates. You can run again the Data Source Report to store the edited state of the data source.

1.	Run report after import to document the field data as measured.
2.	After edits have been made run a second report to document the current or final state of the field data.

2.2.5 Map Services

2.2.5.1 Base Map

Base Map

It is possible to use a base map in your project. The base map function is a nice way to help visualise, reference and relate your project data.

We distinguish between predefined and user-defined base maps.

The following predefined base maps are available:

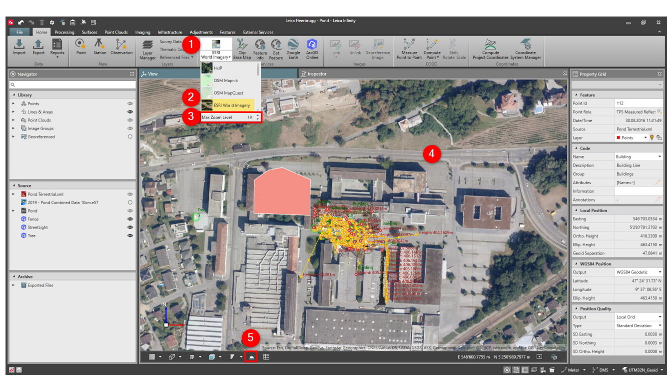
- HxIP (Hexagon Imagery Program).
- OSM (OpenStreetMap).
- ESRI.
- NGS Topo.
- A coordinate system defined in the project.
- At least one point in the local grid value available in the project.
- Internet access to stream the image tiles or maps.
- Valid CCP is required to use Leica predefined base maps.
- Manually added services do not require a valid CCP.

Requirements:

- A coordinate system defined in the project.
- At least one point in the local grid value available in the project.
- Internet access to stream the image tiles or maps.
- Valid CCP is required to use Leica predefined base maps.
- Manually added services do not require a valid CCP.

The base map is drawn below all project data, regardless of height values.

To enable or disable base map:



1.	Go to the **Home** tab and select **Toggle Base Map**.
2.	From the drop-down list, select between the available services.
3.	Define the Max Zoom Level. ☞ The default Max Zoom Level is 19 and in normal circumstances should stay in this setting. ☞ Higher-quality map services can be viewed in a higher zoom level. The larger the values of the Max Zoom Level, the better the resolution.
4.	The base map is drawn below all project data, regardless of the height values.
5.	Enable the 3D terrain from the graphical view tool bar to view base maps in 3D.

To add a new map service, see Map Services.

☞ All base maps can be viewed in 3D. Enable the 3D terrain from the graphical view tool bar. The default DEM has a resolution of approximately one arcsecond.

☞ Log in to Earth Data from Services in order to enable the 3D Terrain model option on the view tool bar.
Elevation data is downloaded and shown once the login is established and the 3D Terrain model option is active.
The downloaded elevation data is stored by Infinity in the folder: C:\ProgramData\Leica Geosystems\Infinity\dems\SRTM\version3_0 to be also used in offline mode.
It is possible to add elevation data to the folder manually, to work with the 3D Terrain model offline.

See also:

Hexagon Imagery Program

OpenStreetMap

Map Services

Get Feature

Google Earth

Feature Info

The video **"Leica Infinity - Home Module - Using Base Maps"** https://www.youtube.com/watch?v=ZPXJSI3WWVM

Once the login is done, and the 3D Terrain model is active,

2.2.5.2 Clip Base Map

Clip Base Map

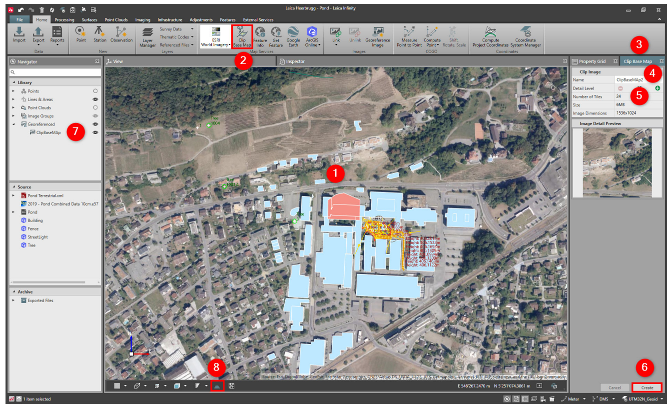
With clip base map, it is possible to clip the base map and store it to the project as a georeferenced image.

When you clip the base map, the image or map that is downloaded is what you see as displayed in the graphical view.

Requirements:

- Active base map.

To clip base map:



1.	Navigate to the map area you want to download.
2.	Select **Clip Base Map** from the ribbon bar.
3.	The clip base map window opens.
4.	Enter a **Name** for the georeferenced image to download.
5.	Select the level of detail. ☞ Available depending on the service set as the active base map. ☞ Base map image services, use tiles of images to cover the area of display. When you download an image, it is therefore downloading tiles, the detail level of the tiles starts from the display in the graphical view. You can increase the level of detail for the image tiles to download. The detail level is normally indicated by a number between the range of 1 - 20, where 20 is the highest level of detail. Keep in mind that the more detail you want to download, the larger the size of image tiles being downloaded. The maximum number of tiles possible to download is 1200, regardless of the level of detail.
6.	Select **Create**.
7.	Georeferenced images are shown as objects in the library and can be set to **visible** or **not visible**.
8.	Enable the **3D terrain** from the graphical view tool bar. ☞ All clipped base maps can be viewed in 3D. ☞ The default DEM has a resolution of approximately one arcsecond.

☞ When exporting a project with georeferenced images to Captivate, the georeferenced images are also exported. To export a georeferenced image to Captivate simply attach it to a project and export the project to Captivate.

See also:

Base Map

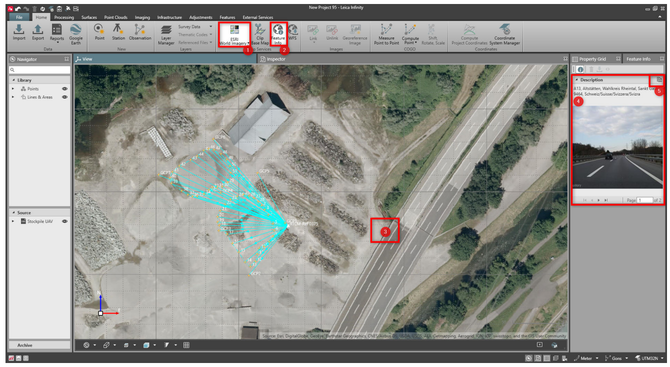
The video **"Leica Infinity - Home Module - Using Base Maps"** https://www.youtube.com/watch?v=ZPXJSI3WWVM

2.2.5.3 Feature Info

Feature Info

With the feature info it is possible to discover data directly from the base map view.

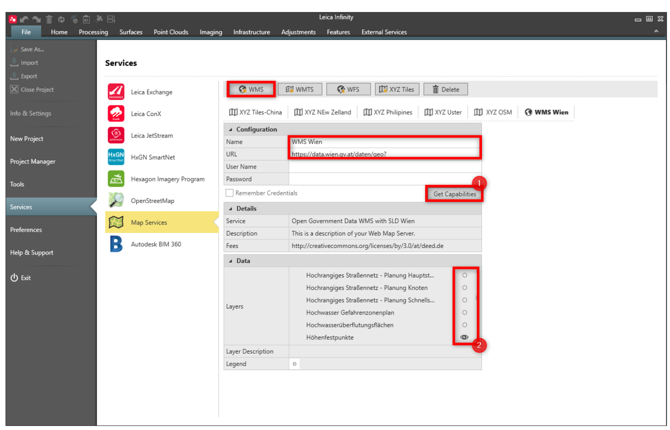
To get the basic feature info from predefined base maps:

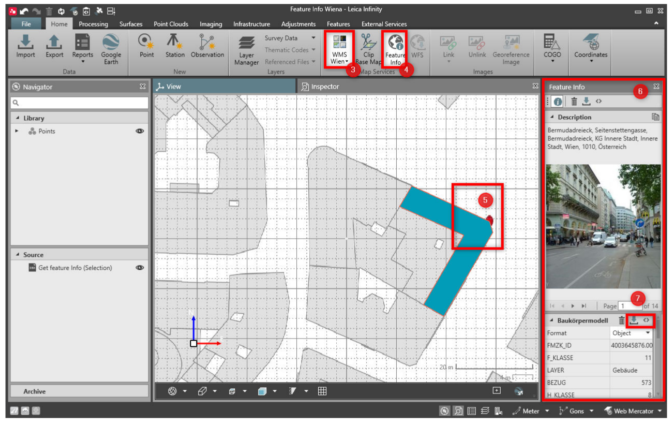


1.	Enable any of the predefined base maps.
2.	From the **Home** tab, select the **Feature Info** option.
3.	Click into the desired area of the displayed map.
4.	Available position information, including street view images from mapillary service is displayed in the property grid.
5.	Copy the active image to the library and show its position in graphical view.

To download feature info like attributes or geometry from user-defined web map service:

☞ Identifying features is done on visible web map service layers.





1. In **File**, define the web map service which supports querying of the features.
   For more information, refer to **File** > **Services** > **Map services**.
2. In **File**, turn the layers **on** or **off**.
   ☞ If too many layers are on, the performance is affected.
3. In the project, select the defined **Web Map Service** as a base map.
4. Select the **Feature Info** option.
5. Click into the desired area of the displayed map.
6. Available position information, including images, feature geometry and attributes is displayed in the property grid.
7. Download the single feature or only the code with attributes.

2.2.5.4 Get Feature

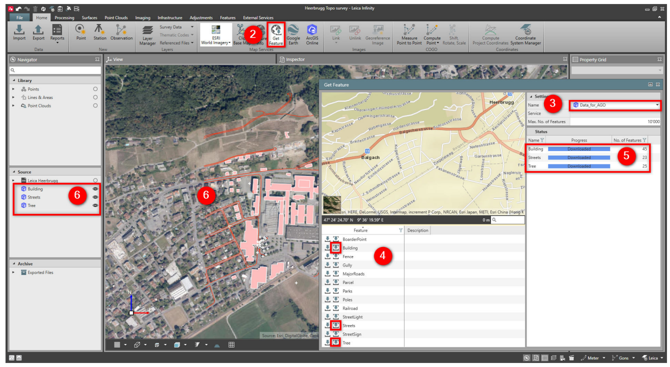
Get Feature

With the get feature it is possible to download features like points, lines, areas, together with their codes and attributes from user-defined Web Feature Service (WFS) or ArcGIS Online feature servers.

☞ When logged in to ArcGIS Online the feature servers are added automatically to the list.

☞ To visualise the data in Infinity, it is necessary to create/download a coordinate system.

To get features:



1.	Define a **WFS** or **ArcGIS feature server** in from the File tab. Refer to Map Services.
2.	To access the service within an open project, go to the **Home** tab > **Map Services** and select the **Get Feature** icon. The Get Feature dialog opens. ☞ The map is located around your project data by default. If the project is empty, the map is located around Heerbrugg, Switzerland by default.
3.	In the Settings table, you can select the **Service** and the **Max. No. of Features**. To select the **Service**, select the arrow of the **Name** field to open a drop-down menu of available services. To select **Max. No. of Features**, click into the **Max. No. of Features** field and enter a number to limit the maximum number of features in one single download. ☞ By default, the Max. No. of Features are set to 10000.
4.	To download the data, select the **Download all** icon or the **Download current view** icon. **Download all**: For downloading all features from the selected layer. **Download current view**: For downloading features from current map view only. ☞ It might happen that some services do not support the Download current view option. If no data can be downloaded, use the **Download all** option.
5.	After starting the download, follow the download in the status section. In the status section, you can view the progress, the number of downloaded features and download errors.
6.	After the download, you can see the features in your project. For selected features, you can see single attributes. From the **Navigator>Source**, you can open an attribute table from the context menu for all features.

It is possible to select an object and run **Copy from CAD**. Refer to How to Copy from CAD?.

2.2.5.5 Google Earth

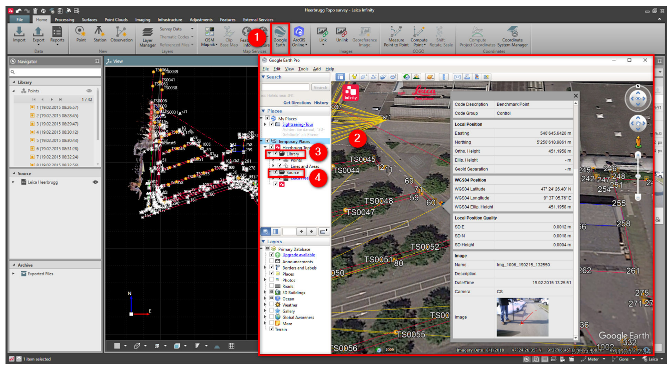
Google Earth

Infinity offers the possibility to view and inspect your project data in Google Earth.

Requirements:

- Google Earth is installed.
- Project data has either WGS84 coordinates or a coordinate system attached.

To visualise the data in Google Earth:



1. Go to the **Home** tab and select the **Google Earth** option from inside the ribbon bar.
2. Google Earth is started and the data is displayed as a Temporary Place.
3. Under **Library**, click the links to get information on the properties of single points, lines, areas or surfaces in a Google Earth flyout.
4. Under **Source**, click the links to get information on the properties of single data sources in a Google Earth flyout.

By default you see all GNSS and TPS observations as well as the point symbols and point ID, each on a separate layer:

To switch off a layer in Google Earth **deselect** it in the tree structure on the left.

To synchronise Google Earth with Infinity:

1. Turn on the Google Earth view synchronisation.
2. Google Earth then follows all zoom, pan and rotation operations you make in Infinity.

2.2.6 COGO Calculations

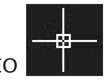
2.2.6.1 Measure Point to Point

Measure Point to Point

Measure point to point is a COGO function and can be used to determine horizontal and vertical angle differences as well as horizontal and slope distance differences between two points in your project. The coordinate differences in Easting, Northing and height are also computed and displayed in the Measure: Point to Point window.

To start the measure point to point function:

Select the  **Measure Point to Point** option in the Home tab to open the Measure: Point to Point window and switch to the Measure Point to Point COGO mode.

The cursor changes to  and you are able to pick start and end points graphically.

Overall behaviour:

Apart from the changed appearance of the cursor in the graphical view, a green line indicates that you are in the measure point to point mode once you have selected a start point. When you select the **Cancel** option in the right bottom corner of the window:

- The cursor returns to its standard appearance.
- The green line disappears.
- The Cancel option becomes a measure option.

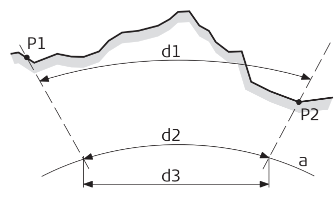
You can leave the Measure Point to Point window open next to the Propertiy Grid and by selecting the **Measure** option, you can always pick up the COGO calculation again. By default, the Measure Point to Point window is docked to the Propertiy Grid and opened in its own tab. When you close the Measure Point to Point window, the Propertiy Grid remains.

Input options:

Field	Option	Description
Mode		There are two input modes of determining angles and distances between two points. You can switch the mode without having to re-invoke the function.
	Radial	In the Radial mode you can select a central point from which angles and distances to other points shall be determined in a star-like manner around the selected point. Select the **Finish** option to: • Complete a measuring loop. • Stop the COGO function. • Return to the standard mode.
	Sequential	In the Sequential mode you can select a start point and an end point for your calculations and after that another start and another end point. Select the **Finish** option in the right bottom corner of the window to: • Stop the COGO function. • Return to the standard mode.

☞ In the Sequential mode a measuring loop is automatically finished when the end point has been selected. In the Radial mode the measuring loop is kept open (graphically this is visualised by the green line sticking to the cursor) until you manually select to finish it.

Field	Option	Description
Distance type		The type of distances displayed in the results.
	Grid	Distances are calculated as the trigonometric distance between the position of two points. The distance field is horizontal distance.
	Ground	Distances are horizontal distances between two points at the mean elevation parallel to the ellipsoid of the active coordinate system. The distance field is horizontal distance (ground).
	Ellipsoid	Distances are reduced to the ellipsoid. They are calculated as the shortest distance between the two points on the ellipsoid. A scale factor is applied. The distance field is horizontal distance (ellipsoid). ☞ In the attached coordinate system, a projection, an ellipsoid and a transformation have to be defined to calculate grid, ground and ellipsoid coordinates.



a Ellipsoid

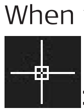
Known

P1 First known point
P2 Second known point

Unknown

d1 Ground distance
d2 Ellipsoid distance
d3 Grid distance

How to measure point to point?

When you are in the Measure Point to Point mode and the cursor appears as :

1. In the graphical view, click on the **start** point from which you want to determine angles and distances to one or more end points.
   The Point ID is automatically entered into the 1st Point field at the top of the window.
2. Click on the **end** point. The Point ID is automatically entered into the 2nd Point field at the top of the window.

☞ If there exists more than one point role for the selected points, pick the required one from the point selection menu that automatically appears if for a point more than one point role is available.

☞ Select the  option in the 1st Point/2nd Point fields to select the start and end points for the calculation non-graphically from a flyout.

3. Inspect the results in the Measure: Point to Point window. The results are listed one below the other. Each result section can be collapsed or closed individually.

Create a report:

You can create a report of the measured points by selecting the **Report** option at the bottom of the Measure Point to Point window.

The Report Manager opens and shows the generated report.

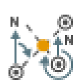
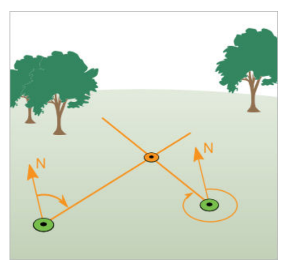
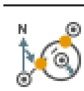
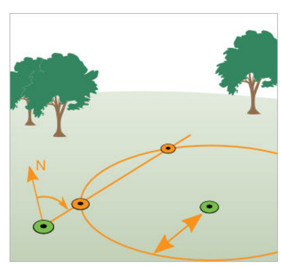
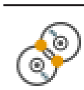
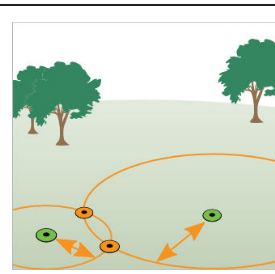
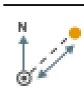
2.2.6.2 Compute Point

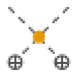
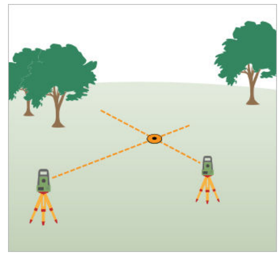
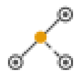
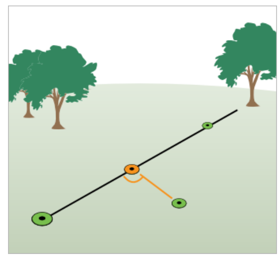
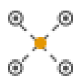
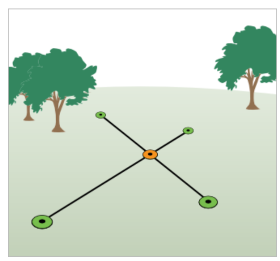
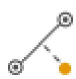
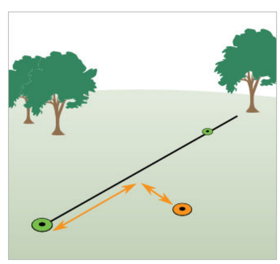
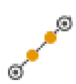
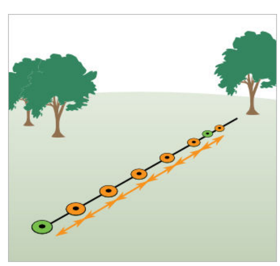
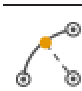
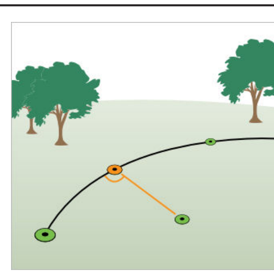
Compute Point

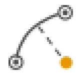
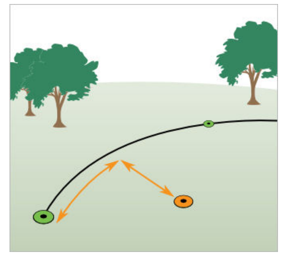
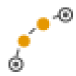
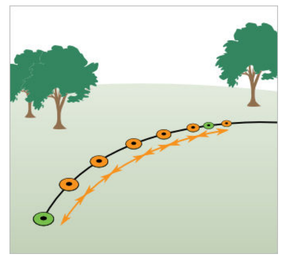
Compute point is one of the COGO functions and can be used:

- To determine point positions relative to existing geometry objects, like base points and offset points.
- To determine point positions by intersection or segmentation of existing geometry objects.
- To determine point positions by defining a direction and distance from a known point.

The following methods are supported:

Icon	Method	Illustration	Description
	Bearing Bearing		Calculates the intersection point between two selected points using defined bearings.
	Bearing Distance		Calculates the intersection points between two selected points using a defined bearing from one point and a defined distance from the other.
	Distance Distance		Calculates the intersection points between two selected points using defined distances.
	Point in Direction		Calculates a point using a defined direction and distance from a known point.

Icon	Method	Illustration	Description
	Two Observations		Calculates the intersection point between two selected TPS observations.
	Line Base Point		Calculates the position of a base point on a defined line where the offset to a defined point is perpendicular to the line.
	Line Intersection		Calculates the intersection point between two defined lines.
	Line Offset Point		Calculates the position of an offset point using an entered distance along and a perpendicular offset from a defined line.
	Line Segmentation		Splits a line into multiple segments. The segment length is derived from either a given number of segments or a specified segment length.
	Arc Base Point		Calculates the position of a base point on a defined arc where the offset to a defined point is perpendicular to the arc.

Icon	Method	Illustration	Description
	Arc Offset Point		Calculates the position of an offset point using an entered distance along and a perpendicular offset from a defined arc.
	Arc Segmentation		Splits an arc into multiple segments. The segment length is derived from either a required number of segments or a specified segment length.

How to compute points?

1. Go to **COGO** > **Compute Point** in the Home tab and select a method from the drop-down menu.
   The Compute Point Property Grid opens up next to the regular Property Grid.
   ☞ You can still change the method here.
   ☞ For some methods, it is necessary to select how the geometry shall be defined.
   Select a Mode from the drop-down box.

2. Select  the points and/or observations based on which the computation shall be performed. You can only select existing Infinity objects as input to the computation.
   Objects can also be selected graphically in the graphical view.
   To select observations, TPS observations have to visible in the graphical view. See also Layer Manager.
   ☞ If you cancel the operation or if you have already run a computation and want to run another, then select **Measure** at the bottom of the Property Grid to activate point selection again in the graphical view.

3. Depending on the calculation method, you may need or want to define azimuths, distances or offsets.
   Enter the values or select the  option to calculate the values.
   ☞ For methods Bearing Bearing, Bearing Distance and Distance Distance the values can also be derived from selecting points graphically for computation.

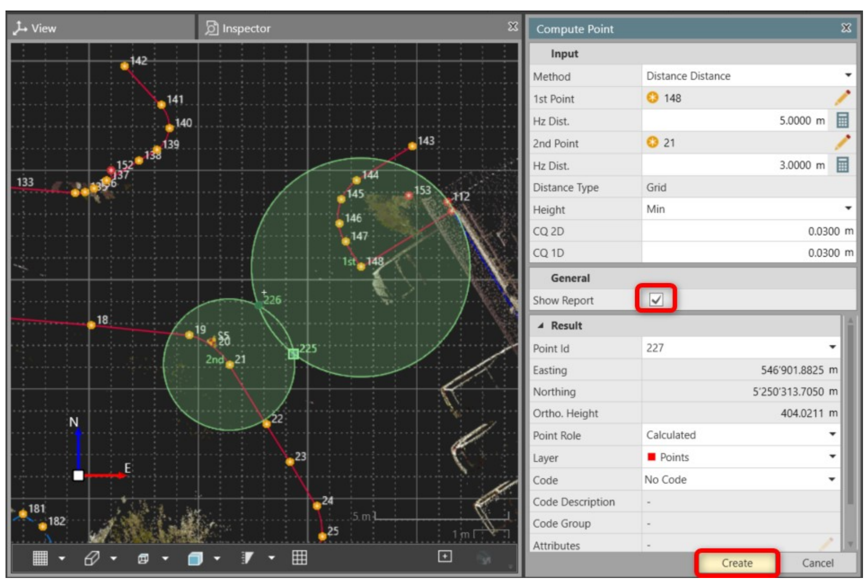
4. Select the Height at which the computed points shall be calculated. You have the choice between:
   None
   No height is computed.
   Max
   The point is computed at the same height as the point with the maximum height, that has been used for computation.
   Min
   The point is computed at the same height as the point with the minimum height, that has been used for computation.
   Average
   The point is computed at an average height of all points that have been used for computation.

5. Check the box for **Show report**, when you need a report of the COGO computation.

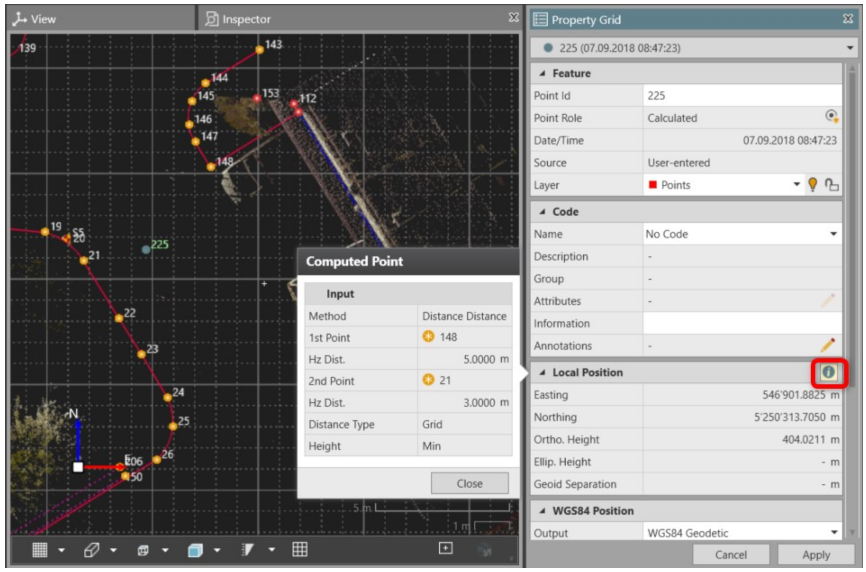
6. The Result of the computation is shown in the Property Grid once all necessary input is available.
   In the Results panel, you can change the Point Role and Coding information of the computed point.

   ☞ If more than one point results from the computation (for example in the case of Distance Distance computations) select which of the points shall be saved.

7. Select **Create** to save the computed points as objects to the project database.
   If selected before, the COGO report opens automatically.



8. Computed COGO points are selectable.
A flyout displays the method and values used to compute the COGO point. In the properties, select in Local Position.



2.2.6.3 GNSS Hidden Points

GNSS Hidden Points

Hidden points are points that cannot be directly accessed with the GS. For example, if a point physically cannot be reached or because no satellites can be tracked in the point to be measured due to obstructions.

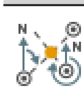
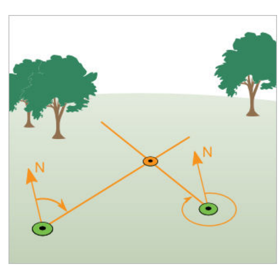
The hidden points can be measured in the field using Leica Captivate, Leica SmartWorx or Leica Viva.

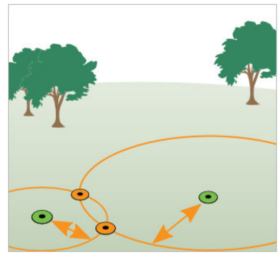
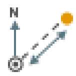
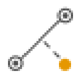
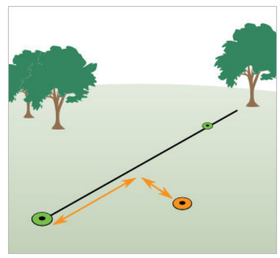
Infinity supports importing, editing and updating these points.

The lines between the points which are used for the calculation are visualised on the hidden point survey layer.

Importing GNSS Hidden Points

The following methods are supported:

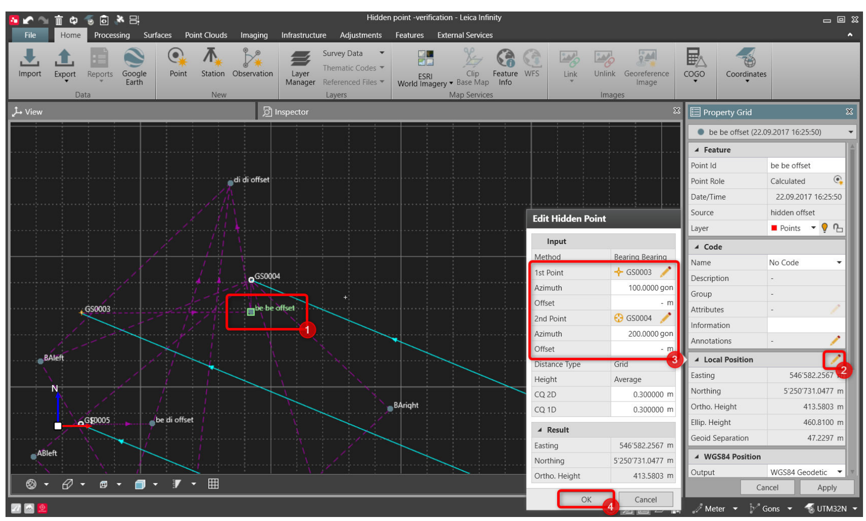
Icon	Method	Illustration	Description
	Bearing Bear-ing		Calculates the intersection point between two selected points using defined bear-ings.

Icon	Method	Illustration	Description
	Distance Distance		Calculates the intersection points between two selected points using defined distances.
	Point in Direction		Calculates a point using a defined direction and distance from a known point.
	Line Offset Point		Calculates the position of an offset point using an entered distance along and a perpendicular offset from a defined line.

Hidden points are included in the COGO report and in the data source report.

Editing GNSS hidden points:

1. Select a computed hidden point.
2. In Local Position, select ✎.
   A flyout displays the method and values used to compute the point. The parameters available in the flyout are the same available in the COGO report for that specific point.



3. Change the values.
   ☞ New coordinates of the hidden point are computed as soon as some values are changed. The new coordinates are shown in the flyout.

4. Select **OK**.

   ☞ The Property Grid is updated and new coordinates are stored.

Updating GNSS Hidden Points

The coordinates of a hidden point are automatically recomputed when the coordinates of a point which has been previously used in hidden point measurements are changed.

2.2.6.4 Shift, Rotate, Scale

Shift, Rotate, Scale

Shift, rotate, scale is a COGO function and can be used to match a set of preliminary points with a set of control points.

You can also shift, rotate, scale point clouds, CAD geometries, BIM objects and TPS setups.

☞ All BIM entities in a file are transformed even if only a single element has been selected.
The BIM entities used as common points, must be copied to the Infinity library first. This can be done by using the **New Point** option or **Copy from CAD**.

See also:

How to Create New Points from CAD Entities?

How to Copy from CAD?

To start the shift, rotate, scale wizard:

1. Select the entity to be transformed in the graphical view or in the Inspector.
   Setups can also be selected in the Source section of the Navigator.

2. Go to the Home tab and select **Shift, Rotate, Scale** in the COGO group.

In the shift, rotate, scale wizard:

1. Select the type and the method.
   You can enter the parameters for shift, rotate, scale manually or allow Infinity to compute them using common points.

When you select the method: Enter manually or compute separately, you must:

2. Enter the Shift in Easting, Northing and Height. Select the option to compute the shift:
   Under 'From', select a point in the preliminary system. Under 'To,' select the Control point in the target system.

3.	Enter the Rotation angle. Select the option to compute the rotation angle. Under From, select a pair of points in the preliminary system. Under To, select the same pair of control points in the target system. Select the to select a point as the origin for the rotation. This can be either in the preliminary system or in the target system.
4.	Enter the Scale Factor. Select the option to compute the scale using points or known distances. Under From, select a pair of points in the preliminary system. Under To, select the same pair of control points in the target system.
5.	Select **Next** to inspect the results.
6.	Select **Finish** to perform the operation.

When you select the method: Compute using common points, the transformation parameters are computed automatically in the background using the common points:

2.	Select whether you want to compute a Rotation and a Scale using common points or None.
3.	Select **Next** to match points.
4.	Under System A, select the preliminary measured points. Under System B, select the corresponding control points.
5.	Select to match the pair of points. It is listed in the following view: ☞ If you match two pairs of points height residuals are calculated. If you match three or more pairs residuals are calculated for position and height. Errors are distributed. ☞ To delete a mismatched pair select .
6.	Select **Next** to inspect the results.
7.	Select **Finish** to perform the operation.

By default, a report is created and opened when you select **Finish**. You can Save and/or Print the report from within the report window.

Saved reports are written as PDF to the archive. You can reopen them from there.

☞ Unsaved shift, rotate, scale reports are lost when you close them.

☞ Only coordinates stored as local grid can be transformed.

☞ The following two types of transformations are supported:
The 5 - Parameter transformation, is ideal for shifting, rotating and scaling TPS data to fit to GNSS data. The main assumption here is that the source and the target data have parallel height axes. The scale is applied in Easting and Northing.
The 7 - Parameter transformation is a 3D Helmert transformation. It can be used to shift, rotate and scale a BIM object to fit to a point cloud. The scale is uniform in all three dimensions. The additional parameters are the rotations around the East and North axes.

2.2.7 Compute Project Coordinates

Compute Project Coordinates

Allows you to apply a combined scale factor and additional shifts in Easting and Northing in order to compute project coordinates from existing grid coordinates.

In the Home tab, select the **Compute Project Coordinates** option.

The Project Coordinates tool opens up next to the Property Grid.

1. Select a method.
   The following methods are available:
   User entered
   Enter the required combined scale factor manually, and optionally a shift in Easting and Northing.
   Base Point
   Select the Edit icon and select a base point from the flyout.
   The grid scale, the height scale and the combined scale factors are computed based on the selected point.
   Additional shifts in Easting and Northing can be entered manually.
   If the option **Hold Base Point Coordinates** is selected and no additional shifts are applied, the grid coordinates of the selected Base Point are equal to its project coordinates.
   Selection
   The average grid scale, height scale and combined scale factors are computed based on the values that are computed for each of the selected points individually.
   Additional shifts in Easting and Northing can be entered manually.
2. Select **Apply** at the bottom of the tool window.

Project coordinates are computed for all points that are either stored as local grid or can be converted to local grid using the attached coordinate system.

☞ The project coordinate system must have a map projection of type transverse mercator, UTM, Lambert two or double stereographic. Check the attached coordinate system in the coordinate system manager.

Inspect the results in the **Inspector** > **Features** tab under **Points**. If not switched on already right-click on the column headers and select the columns **Project Easting** and **Project Northing** to be displayed in addition to the regular Easting and Northing columns. There are also columns for the three scale factors for each point.

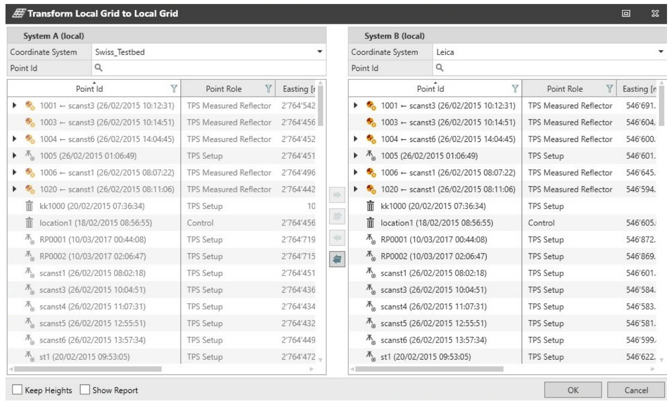
☞ The project coordinates including scale factors can be exported to an ASCII file using the ASCII export.

2.2.8 Transform Local Grid to Local Grid

Transform Local Grid to Local Grid

Allows you to transform the points stored with local grid coordinates of initial system A to local grid coordinates in the target system B.

To transform local grid coordinates:



1. Open the **Transform Local Grid to Local Grid** tool.

2. Select the points you want to transform from the left pane.

3. Move the points to the right pane; a preview of the transformed coordinates is available.

   ☞ Infinity uses the parameters of coordinate system A to convert the Easting, Northing and Orthometric Height of each selected point to WGS84 coordinates.
   It then uses the parameters of coordinate system B to convert the WGS84 coordinates of each selected point to the new local Easting, Northing, Orthometric Height.
   Along with each TPS setup, the sideshots are also transformed.
   Points with no horizontal position are not transformed.

4. Enable the option to **Keep Heights**.

   ☞ If the option Keep Heights is enabled, the point heights are not transformed in system B, even though the new heights are displayed in the preview.

5. Enable the option to generate the report.

   ☞ You can create a report that has the transformed points. In the Results section of the report, the WGS84 coordinates of the points are also displayed.

6. Select **OK**.

   ☞ The coordinate system B is automatically set as the Master, to ensure that the projects GNSS data (if any) fits into the transformed points.

2.3 Features

2.3.1 Overview

Features

In the features module, you can manage the thematic information of imported or created data within the current project. Points, lines and areas are features and can have thematic data assigned to them.

Feature coding is used to describe topographical details of features in the field. The code table holds all the thematic and style information that is applicable to the field data in Infinity. Code tables can be created or registered from within the **File** > **Code Table Management**.

See also:

Code Tables

To apply coding style information to the field data, you can assign a code table to the project while creating a new project or later from within the **File > Info & Settings**.

See also:

New Project

Info & Settings

After importing data to the project, use the inspector to view all features collected with or without codes. This is also where you can see if a code was created in the field and is not in the code table.

You can edit a feature code for a single point in the properties.

Feature codes can be managed locally inside a project by the project code manager, which allows you to edit or apply styling. Features in the project, can be edited to have an individual style. All feature code edits made with the project code manager are project-specific.

See also:

Project Code Manager

Features in the Inspector

In the **Feature Coding** tab of the Inspector, all the thematic codes attached to your current project are displayed. You can sort the view by code groups or layer or code name.

The coding information is grouped into two groups:

- **Code Table**
  Lists the codes that belong to the projects code table.
- **Field Codes**
  Lists imported new codes that are not part of the code table.

☞ You cannot modify any information in this view.

Features in the Properties window

In the **Properties** window, you can find also coding information for the selected object (point, line and area). The information is divided into Feature, Styling and Code sections. These properties can be changed individually:

- In the **Feature** section, the Layer can be selected or modified to which the object belongs to.
- In the **Styling** section, the line colour, line width and shading colour for lines and areas can be modified.
- In **Code** section, you can assign another code to the object within a code group.

Feature coding workflow

General steps to follow:

1. Create a Code Table with codes, coding style and layer information including blocks in Infinity. You can also register an existing Code Table.

2.	Export a Code Table to a codelist for a specific instrument and use it during data collection in the field.
3.	Assign a Code Table to your project while creating it. You can assign a Code Table also later on when you miss this step.
4.	Import data to the project. The data is processed based on the assigned Code Table.
5.	Check field data to be correct with the thematic information. If needed then manage codes using the Project Code Manager and export the modified Code Table to the instrument.
6.	Export the data to a DXF/DWG file.

2.3.2 How to Create New Features?

How to Create New Features?

Functionality to create new features is available through the ribbon bar in the Features tab. Alternatively, you can access the same functions from within the context menu in the graphical view.

☞ New points can also be created through the ribbon bar of the Home tab or the Processing tab.

To indicate that feature creation is on, the cursor appears as a crosshair during all create operations.

New point:

New points can be defined:

- Through the Property Grid. Enter the point details and select **Create** at the bottom.
- By picking points from CAD, BIM, ESRI Shape files entities as well as base maps.

See also:

How to Create New Points from CAD Entities?

New line, arc, spline and area:

New lines, arcs, splines or areas are drawn in the graphical view. Their properties can be modified through the property grid.

1.	Select the first point by clicking onto it in the graphical view.
2.	Click the next point and continue until you come to the last point.
3.	Right-click and select: **End**, to finalise the feature. Alternatively, select **Create** in the Property Grid. The feature is created as specified in the Property Grid. The cursor changes back to normal. **Remove last point**, to remove the point that you last selected. Through this function, you undo your operations step-by-step. **Cancel**, to cancel the operation. Alternatively, select **Cancel** in the Property Grid. **Close**, to end a line feature on its start point. **Continue...**, to continue a feature using a different type of geometry.

See also:

How to Edit the Feature Geometry?

☞ Lines have to consist of two points minimum. Arcs can always only consist of three points maximum.

☞ Areas and closed lines have to consist of three points minimum.

2.3.3 How to Create New Points from CAD Entities?

How to Create New Points from CAD Entities?

1. Click on  **New Point** in the ribbon bar.

   ☞ To indicate that feature creation is on, the cursor appears as a cross hair during all Create operations.

2. In the graphical view pick a point. This can be the vertex point of CAD/BIM entities, ESRI Shape files or of a point cloud (surface).
   The new point snaps to the existing point/vertex and adopts its coordinates.

   ☞ The coordinates and other point properties can be viewed and edited in the New Point tool.

   Change the Point ID, Code and Layer information if necessary.

3. Pick the next point.
   Every time you pick a point it is added to the list in the Property Grid. The grid always displays the properties of the point that has been picked last.

   ☞ To view/edit the properties of a previously picked point open the drop-down list at the top of the Property Grid and select it from the list.

4. When you have picked all points that shall be created press the **Create** button in the Property Grid.
   Only then are the points created and added to the Infinity library.

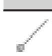
2.3.4 How to Edit the Feature Geometry?

How to Edit the Feature Geometry?

The functionality to edit features is available from within the context menu in the graphical view.

To edit features through the ribbon bar go to the **Features** tab. Activate the feature to be edited by selecting it.

To indicate that feature editing is on, the cursor appears as a cross hair during all edit operations.

Icon	Function	Description
 	Add point at Start/End of a Line	You can choose to add points either at the start or the end of a line or spline. Click the points to be added. When you are done, right-click and select **End** from the context menu or select **Create** in the property grid.

Icon	Function	Description	
	Add point in Between	You can choose to add points in between. Right-click the segment where additional points shall be inserted and select **Add Point**, then **in Between**. The existing segment is removed. When you are done, right-click and select **End** from the context menu or select **Create** in the property grid. ☞ In Between is only available through the context menu.	
	Remove	You can remove points from a feature. Select the feature and click the point to be removed. ☞ In order to remove a point through the context menu, select the point and not the feature.	
	Continue	You can continue features with the same or another feature type. Follow the Add Point workflow. The feature is continued at its end. ☞ You cannot use continue with closed lines or area features to continue a closed line or an area, go and add points in between (see previous).	
	Convert	You can convert lines to splines or splines to lines.	
	Convert to Arc (Midpoint)	You can also convert a feature to an arc. Right-click on a point and select **Convert to Arc** from the context menu. The arc is created from the previous to the next point. ☞ Features consisting of less than three points cannot be converted.	
	Join	You can join line features that share their start and/or end points.	
	Split	You can split features. Click the point on which you want to split the feature. ☞ When you split an area it is converted into lines.	
	Reverse	You can reverse the direction of a feature, which means that the start point becomes the end point and vice versa.	

2.3.5 Multi-Editing of Point Code Information

Multi-Editing of Point Code Information

To edit the coding information for more than one point at a time:

1. Select the points for which you want to edit the code information either from inside the Library section of the Navigator or from inside the Points and Lines & Areas subtabs of the Inspector.
   In the **Points** and **Lines & Areas** subtabs of the Inspector, go to the **Points** subtab and select the **points**.
2. Right-click into the selection and select **Edit Code** from the context menu.

The edit code operation is done for all selected points.

☞ For observations, you can multi-edit the target point codes. For further information, see Observation Properties.

2.3.6 How to Copy from CAD?

How to Copy from CAD?

Functionality to copy from CAD is available through the ribbon bar in the features tab. Alternatively, you can access the same functions from within the context menu in the graphical view.

You can import entities from *.dxf files (CAD data), from *.shp files (GIS data) as well as from *.ifc files (BIM data).

1. In the graphical view, select the entities that you want to import. To select more than one element at a time keep the Ctrl-key pressed while clicking onto each element or keep the Shift-key pressed while dragging a rectangle around the group of elements to be selected.
   See also:
   Graphical View
2. Modify the Copy Settings if necessary.
   See also:
   Copy from CAD: Settings
3. Select one of the following options:
   All, to import the selected entities and their points.

   ☞ If you select a block this option is not available.

   ☞ You can also copy from CAD through the property grid. In the feature tab, select the option next to Source. This always imports all.

   ☞ If you select a surface, points are not included.

   Points, to import only the points belonging to the selected entities.
   All from Layer, to import all features and points that are on the same layer as the selected entity.
   All Points from Layer, to import only the points that are on the same layer as the selected entity.

   ☞ If you have selected more than one entity then this function is only available if both entities are on the same layer.

   To Alignment, to import all selected entities to the project as alignment objects.
   To Road, to import all selected entities to the project as a road object.

The entities and/or points are copied to the Infinity library.

The reference layer is copied to the thematical layers and the imported feature belongs to it.

☞ On how to manage layers, see Project Code Manager.

☞ Go to **File** > **Info & Settings** to define defaults for how CAD entities shall be written to the Infinity library.

2.3.7 Copy from CAD: Settings

Copy from CAD: Settings

To open the copy CAD settings:

Select **Copy Settings** in the Features ribbon bar.

By default the Copy CAD Setting window opens up as an extra pane next to the property grid.

The copy CAD Settings can be set separately for points, lines, areas, surfaces, alignments and roads.

☞ For roading features (alignments, stringlines, roads and material layers), the default naming is done by using the CAD layer name. If manual input is desired, you can activate the checkbox and specify a different ID. When there are multiple features in one layer, the features are incremented accordingly.

☞ In the File tab under Info & Settings, you can pre-define which settings are assumed as default in each tab.
To restore the Infinity defaults go to the **File** > **Info & Settings** > **Copy CAD to library** and select the **Default** option.
Defaults can be restored separately for points, lines, areas, surfaces, alignments and roads.

See also:

Info & Settings

2.3.8 Project Code Manager

Project Code Manager

The project code manager allows you to manage all the thematic codes and layers attached to your current project. Including the structured code table that you may have attached to your project and unassigned field codes that have been imported with your data.

When importing data into the project codes that are related to data are checked against the projects code table. If an imported code is not included in the projects code table or it has more attributes than the same code in the code table then the imported code is shown in the field codes part of the navigator.

To launch the project code manager:

In the **Features** module, select **Code Manager** from within the ribbon bar.

The Project Code Manager opens in a separate window.

The assigned Code Table and the Field Codes are shown even if they are empty.

To manage coding and coding style information:

In the Project Code Manager, select the project Code Table and go to the **Code Table** tab.

Choose from the following possibilities:

- Add a new code group/code/attribute.
- Edit a code group/code/attribute.
- Delete a code group/code/attribute.

☞ Field codes cannot be edited within the project. To manage field codes within the project add them to the project code table.

To add imported field codes to the project code table:

1.	Under **Field Codes**, navigate to the Code Group that has the code that you want to add to the project Code Table.
2.	Select the code in the **Content View** and cut it off from Field Codes by pressing **Ctrl+X** on the keyboard.
3.	Navigate to the Code Group where you want to insert the code in your Code Table and press **Ctrl+V**.
4.	Edit the code properties if needed.
5.	To save your changes select **Apply** and **OK**.

☞ You can select and move not only one code but more codes, a whole code group or a single attribute from field codes to the code table.

☞ Conflicts between the project code table content and the newly dropped in content is highlighted. Resolve conflicts before applying changes.

To manage thematic layers:

In the Project Code Manager, select the Code Table and go to the **Layers** tab.

Choose from the following possibilities:

- Add a new Layer to extend thematic grouping for the project features.
- Edit a layer to edit the style of all features on this layer.
- Delete a layer to remove the thematic grouping and style from the project.

To manage blocks:

In the Project Code Manager, select the Code Table and go to the **Blocks** tab.

Choose from the following possibilities:

- Edit a Block Name.
- Delete a Block to remove it from the project.

Export a project Code Table:

The project Code Table can be exported to:

- A **Infinity Code Table file** for use by another Infinity user.
- A **SmartWorx DBX** to be used on an instrument.

2.3.9 Rename

Rename

The rename tool, allows you to rename a selection of point ID, and likewise line, area, image and point cloud ID.

You can rename the selected items:

- Using search & replace to replace (or remove) some characters.
- Using rename by renumbering starting from a specific value using a counter.
- Using rename by adding fixed characters to the current ID.

Search & Replace

To search:

You may search for parts of the current ID that shall be renamed and replace these parts with a new combination of alphanumeric characters.

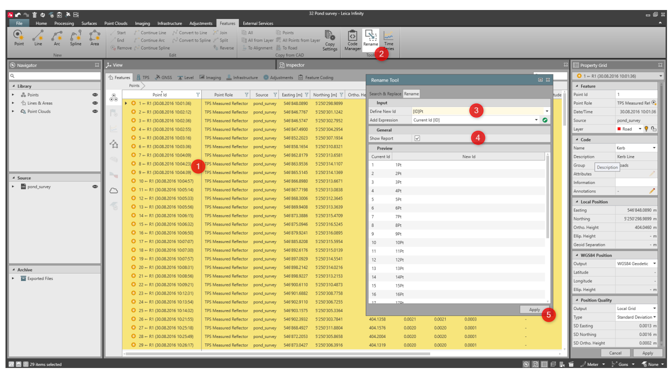
You may also search for spaces, dashes, dots and underscores to be replaced.

To replace:

Define a replacement for the search string. Spaces, dashes, dots and underscores may also serve or be included as part of the replace string. If you want to remove a fixed part of the current ID leave the replace string empty.

Rename

To rename points, lines, areas, images or point clouds:



1. In the Inspector, go to the **Points, Lines & Areas** tab or to the **Observations** tab and select the items you want to rename.
   In the Navigator, items can be selected from the **Library** section.
   Points can also be selected from inside the **Source** section by selecting the job.
   Alternatively, you can also select items in the graphical view.
2. Select the **Rename** option in the **Features** ribbon bar or select **Rename** from the context menu to open the Rename Tool.
3. Define a new ID.
4. Check the **Show Report** checkbox to show the Rename Report.
5. Select **Apply**.

☞ When changing the selection, the items to rename are updated in the preview.

☞ To remove a selected item from the list, so it is not renamed, select the items and select the **Remove** option.

To define a new ID:

Enter any combination of alphanumeric characters and/or add the expressions by selecting the **Add** option.

☞ Alternatively type [C], [ID] or [ID#-#].

To add an expression:

Select an expression and select the **Add** option.

Three expressions are available:

Numeric Counter [C]:	Numeric Counter for renumbering the ID. The following selection fields become active to define the counter: **Start Counter at:** Define the number to start counting with. **Increment Counter by:** Define how the counter shall be incremented. **Counter-Digits:** Define the digits that shall be reserved for inserting the counter.
Current ID [ID]:	The ID which the selected items currently have. To modify the current ID, you can enter any combination of alphanumeric characters or add a counter before or after.
Part of Current ID [ID #-#]:	A part of the ID which the selected items currently have. To modify a part of current ID, you can enter any combination of alphanumeric characters or add a counter before or after.

2.3.10 Time Series

Time Series

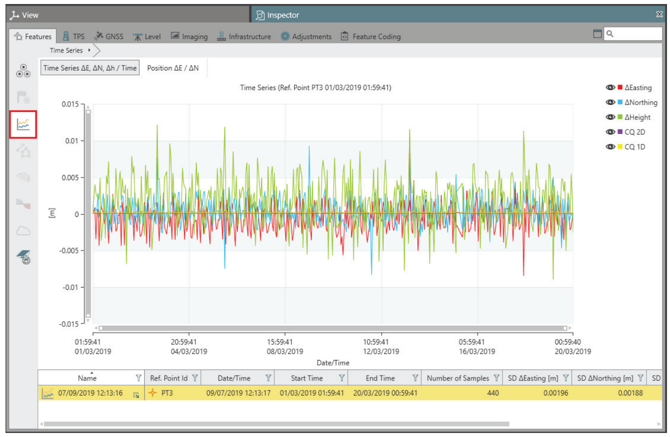
The time series feature, allows you to analyse how coordinates of a point vary over a period, by creating plots over the time.

The plots are created choosing one point as a reference, the highest point role or first measured point in time and computing the differences from that point.

The pre-condition to use the option is the availability of measurements of the same point (same point ID) at different dates and times.

Once the time series plots have been created, it is possible to edit the reference points, see **To edit time series reference points:**.

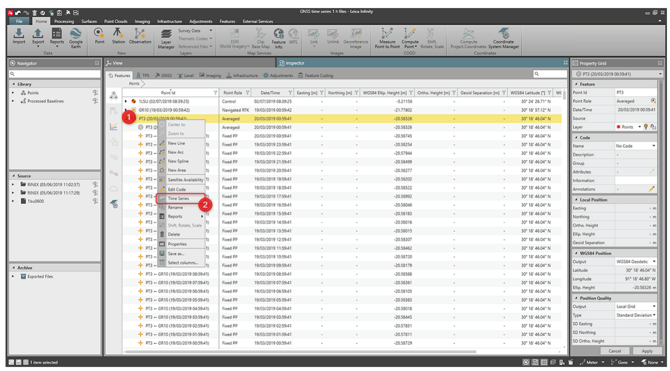
The time series results are always available in the application and are accessible from the **Inspector** > **Features** > **Time Series**.



To create a time series plot:

From Points

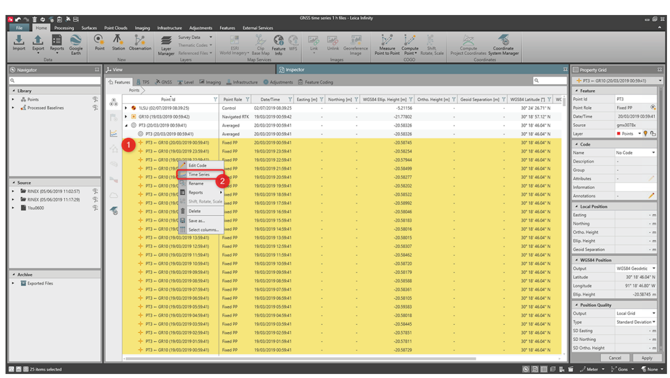
It is possible to create time series for a point measured over a time, independently from how it has been measured (TPS, GNSS, Level).



1. In the **Inspector**, select **Features**, then **Points** and then the point to be plotted.
2. Right-click to open the context menu and select **Time Series**.

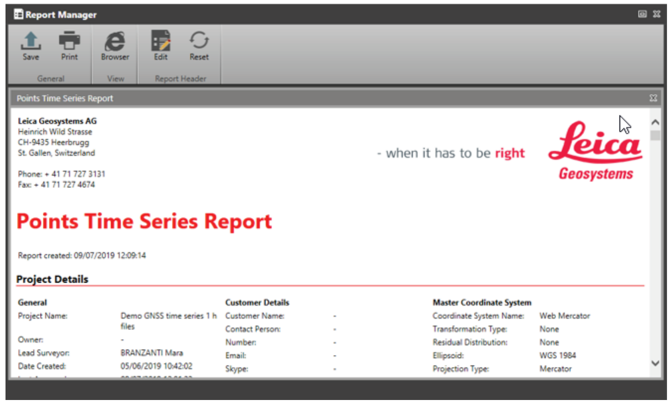
☞ To plot all the points for the highest point role, select only the highest point role. In the example, all the contributors are plotted with the average coordinates of PT3.

To plot a subset of points for the first measured points:



1. In the **Inspector**, select **Features**, then **Points** and then the points of interest.
2. Right-click to open the context menu and select **Time Series**.

After the process, the Points Time Series Report with the results opens automatically.

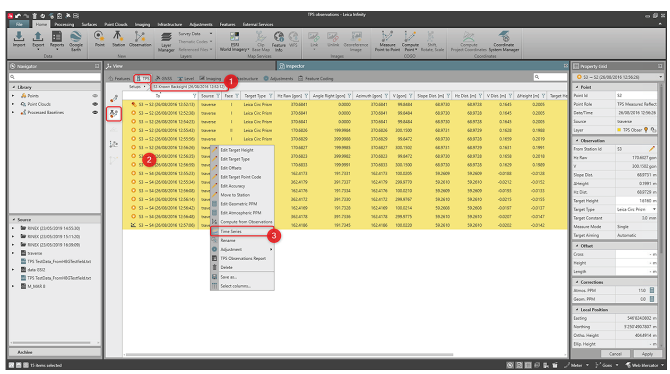


Results are stored and are always accessible from the **Inspector** > **Features** > **Time Series**.

From TPS Observations:

With TPS measurements, it can be convenient to start the process from TPS observations, to select only the ones coming from the same setup.

In the following picture, the TPS reduced measurement is taken as the reference.



1. In the **Inspector**, select **TPS**, then **TPS Observations by Station Source** and then drill into the station source.
2. Select the observations to plot.
3. Select **Time Series** from the context menu to create the point time series plot.

☞ If all the observations have the same point role, the first measurement is taken as the reference.

☞ If available, the highest post role is taken as the reference.

The Points Time Series Report opens automatically, and the results are always accessible from the **Inspector** > **Features** > **Time Series**.

From GNSS Observations:

The time series option can also be started from GNSS observations, so that the relative measured points are plotted.

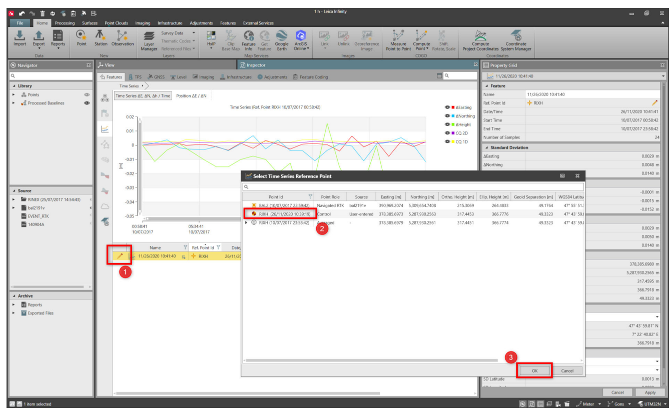
To do that:

1.	In the **Inspector**, select **GNSS**, then **GNSS Observations by Station Source** and drill into station source.
2.	Select the observations to plot.
3.	Select **Time Series** from the context menu to create the point time series plot starting from the selected observations.

The Points Time Series Report opens automatically, and the results are always available from the **Inspector** > **Features** > **Time Series**.

To edit time series reference points:

Once time series have been created, it is possible to change the reference points used to compute the residuals, by choosing another existing point in the project. Plots and statistics are automatically updated.



1.	In the **Inspector**, select **Features**, then **Time Series** and then the **Edit Pencil** option, to select the results you want to edit.
2.	Select a point from the project, that you want to use as a new reference point.
3.	Select **OK**.

2.3.11 Staked Points in Infinity

Staked Points in Infinity

Make use of Infinity to:

- Prepare data for use with stakeout applications in the field. Such data can come from imported files including CAD files.
- Import field stakeout application data to a project and view the details in the inspector and the property grid; as well as within specific stakeout reports.

☞ To see the differences in the graphical view the view needs to be zoomed in on single points.

Supported is the import of job data collected with the applications:

- Stake points
- Stake DTM
- Stake points & DTM
- Stake to line (line, line with slope, grid, segment, segment with slope, quick line)
- Stake road/check road

In Infinity staked points are written to the points library as well as to a library of their own called staked points.

- Staked points are grouped for easy viewing in the **Inspector** > **Features** tab.
- Stake road/check road results are grouped for easy viewing in the **Inspector** > **Infrastructure** tab.
- Run a report on selected results either by selecting the **Reports** option in the **Home** tab or by selecting **Report** > **Staked Points** from the context menu.
  There are additional reports for staked infrastructure and checked infrastructure. You can run them from general list of reports but also from the Infrastructure ribbon bar.

Define the stakeout tolerances in **File** > **Info & Settings**. Stakeout results exceeding the tolerances are marked in bold red in the report.

See also:

Reports

Stakeout Workflows

Make design points coming from CAD available for use with stakeout field applications:

1.	Import of the CAD job into Infinity.
2.	Copy the CAD entities to be staked out from CAD into a project.
3.	Export the project for use within the stakeout application in the field.

See also:

How to Copy from CAD?

To check and document stakeout results import the stakeout job into Infinity.

The aim is to check the stakeout results. The differences between design coordinates and stakeout results can be:

- Seen in the graphical view.
- Checked in the **Inspector** > **Features** tab or in the **Infrastructure** tab under **Checks and Stakes**.
- Saved in a report.

2.3.12 Points Measured to Reference

Points Measured to Reference

Import measured to line data from the field application to an Infinity project.

View the details in the inspector and the property grid, and document the results with a specific points measured to reference report.

In Infinity measured to reference points are written to the points library as well as to a library of their own called points measured to reference.

- Measured to Reference points are grouped for easy viewing in the **Inspector** > **Features** tab.
- Run a report on selected results by selecting the **Reports** option in the Home tab or by selecting **Report** > **Measured to Reference Points** from the context menu.

See also:

Reports

2.4 External Services

2.4.1 Leica ConX

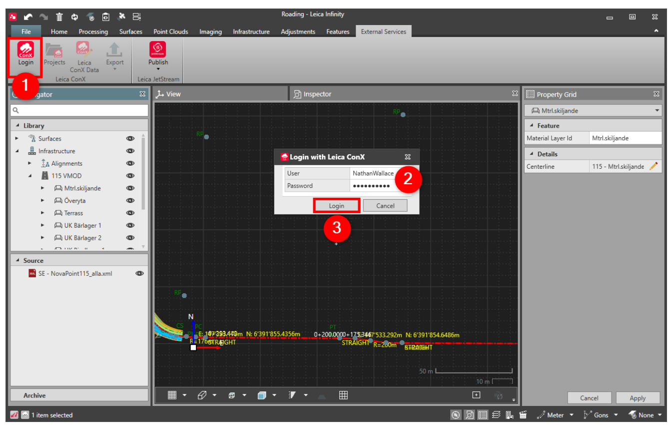
Leica ConX

With the Leica ConX service, you can log in to iCON Build/iCON Site projects and assign data to the project or directly to the units (that are the machines working on building sites).

Requirements:

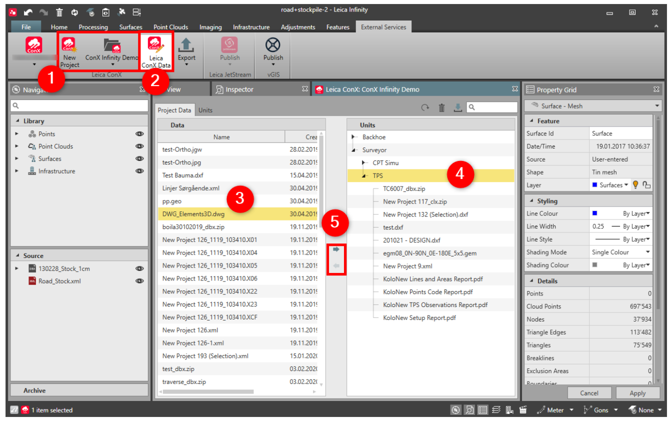
- Valid subscription.

To connect to the service:



1.	Select the **ConX Login** from the **External Services** tab.
2.	Enter the **User** and the **Password**. ☞ A default user can be set in Leica ConX.
3.	Select **Login**. ☞ The New Project option becomes active.

To assign/unassign Leica ConX data:



1.	Create a new project or select one of the available projects. ☞ The ConX user ID requires administrator rights to create new projects.
2.	Select **Leica ConX Data** to open the Leica ConX project viewer.
3.	Select the project data you want to assign or the units you want to reassign.
4.	Select the unit to which you want to assign.
5.	Select **assign** or **reassign**.

See also:

Leica ConX

Export to Leica ConX

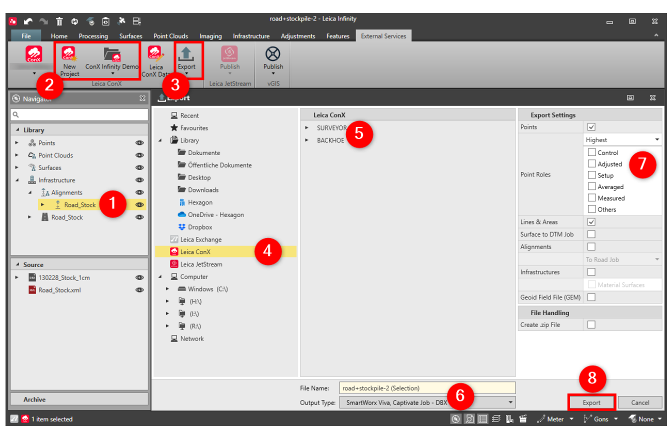
2.4.2 Export to Leica ConX

Export to Leica ConX

Requirements:

- Valid subscription.

To export data to an iCON project:



1.	In the Infinity project, select the data to export, for example the centreline of the road.

2. Create a new project or select one of the available projects.

   ☞ The ConX user ID requires administrator rights to create new projects.

3. Select **Export** and then **Selection** from the **External Services** tab.

   ☞ Alternatively you can select **Export** from the **Home** tab.

4. In the Export dialog, select **Leica ConX** in the tree view.
5. Optionally select a unit.
6. Define the File Name and Output Type.

   ☞ HeXML is the default.

7. Define the Export Settings.
8. Select **Export** .

   ☞ If selected, the HeXML file is written to the iCON project on the server and to the unit.

   ☞ See the data being assigned to the iCON project in the Leica ConX project viewer.

   ☞ The exported file is written to the Leica Infinity **Archive** > **Exported Files**.

See also:

Leica ConX

Leica ConX

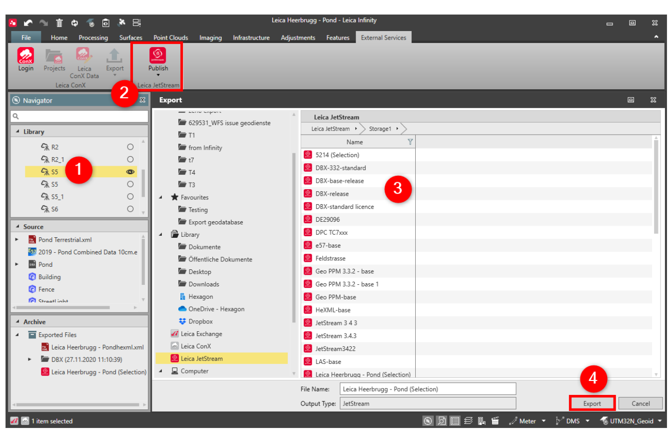
2.4.3 Publish to Leica JetStream

Publish to Leica JetStream

Infinity supports the Leica JetStream service and allows you to publish point clouds to an existing storage location.

☞ When exporting in .lgs format, point cloud includes:
MSxx/TLS data - Point cloud arrived from instruments and panoramas.
UAV/GS18 I data - Dense point cloud and images which were used for point cloud reconstruction.

To publish data:



1. In the Infinity project, select the point clouds for publication.

   ☞ If no point cloud is selected, all point clouds are exported.

2. Select **Publish** and then **Selection** from the **External Services** tab.

☞ Alternatively, select **Export** and then **Selection** from the **Home** tab.

3. In the Export dialog, select **Leica JetStream** in the tree view and go to the required storage location.

4. Select **Export**.

☞ The point clouds are stored in the defined storage location on the server.

☞ The exported file is written to the Leica Infinity **Archive** > **Exported Files**.

See also:

Leica JetStream

https://leica-geosystems.com/products/laser-scanners/software/leica-jet-stream

2.4.4 Publish to vGIS

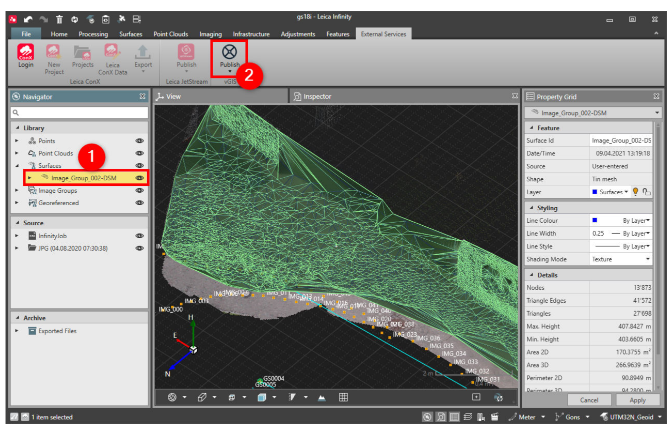
Publish to vGIS

Infinity supports the vGIS service and allows you to publish digital surface models in the vGIS augmented reality field application.

Requirements:

- Connection to vGIS.
- Project with digital surface model.

To publish data to vGIS:



1. In the Infinity project, select the digital surface model DSM that you want to publish.

2. Select **Publish** from under **vGIS** in the **External Services** tab.

☞ Alternatively you can choose Publish all.

See also:

vGIS

https://www.vgis.io/

2.4.5 Save as an ArcGIS Online Web Map

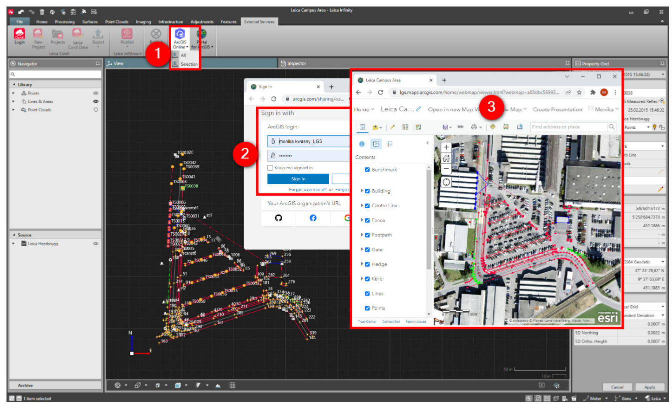
Save as an ArcGIS Online Web Map

With the save as an ArcGIS Online web map, it is possible to upload Infinity project data to ArcGIS Online.

☞ Either all library data or selected objects or reference files as .shp or .dxf/dwg can be exported.

Requirements:
- Logged in with ArcGIS Online.
- Subscription with rights to publish feature.
- Project data must have either WGS84 coordinates or coordinate systems attached.

To save the Infinity project as an ArcGIS Online web map:



1.	From within the open project, select the **Save as ArcGIS Online Web Map** option.
2.	Sign in to your ArcGIS Online account.
3.	The project data opens as a ArcGIS Online web map, with objects ordered by the code.

☞ A new web map is created, each time the icon is selected.

☞ Created web maps are accessible in ArcGIS Online/My Content for further edits/sharing.

See also:

Get Feature

Save as Portal for ArcGIS Web Map

Map Services

2.4.6 Save as Portal for ArcGIS Web Map

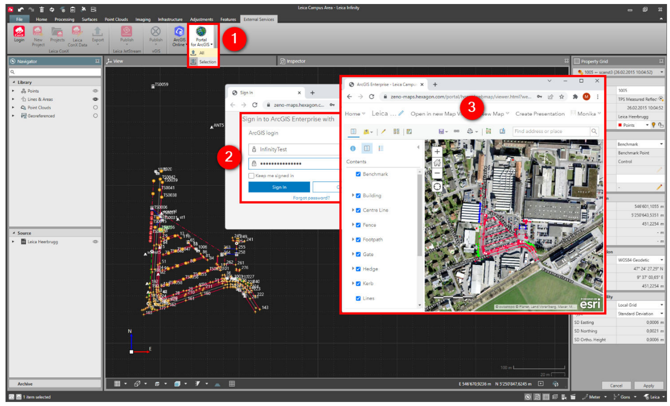
Save as Portal for ArcGIS Web Map

With the save as portal for ArcGIS, it is possible to upload Infinity project data to the ArcGIS portal.

☞ Either all library data or selected objects or reference files as .shp or .dxf/dwg can be exported.

Requirements:
- Logged in with Portal for ArcGIS.
- Subscription with rights to publish feature.
- Project data must have either WGS84 coordinates or coordinate systems attached.

To save the Infinity project as a new portal for the ArcGIS Web Map:



1.	From within the open project, select the **Save as Portal for ArcGIS** option.
2.	Sign in to your Portal for ArcGIS Online account.
3.	The project data opens as a Portal for ArcGIS Online web map, with objects ordered by the code.

☞ A new web map is created, each time the icon is selected.

☞ Created web maps are accessible in Portal for ArcGIS Online/My Content for further edits/sharing.

See also:

Get Feature

Save as an ArcGIS Online Web Map

Map Services

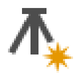
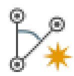
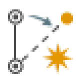
3 Optional Modules

3.1 TPS-Processing

3.1.1 Overview

TPS-Processing

The processing module allows you to:

	Create **new Points**. New points can also be created in the home module.
	Create a **new Station** on any point in your project. New stations can also be created in the Home module. Stations for which new observations have been entered, are identified as setups in the **TPS** tab of the Inspector.
	Create **new Observations** on any station in your project. New observations can also be created in the Home module.
	Create **new Traverses**. Only setups can serve as points in a traverse.
	Create **new Sets of Angles** on any setup in the project.

You can also:

- Edit setups, see Edit Setup Wizard.
- Edit traverses, see Traverse Wizard.
- Edit sets of angles, see Sets of Angles Wizard.

From inside the **TPS** tab in the Inspector.

3.1.2 New Point

New Point

In Infinity you can create points manually. By default, manually created points get the point role user-entered.

Creating points:

1.	Select the **New Point** option in the ribbon bar of the Home tab or the Processing tab.
2.	New points have to be defined in the point properties window. Enter a **Point ID** for the new point.
3.	Choose whether the new point shall get the Point Role *User-entered* or whether it shall become a *Control* point. ☞ You can always create control points from existing points from inside the point properties, too, by selecting the **Create Control** option next to the current point role.
4.	Enter the coordinates either **Local** or **WGS84**. If you have a coordinate system attached to the project, then the coordinates are automatically available in both systems.

5.	Optionally, enter the local **Position Quality** and select a **Code**. codes can only be selected if a code table is defined within or has been assigned to the project before.
6.	Select the **Create** option to create the point.

The new point is automatically added to the survey layer points.

See also:

Layer Manager

3.1.3 New Station

New Station

In Infinity you can create stations manually. When you define observations on a manually created station, the station becomes a setup with the orientation unknown.

Creating a station:

1.	Select **New Station** from the Home tab or the Processing tab.
2.	The new station has to be defined in the Property Grid: In the Station section, select the **Position Source**. Select the option on the right to search for and select a point from the fly-out list.
3.	Optionally, change the coordinates and/or attach a code.
4.	Select **Create** to create the station.

When you create a new station its orientation is unknown. It does not have any observations attached to it yet.

The new, manually entered station is added to:

- The Points section in the **Navigator** > **Library**.
- The Points section of the **Inspector** > **Points, Lines & Areas**.

Results:

You see that:

- With creating the first observation on a newly defined station, its point role changes to be a TPS setup with a still unknown orientation.

The new, manually created setup is added to:

- The navigator as manually entered in the source section.
- The inspector as a setup in the TPS tab.

3.1.4 New Observation

New Observation

In Infinity you can create stations manually. When you define observations on a manually created station, the station becomes a setup with the orientation unknown.

Creating observations on a new, manually entered station:

1.	Select New observation icon **New observation** from the Home tab or the Processing tab.
2.	New observations have to be defined in the Property Grid: Enter an ID for the new point that is to be defined by the New Observation.
3.	Select the station ID from which the New Observation aims at the new point. Select the pencil option on the right to search for and select a station from the fly-out list.
4.	Enter the observation details: Horizontal angle (Hz raw), Vertical angle (V), Slope Distance (SD) as well as the target height and type. ☞ Optionally, you can also enter offsets, if necessary, atmospheric and geometric corrections (ppm values) and codes.
5.	Select **Create** to create the observation.
6.	Repeat steps 1 to 5 until you have entered all observations for a station.

Results:

You see that:

- With creating the first observation on a newly defined station, its point role changes to be a TPS setup with a still unknown orientation.

The new, manually created setup is added to:

- The navigator as manually entered in the source section.
- The inspector as a setup in the TPS tab.

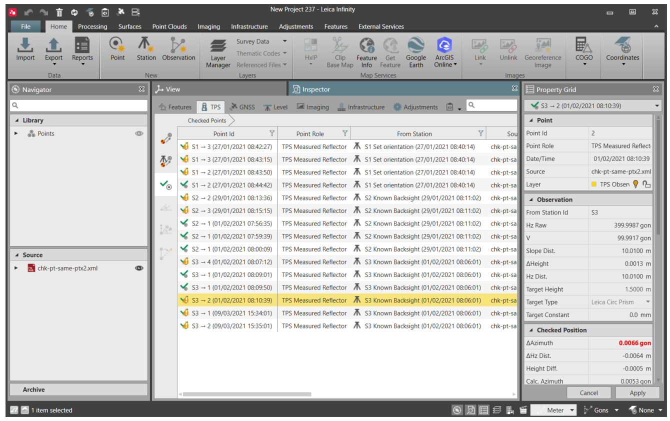
3.1.5 Checked Points

Checked Points

The checked points subtab lists all the points measured in the field with Captivate using the checked point application.

Points with at least one of the checked values out of tolerance, are indicated with an exclamation mark. You can find out more information about the out of tolerance values, by hovering the mouse over the icon with the exclamation mark.

Infinity can create Reports for the checked points application results.



The out of tolerance limits are defined in Checked Points.

☞ Currently, the checked point application results are not updated when the setup is updated and these observations cannot be modified.

See also:

Reports

3.1.6 Observation Properties

Observation Properties

In the property grid you can change the properties for single observations.

You can:

1. Edit the **Target Point ID**.
2. Select the option and select a different **Station ID**.
3. Edit the **Target Height** and select a different **Target Type**.
4. Edit the **Target Offsets**.
5. Select the option to re-calculate the **Atmospheric** and/or **Geometric PPM**.
6. Select a different **Target Point Code** from the drop-down list.
7. Set how the Local Position Quality shall be defined (choose between **Standard Deviations** and **Coordinate Quality**).

☞ To re-calculate a setup with another orientation method and using different setup observations make use of the edit setup wizard. To invoke the wizard from inside the Property Grid select the **Open Setup Wizard** option in the Orientation section.

Multi-edit

To edit the properties for one or more observations:

1. Select the observations to be edited either from inside the Source section of the Navigator or from inside the TPS tab or the Adjustments tab of the Inspector.
   In the TPS tab of the Inspector drill into **Setups** by selecting the little arrow. Drill further into a **Station** to see all connected observations and select the observations to be edited.

   In the Adjustments tab of the Inspector go to the **Observations** section and select the observations to be edited.
2. Right-click into the selection and select the **Edit** operation you want to perform from the context menu.

You can select more than one observation at a time. The edit operation is executed for all selected observations/target points.

3.1.7 Setup Properties

Setup Properties

Some setup properties can be edited in the property grid.

Depending on the setup method you can:

1. Select the option and select a different **Station Source**.
2. Edit the **Instrument Height**.
3. Edit the **Station Coordinates**.
4. Edit or select the option to compute an **Azimuth**.
5. Select the option and select a different **Target Point**.
6. Edit the **Target Height**.
7. If a station scale has been applied, select the option to remove the **Station Scale** from all observations to which it has been applied before.

☞ To re-calculate the setup with another orientation method and using different setup observations make use of the edit setup wizard. To invoke the wizard from inside the Property Grid select the **Open Setup Wizard** option in the Orientation section.

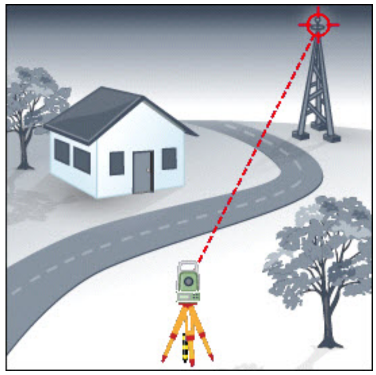
☞ When TPS setups are edited in Infinity then all scan data connected to that setup is shifted and/or rotated accordingly.

3.1.8 Setup Methods

Setup Methods

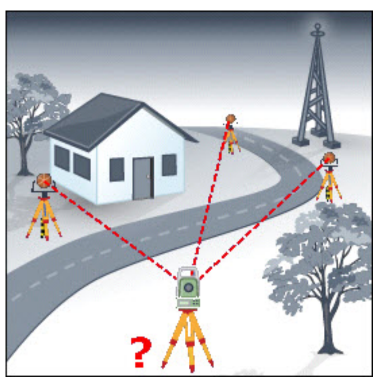
There are several methods how a setup can be calculated.

These are:

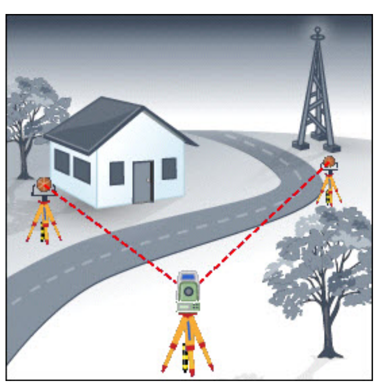


Set Orientation:
Instrument was set up on a known point and oriented to a known azimuth.

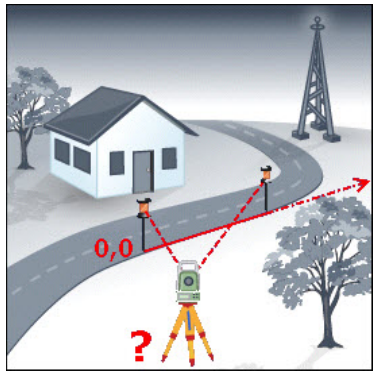
Known Backsight:
Instrument was set up on a known point and oriented to a known backsight point.



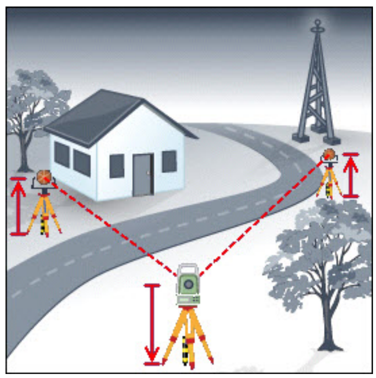
Resection/Resection Helmert:
Instrument was set up on an unknown point. Station coordinates and the orientation were calculated by measuring to known target points.



Multiple Backsights:
Instrument was set up on a known point. The Orientation and/or height were calculated by measuring to known target points.



Local Resection:
Station coordinates and Orientation were derived from measuring to two points which define a local coordinate system.



Height Transfer:
Instrument was set up on a known point. The height was calculated by measuring to known target points.

3.1.9 Edit Station Setup

3.1.9.1 Overview

Edit Setup Wizard

The edit setup wizard allows you to recalculate setups defined in a project.

It automatically detects the setup method that has been used in the field and suggests a station point source.

See also:

Setup Methods

To edit setups:

Select the setup to be edited either from the Source section of the Navigator or from inside the TPS tab of the Inspector.

In the Navigator, right-click on the setup to be edited and select **Open Setup Wizard** from the context menu.

In the Inspector, drill down into the Setups, select the setup to be edited and select the option.

Once selected you can also select the **Open Setup Wizard** option in the Orientation section of the Setup Property Grid.

The edit setup wizard starts with, Edit Setup Wizard: Station and Method.

See also:

The tutorial **"How to manage TPS setups and data edits"** https://leica-geo-systems.com/-/media/6d5180b3d45d4102ace367b9dc6309ad.ashx

☞ Tutorial data can be downloaded in the Localisation Tool.

3.1.9.2 Edit Setup Wizard: Station and Method

Edit Setup Wizard: Station and Method

A common use for the edit setup wizard is to update station coordinates and/or the orientation.

1. Check whether the setup method itself can be left as set in the field data or whether it needs to be changed.
2. For Orientation methods that use a known station point, check the source of your station point.
3. Select the ✏ icon to select another Station Point Source from the flyout list and then **OK** to confirm your selection.
4. Adapt the instrument height if necessary.
5. Select **Next** to proceed with, Edit Setup Wizard: Setup Observations.

Orientation methods using known station points are:

- Set Orientation
- Known/Multiple Backsights
- Height Transfer

Stations to be calculated in a Resection are always set up on unknown points. For resections the station ID can be changed but you cannot select a different station source. You can only choose to re-calculate the resection by using different target points in the next wizard step.

3.1.9.3 Edit Setup Wizard: Setup Observations

Edit Setup Wizard: Setup Observations

1. On the selected station point ➡ select the observations that shall be used for recalculating the orientation.
   For method **Set orientation**:
   Select an observation that shall serve as the known orientation. All measurements on this station point are recalculated with reference to the selected orientation.
   For method **Known/Multiple Backsight**:
   Select a point/points that shall serve as a known/multiple backsight points. The orientation to the selected points is taken for recalculation.
   For method **Resection/Resection Helmert**:
   Select the observations to points that shall serve as targets in the resection calculation.
   For method **Local Resection**:
   Select the observations to two points that shall define the line to which your station shall be oriented.
   For method **Height Transfer**:
   Select the observations to known target points from which the height shall be transferred to the station point.
2. Select **Next** to proceed with, Edit Setup Wizard: Compute Setup.

☞ It is important to differentiate between the selection of target points and the selection of observations to the target points. The target points are identified in Step 3. In this step of the wizard (Step 2), you choose the observation information. If there exists, for example resulting from a sets of angles calculation, more than one observation to a point, select only the one observation to be used. In the case of a sets of angles existing on a station point, this would be the reduced measurement to the target point.

3.1.9.4 Edit Setup Wizard: Compute Setup

Edit Setup Wizard: Compute Setup

For all target observations, selected in the Step 2: Setup Observations, the target points are listed in the target info content view. But, if a target point disposes of more than one point role, then by default always the role from the top of the hierarchy is selected.

Thus:

1. Select the option to select the target point role needed for recalculating the setup from a flyout list. Leave the flyout with **OK** to take over your selection.
2. When you are recalculating a Resection choose from the Parameters section:
   - The Adjustment Method: Least Squares or Robust.
   - Whether Easting, Northing, Height and Orientation shall be calculated or just the position and the orientation or just the station coordinates.
   - Whether a Scale shall be computed.

   If you decide to compute the scale factor then you can decide further down under Station Point whether the scale shall be **Applied to all observations** that have possibly been measured on the station point of the resection.

   ☞ To remove the scale, select the option under Station Scale in the Setup Properties window.

In the station point section you can review:

- The azimuth, calculated with respect to the chosen target points and observations.
- The calculated orientation correction.
- The station coordinates.

For resections, the standard deviations for the Azimuth and the station coordinates are given as well.

In the tolerances section:

3. Define the limits for how much the measured target point position and height are allowed to differ from the target point position and height after recalculation.
4. For resections also define an orientation limit, as in a limit for how much the Hz observations to each target point are allowed to differ from the calculated values.

5. Decide whether you want to ☑ run a **tolerance check** for the position and/or height and/or orientation of the target points using the given limits.

When you decide to check whether the target points calculated from the recomputed station point lie within the defined tolerances you see that outliers are marked by ❗ and the Δ values are written in red in the **Target Info** content view.

6. Exclude outliers from use in the setup computation by double-clicking slowly onto a target points **Use** field and selecting **None** or use just Position or just Height from the drop-down list. All computation results are recalculated and shown to you instantly.

7. Select **Finish**.

Before the computation results are taken over into your project you get a ⚠ warning message that the coordinates and orientation of the recalculated setup are to change by the given Δ values.

8. Select **OK** to update the setup in your project.

3.1.10 Create or Edit Traverse

3.1.10.1 Overview

Traverse Wizard

The traverse wizard allows you to build up a traverse from the setups defined in a project. You can also edit existing traverses. It automatically detects which setups are connected by back and foresights and suggests a sequence once you have selected the start point.

It also detects the traverse technique from the selected combination of setups and suggests the start and end point as well as the initial backsight and final foresight.

☞ Not all kinds of traverses are defined by the classic setup of a start and an end point and an initial backsight and final foresight.

To create a new traverse:

Go to the **Processing** tab and select **Traverse** from the ribbon bar.

The Create new traverse wizard starts with:

Traverse Wizard: Extract Traverse

Go through the wizard to create a new traverse.

To edit an existing traverse:

Select the traverse to be edited either from the Library section of the Navigator or from inside the TPS tab of the Inspector.

In the Navigator, right-click on the traverse to be edited and select **Open Traverse Wizard** from the context menu.

In the Inspector, move down into the traverse applications, select the traverse to be edited and select the ✎ option.

Once selected you can also select the ✎ **Open Traverse Wizard** option in the Results section of the Property Grid.

The edit traverse wizard starts with:

Traverse Wizard: Extract Traverse

Go through the wizard to edit an existing traverse.

See also:

The tutorial **"How to process a traverse"** https://leica-geosys-tems.com/-/media/d70cbaceb3ea49988bbdf75a92ac0ef3.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

3.1.10.2 Traverse Wizard: Extract Traverse

Traverse Wizard: Extract Traverse

1.	Identify the new traverse by a unique **ID** (name).
2.	Choose the traverse start point from the **available setups** and ➡ **add** it to the new traverse. The next possible setups in the traverse are detected automatically and suggested for selection.
3.	If the next setup in the traverse can be identified uniquely press ➡ **add** again to add the next point. If there is a choice select the next traverse point **manually** and press ➡ **add**.
4.	Repeat step 3 until the traverse is complete.
5.	Under **Traverse Technique** check that the start and end point coordinates are identified correctly. If **Control** coordinates are available these coordinates should be taken. If there is a choice for the initial backsight and/or the final foresight, then choose the correct coordinates to be used.
6.	Optional: If reduced measurements are available for one or more of the traverse points from the **Sets of Angles** or **Measure Foresight** applications. And if you want the reduced measurements to be taken for the traverse calculation then ☑ check this option at the bottom of the traverse wizard page.
7.	Select **Next** to proceed with: Traverse Wizard: Observation Review

3.1.10.3 Traverse Wizard: Observation Review

Traverse Wizard: Observation Review

1.	Review the angle observations and measured distances. If you decide to ☑ **Use tolerances** outliers will be marked by: ❗ You can adjust the tolerances to your needs either for the current computation run in the same wizard page or globally under Info and **Settings** > **Traverse Processing Parameters**. **See also:** Traverse Wizard: Processing Parameters
2.	You can de-select single observations which you do not want to take part in the computation. But at least one backsight and one foresight observation must remain on each setup, otherwise the traverse gets broken up and cannot be calculated. The **Next** button becomes inactive.

3. Select **Next** to proceed with:
Traverse Wizard: Processing Parameters

3.1.10.4 Traverse Wizard: Processing Parameters

Traverse Wizard: Processing Parameters

Configure the traverse processing parameters for the current processing run.

☞ On how to configure the default parameters, refer to Traverse Processing Parameters.

Methods: Compass Rule, Transit Rule:

1. Select the Adjustment Method by which the traverse shall be adjusted. By default the Compass Rule is selected.
2. Define the Standard Error per Angle and Distance that shall be used as the coefficients to calculate the Max. Angular and the Max. Length Errors. By default these values are set to 15'' and 0.01m.
3. Depending on the required level of accuracy adapt the Min. 2D Accuracy, as in, the minimum required position accuracy. By default this value is set to be 1/10'000.
4. Define the Max. allowed Height Error per Traverse Point. By default this value is set to be 0.01m.
5. Depending on the required level of accuracy adapt the Min. 1D Accuracy, as in, the minimum required height accuracy. By default this value is set to be 1/10'000.
6. Define the maximum allowed Station Difference before and after Adjustment. By default these values are set to be 0.01m for Easting, Northing and Height.
7. Select to compute the misclosure, traverse length and accuracies to the end FS.
   Select the desired point to compute the misclosure to. An angular misclosure cannot be computed or distributed when this option is selected. The misclosure is computed to the end station, if this option is unselected.
8. Select to apply curvature and refraction correction to the observations.
9. Select to use forward only observations. Forward and backward observations are mesured for each traverse leg, if this option is unselected.
10. Select **Next** to proceed with Test Criteria.

☞ If you deselect any of the tolerance checks or the maximum station difference check, then these checks are not applied in the traverse calculation. By default all checks are switched on.

Method: Least Squares:

1. Select the Adjustment Method by which the traverse shall be adjusted. In this case, select **Least Squares**.
2. Define the Adjustment Dimension. By default this is set to 3D.

3. To define the settings refer to the adjustment help topics:
   - General Adjustment Settings
   - Test Criteria
   - Advanced Terrestrial Parameters

4. Select **Next** to proceed with Traverse Wizard: Traverse Result.

☞ GNSS settings and level settings in the adjustment help are not applicable to the traverse wizard.

3.1.10.5 Traverse Wizard: Calculate Traverse

Traverse Wizard: Calculate Traverse

Before the traverse is adjusted you are shown its accuracy before adjustment. If you select to distribute the angular misclosure (equally or by distance), the angle distributed misclosure is shown.

You see:

- Its computed total length.
- Its computed 1D and 2D accuracies. If the tolerance values defined in step 3 are exceeded a warning (!) is issued.
- Its length and direction of error. This corresponds to the vector resulting from the computed length and cross errors.
- Its computed start azimuth. You can force a different value to be taken when you select **Use Start Azimuth** and enter a different azimuth value. By default this setting is switched off.
- Its computed scale.
- The misclosure point.
- Its computed coordinate misclosures. In the components where the max. allowed station differences as defined in Step 3 are exceeded a warning (!) is issued.
- Its computed end azimuth. You can force a different value to be taken when you select **Use End Azimuth** and enter a different azimuth value. By default this setting is switched off.

Select a method for misclosure distribution:

Angular misclosure:

1. The maximum allowed angular error, as computed according to the number of stations and the standard error per angle defined in Step 3, is compared with the calculated angular error. If the maximum allowed error is exceeded a warning (!) is issued.

2. Choose the method how the angular misclosure shall be distributed.
   - If you select **Equally**, the angular misclosure is divided by the number of traverse angles and the same correction is applied to each setup.
   - If you select **By Distance**, the angular misclosure is distributed with respect to the length of the traverse legs. The shorter a traverse leg is, the bigger the correction is.
   - If you select **No Distribution**, the angular misclosure is not distributed to the traverse angles.

3. The misclosure values that are shown on this page are determined with the angular misclosure method that is selected. Any coordinate misclosure distribution performed through the compass rule or transit rule are based on these values.

Length misclosure:	
4.	The maximum allowed length error, as computed according to the total traverse length and the standard error per distance defined in Step 3, is compared with the calculated length error. If the maximum allowed error is exceeded a warning ❗ is issued.

Height misclosure:	
5.	The maximum allowed height error per traverse point, as defined in Step 3, is compared with the calculated height error. If the maximum allowed error is exceeded a warning ❗ is issued.
6.	Choose the method how the height misclosure shall be distributed. • If you select **Equally** the height misclosure is divided by the number of stations and the same correction is applied to each station height. • If you select **By Distance** the height misclosure is distributed with respect to the length of the traverse legs. The longer a traverse leg is, the bigger the correction is. • If you select **No Distribution** the height misclosure is not distributed to the station heights.
7.	Select **Next** to proceed with the Traverse Wizard: Traverse Result.
☞	The calculate traverse section of the traverse wizard is skipped when adjusting using the least squares method.

3.1.10.6 Traverse Wizard: Traverse Result

Traverse Wizard: Traverse Result

Before you take over the computation results to your project, the calculation results are listed.

Method: Compass Rule, Transit Rule

You see:

- The total traverse length.
- The number of stations included.
- The achieved 1D and 2D accuracy (if the misclosure is zero these fields are dashed).
- The misclosure point.
- The misclosure values after adjustment.
- The calculated scale.
  Select **Apply scale to observations** if you want to apply the scale factor resulting from the traverse computation to all setups that make up the traverse. For each setup, all observations are scaled with this value.
- The kind of error distribution (balancing) you have chosen for angles and height.
- Transformation values applied as a result of point resourcing (based on start station and initial azimuth).
- The adjusted results calculated for the traverse point coordinates using the control coordinates at the start (and the end) of the traverse plus the Δ values in position and height relative to the coordinates resulting internally from the measurements.
  How large the Δ values are depends on how you choose to distribute the errors in Step 4: Calculate Traverse.

To apply the results and update the traverse point coordinates in your project:

Select **Finish**.

The traverse points get the ⊕ adjusted coordinates added as another point role.

Method: Least Squares

You see:

- **Least squares results:**
  - F-test results and associated critical values.
  - W-test results and associated critical values.
  - Chi-Square test results and associated critical values.
  - 2D and 1D confidence levels.
  - Adjustment dimension.
- Adjusted Points tab.
- Absolute Error Ellipses tab.
- Relative Error Ellipses tab.
- Adjusted Observations tab.

To apply the results and update the traverse point coordinates in your project:

Select **Finish**.

The traverse points get the ⊕ adjusted coordinates added as another point role.

Least squares method - Additional considerations

- Flagged W-test observations can be removed from the observation review section of the traverse wizard.
- Refer to the following help articles for the least squares result interpretation:
  - Statistical Testing
  - F-Test
  - Chi-Square Test
  - W-Test
  - T-Test
  - Interpreting Test Results

☞ The ⊕ adjusted coordinates should be taken as target coordinates when recalculating setups on sideshot points of the traverse.

3.1.10.7 Traverse Processing Parameters

Traverse Processing Parameters

In the file tab, under info & settings you can define the traverse processing parameters that shall be used by default when a new traverse is calculated in Infinity.

See also:

Info & Settings

Default Traverse Adjustment Parameters

Adjustment method:

Choose the method of how the coordinate misclosure (Easting, Northing) shall be distributed.

- If you select **Compass Rule**, the coordinate misclosure is distributed relating to the length of the traverse legs. The Compass Rule assumes that the biggest error comes from the longest traverse observations. This method is suitable when the precision of the angles and distances are approximately equal.
- If you select **Transit Rule**, the coordinate misclosure is distributed relating to the coordinate changes in Easting and Northing. Use this method if the angles were measured with a higher precision than the distances.
- If you select **No Adjustment**, the coordinate misclosures are not distributed to the station coordinates.
- If you select **2D Helmert**, the traverse is adjusted with a 2D Helmert transformation. Shift, rotation and scale factor is computed and applied to the traverse.
- If you select **Least Squares**, the traverse is adjusted with a constrained 2D or 3D least squares adjustment.

☞ The compass rule is also known as the Bowditch method.

☞ Control coordinates have to be stored for the start and end point in the traverse if you want to calculate and distribute the coordinate misclosure by either the compass rule or the transit rule.

Methods: Compass Rule, Transit Rule

Angle Balance

Choose the method of how the angular misclosure shall be distributed.

- If you select **Equally**, the angular misclosure is divided by the number of traverse angles and the same correction is applied to each setup.
- If you select **By Distance**, the angular misclosure is distributed relating to the length of the traverse legs. The shorter a traverse leg is, the bigger the correction is.
- If you select **No Distribution**, the angular misclosure is not distributed to the traverse angles.
- The misclosure values that are computed are determined with the angular misclosure method that is selected. Any coordinate misclosure distribution performed through the compass rule or transit rule are based on these values.

Height Balance

Choose the method of how the height misclosure shall be distributed.

- If you select **Equally**, the height misclosure is divided by the number of stations and the same correction is applied to each station height.
- If you select **By Distance**, the height misclosure is distributed relating to the length of the traverse legs. The longer a traverse leg is, the bigger the correction is.
- If you select **No Distribution**, the height misclosure is not distributed to the station heights.

Apply scale to observations

Select this setting, if you want to apply the scale factor resulting from the traverse computation to all setups that make up the traverse. For each setup, all observations are scaled with this value.

☞ When you recompute the same traverse then the scale is 1.0 or very close to it.

☞ To remove the scale select the 🗑 option, under Station Scale in the Traverse Properties window.
If you delete the traverse the scale is also deleted from the setups and observations to which it has been applied.

Tolerance Checks in Traverse Wizard

Observation Review

Residuals Hz/V

Specify a limit for the value that the residuals of your angular observations are allowed to take. When the residuals exceed this limit then the observations are marked with a ❗ in the Observation Review of the New Traverse wizard.

Residuals Slope Distance

Specify a limit for the value that the residuals of your slope distance measurements are allowed to take. When the residuals exceed this limit then the measurements are marked with a ❗ in the Observation Review of the New Traverse wizard.

Distance Difference

Specify a limit for the difference that shall be allowed between distances measured as a foresight and a backsight between two traverse stations.

Δ Height Difference

Specify a limit for the difference that shall be allowed between height differences measured as a foresight and a backsight between two traverse stations.

Processing Parameters - Compass Rule, Transit Rule

Use max. 2D error

Max. Angular Error:

The tolerance for the angular error is defined as **F = k*sqrt(n)**, with **k** being the constant value that you have to enter as the Standard Error per Angle (the default value is 15'') and **n** being the number of traverse points (angles). If the calculated angular error exceeds the Max. Angular Error, the angular misclosure is marked with a ❗ in the calculate traverse step of the traverse wizard.

Max. Length Error:

The tolerance for the length error is defined as **F = k*sqrt(L)**, with **k** being the constant value that you have to enter as the Standard Error per Distance (the default value is 0.010m) and **L** being the total length of the traverse. If the calculated length error exceeds the Max. Length Error, the length misclosure is marked with a ❗ in the calculate traverse step of the traverse wizard.

☞ If you choose to adjust the traverse by a 2D Helmert transformation it is not necessary to define a maximum length error and a maximum cross error. The corresponding functionality is not available. The scale factor is automatically applied as part of the transformation.

Min. 2D Accuracy:

The accuracy in position is achieved by dividing the calculated length of error by the total traverse length, with the length of error being the length of the vector resulting from taking the length and the cross error into account. The minimum position accuracy that should be achieved is defined as 1 divided by a user-defined value. By default this value is set to be 10'000. If the achieved accuracy is bigger than the defined fraction value then the Min. 2D Accuracy is marked with a ❗ as exceeded.

Use max. 1D error

Max. Height Error per Traverse Point:

Define the tolerance for the height misclosure. If the calculated height misclosure divided by the number of stations exceeds the Max. Height Error per Station, the height misclosure is marked with a ❗ in the calculate traverse step of the traverse wizard.

Min. 1D Accuracy:

The accuracy in height is achieved by dividing the calculated height error by the total traverse length. The minimum height accuracy that should be achieved is defined as 1 divided by a user-defined value. By default this value is set to be 10'000. If the achieved accuracy is bigger than the defined fraction value then the Min. 1D Accuracy is marked with a ❗ as exceeded.

Station Difference before and after Adjustment

Use station difference

Max. Station Difference before and after Adjustment:

Define the maximum difference that shall be allowed between measured and adjusted station coordinates. If the adjusted station coordinates differ by more than the defined values from the measured coordinates then the coordinate misclosure is marked with a ❗ in the calculate traverse step of the traverse wizard.

☞ If you deselect any of the tolerance checks or the max. station difference check, then these checks are by default not applied in the new traverse wizard. But you can always select or deselect single checks while calculating a traverse in the wizard independent of the default settings.

Processing Parameters - Least squares method

Refer to the following adjustment help topics:

- General Adjustment Settings
- Test Criteria
- Advanced Terrestrial Parameters

Least squares method - Processing considerations:

- GNSS settings and Level settings in the adjustment help topics are not applicable to the traverse wizard.
- The least squares method does not perform a transformation. If this is required, you can apply a transformation through the Shift, Rotate, Scale wizard.
- Flagged observations in the observation review section of the traverse wizard are a good early indicator of a failing F-test.
- Flagged W-test observations can be removed from the observation review.

3.1.11 Create or edit Sets of Angles

3.1.11.1 Overview

Sets of Angles Wizard

The sets of angles wizard reduces a sequence of TPS observations from a common TPS setup.

The result also includes a reduced observation available to use for a traverse application.

The sets of angles are reduced to the first target, which can be different than the backsight target of the selected station setup.

The reduction to the station setup results in an orientation correction applied from the backsight to the first target used in the sets of angles. Only one orientation correction can exist per station setup.

To create new sets of angles:

Go to the **Processing** tab and select **Sets** from the ribbon bar.

The new sets of angles wizard starts with:

Sets of Angles Wizard: Select Setup and Points

Go through the wizard to create a new sets of angles.

To edit existing sets of angles:

Select the sets of angles to be edited from inside the TPS tab of the Inspector.

Drill down into the Sets of Angles Applications, select the station on which the sets to be edited is defined and select the option.

Once selected you can also select the **Open Sets of Angles** option in the Results section of the Sets of Angles Property Grid.

The edit sets of angles wizard starts with:

Sets of Angles Wizard: Select Setup and Points

Go through the wizard to edit an existing sets of angles.

3.1.11.2 Sets of Angles Wizard: Select Setup and Points

Sets of Angles Wizard: Select Setup and Points

1. On the left side select the **Setup** upon which the measurements have been taken, that shall be used for the sets of angles computation.

2. Choose whether you want to compute the **maximum number of sets** or the **maximum number of points**.

   ☞ When you choose to compute the maximum number of sets, then the maximum number of sets that have been measured on the selected setup for either Face I or for Face I & II are detected, but only those points are offered for selection as available targets which have been measured in all sets.
   Points that have been skipped in one of the sets, are not offered for selection. Thus, the calculation method is called without skipped points.

   ☞ When you choose to compute the maximum number of points, then all points that have been measured on the selected setup for either Face I or for Face I & II are detected and offered for selection as available targets, independent of how many sets can be computed in relation to the current selection.
   Sets that have been measured excluding one or more of the selected target points are incomplete and thus ignored. The calculation method is called without incomplete sets.

3. Select either observation type **Face I & II** or **Face I**.

4. Select the **Available Targets** and select the ➡ option to add all related measurements to the right side.

5. Select **Next** to proceed with the Sets of Angles Wizard: Sets Review.

3.1.11.3 Sets of Angles Wizard: Sets Review

Sets of Angles Wizard: Sets Review

In the sets review all used target points are listed.

1. Review the angle observations and measured distances for each target point.

   ☞ When in Step 2 you chose to compute the maximum number of points without using incomplete sets, all those instances of a point, are unavailable for selection that are part of incomplete sets.

2. Depending on the Tolerance settings, outliers are marked by ❗. You can adjust the tolerances to your needs either for the current computation run in the same wizard page or globally under **Info & Settings > Averaging & Reduction**.

   ☞ The tolerance settings can also be changed for single targets once the sets of angles computation is finished. To do so select the ✏ option in the Observation section of the Sets of Angles Target Properties window. The tolerances shown correspond to those last used but can be changed for single targets if necessary.

3. Exclude outliers from the computation by de-selecting them.

☞ When you de-select a target point from one of the sets then the set becomes incomplete and thus be de-activated for all points. At least one set has to remain active to be able to compute reduced measurements.

4. Select **Next** to proceed with, Sets of Angles Wizard: Sets of Angles Results.

3.1.11.4 Sets of Angles Wizard: Sets of Angles Results

Sets of Angles Wizard: Sets of Angles Results

In the errors section the resulting standard deviations are listed for:

- A single Hz direction in a single set.
- A single V angle in a single set.
- A single slope distance in a single set.

and for:

- The reduced Hz direction.
- The reduced vertical angle.
- The reduced slope distance.

With the standard deviations for the reduced observations being the standard deviation for a single observation divided by the square root of the number of sets.

In the targets section each target point is listed with:

- Its reduced values for Hz, V, slope and horizontal distances.
- Its local grid coordinates resulting from the sets of angles computation.
- The standard deviation calculated from the correction values for each point in all sets.

3.1.12 Reduce Foresights

3.1.12.1 Overview

Reduce Foresights Wizard

The reduce foresights wizard reduces a sequence of TPS observations from a common TPS setup.

Each foresights application is using the station setup backsight target point as the starting reference to compute the foresight targets. This is different to the sets of angles application which references the first target to reference the reduced angles.

The reduced foresights also result with a reduced observation.

☞ Reduced Foresights reduced measurements are not available to use for the traverse application.
Delete the reduce foresights applications from the **Inspector** > **Processing** > **TPS** application list and then create a traverse.

To create new reduced foresight:

Go to the **Processing** tab and select **Foresights** from the ribbon bar.

The wizard starts with:

Reduce Foresights Wizard: Select Setup and Points

Finish the wizard to create a new reduced foresight app that you can run a report on.

To edit existing reduced foresights:

Select the target to be edited from inside the **TPS** tab of the Inspector.

Drill down into the Foresights Applications, select the station on which the sets to be edited is defined and select the option.

Once selected you can also select the **Open Reduced Foresights** option in the Results section of the Foresights Property Grid.

The edit reduced foresights wizard starts with:

Reduce Foresights Wizard: Select Setup and Points

Go through the wizard to edit existing foresights.

3.1.12.2 Reduce Foresights Wizard: Select Setup and Points

Reduce Foresights Wizard: Select Setup and Points

1. On the left side, select **Setup** upon which the measurements that have been taken, shall be used for the foresights computation.
2. Select either **observation type** Face I & II or just Face I.
3. Select **Available Foresights** and select the option to add all related measurements to the right side.
4. Select **Next** to proceed with, Reduce Foresights Wizard: Foresights Review.

3.1.12.3 Reduce Foresights Wizard: Foresights Review

Reduce Foresights Wizard: Foresights Review

In the foresights review, all used target points are listed.

1. Review the angle observations and measured distances for each target point.

   ☞ When in Step 2 you chose to compute the maximum number of points without using incomplete sets, all those instances of a point, are unavailable for selection, that are part of incomplete sets.

2. Depending on the tolerance settings, outliers are marked by ❗. You can adjust the tolerances to your needs either for the current computation run in the same wizard page or globally under **Info & Settings** > **Averaging & Reduction**.

   ☞ The tolerance settings can also be changed for single targets once the foresights computation is finished. To do so select the option in the Observation section of the Foresights Target Properties window. The tolerances shown correspond to those last used, but can be changed for single targets if necessary.

3. Exclude outliers from the computation by de-selecting them.

   ☞ When you de-select a target point from one of the sets then the set becomes incomplete and thus be de-activated for all points. At least one set has to remain active to be able to compute reduced measurements.

4. Select **Next** to proceed with the Reduce Foresights Wizard: Results.

3.1.12.4 Reduce Foresights Wizard: Results

Reduce Foresights Wizard: Results

In the errors section the resulting standard deviations are listed for:

- A single Hz direction in a single set.
- A single V angle in a single set.
- A single slope distance in a single set.

and for

- The reduced Hz direction.
- The reduced vertical angle.
- The reduced slope distance.

With the standard deviations for the reduced observations being the standard deviation for a single observation divided by the square root of the number of sets.

In the targets section each target point is listed with:

- Its reduced values for Hz, V, slope and horizontal distances.
- Its local grid coordinates resulting from the foresights computation.
- The standard deviation calculated from the correction values for each point in all sets.

3.1.13 Update Stations

3.1.13.1 Overview

Update Stations

The update stations wizard enables you to recalculate multiple setups when new coordinates are available for your stations and target points.

For setup method unknown, local resection, height transfer, orient to object and adjusted traverse the update stations is not available. All computations are run with the current individual setup settings.

To update stations, go to the **Processing** tab and select **Update Stations** from the ribbon bar.

The Update Stations wizard opens.

3.1.13.2 Update Stations Wizard

Update Stations Wizard: Select Stations

In the Select Stations tab, all available or selected stations are displayed on the left side.

☞ By default, stations are sorted by time and date. You can use filters and search to find required stations.

1. Select the option to add across selected stations or select the option to add all stations.
2. Select the option to remove a selected station from the list of setups. Select the option to remove all stations from the list of setups.
3. Select **Next** to progress to the Update Stations Wizard: Review & Update Stations.

Update Stations Wizard: Review & Update Stations

The Review & Update Stations tab shows computed setups with the highest class for the station and targets available in the project.

1.	Review suggested point source for stations.
2.	Review suggested point source for targets by selecting the ▸ **Expander** option next to the station.
3.	Select the ✎ **Edit** option to select different point sources for the computation.
4.	Disable the ☑ **Update** checkbox if the station should not be updated.
5.	Select the ☑ **Apply scale to observations** checkbox to apply pre-computed scale to the observations.
6.	By default, a report runs and opens when you select **Finish**. ☞ Deselect the ☑ **Show Report** checkbox to hide the report .
7.	Select **Finish** to start the operation.

3.2 GNSS-Processing

3.2.1 Overview

GNSS-Processing

GNSS post-processing considers the three influences of baseline processing to arrive at the most reliable and accurate coordinates.

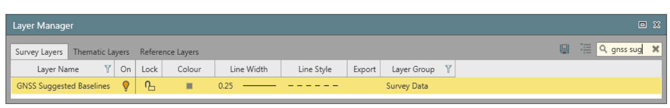
These three influences are:

- Defining what data the processing engine should consider.
- Selecting the antenna calibration set to minimise errors in the position solution.
- Setting the processing strategy to be applied to the data.

You can define the baseline to process in two ways:

- Manually: Setting the reference and rover station.
- Automatically: Allowing the application to find the possible baseline combination, according to the constrains defined in the auto settings.

In both configurations, the graphical view helps you to visualise (to have a preview) of the possible baselines through the GNSS Suggested Baselines layer.



The data processing and the automatic baselines selection are defined by the following project settings:

Data:

Allow you to filter or set which data is sent to the processing engine.

This can be:

- Choosing which constellations the processing should include to the elevation angle of observation data.
- The ephemeris type or even the observation rate.
- A key data setting is the antenna calibration set, that is considered during processing. Ensuring that both the reference and the rover stations are referring to the same calibration model of determining POC and PCV values ensures minimising the physical error sources to achieve the most reliable solution.

Strategy:

Allows you to set parameters to influence the baseline computation based on:

- The length of baseline.
- Common observation times.
- The atmospheric influences.

Auto Settings:

The automatic processing settings define the logic to identify intelligently pairs of reference and rover stations, building up all possible combinations of baselines.

The rules to create a baseline consider the baselines duration (amount of time of simultaneous measurement at reference and rover station) and the baseline length.

The baseline processing order is defined according to the points role, for example if control points are available, they are used as a starting point for the baselines automatic processing, and the automatic processing settings defined by you.

Advanced:

Allows you to tune the processing through specific settings, such as the frequency to use in the computation.

It is recommended to leave these settings at system defaults in order that the processing engine determines the highest reliability of the solution.

See also:

The tutorial **"How to process GNSS Baselines"** https://leica-geosystems.com/-/media/c614b534942141c8952e7fa8973bc7e2.ashx

The tutorial **"Advanced GNSS Processing"** https://leica-geosystems.com/-/media/539597242bfa46b99d1366acae90d133.ashx

The tutorial **"Ambiguity Statistics Interpretation"** https://leica-geosystems.com/-/media/41ee41837f7442bfa42d4b746ed088c0.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

3.2.2 How to Post-Process GNSS Data?

How to Post-Process GNSS Data?

Setting up your project

Setting up your project to post-process GNSS data. Import data to a project and adapt the processing parameters if necessary.

1. Import the GNSS raw data to your project.
See also:
New Project
Data Import
☞ GNSS raw data is only supported in the DBX (SmartWorx) or RINEX format.

2. Import and select a local grid coordinate system.
☞ Infinity can process pure WGS84 data, but it can only display local grid coordinates in the graphical view. This is the reason why you cannot view the points. The navigator shows you the following icon: .
Once you have selected a coordinate system the icon in the navigator changes to and you can see the points in the graphical view.
On how to import or define a coordinate system, see Coordinate Systems.
On how to select a coordinate system within a project, see Coordinate Systems inside a Project.

3. Adapt the processing parameters. In the main menu, go to the Processing tab. Its ribbon bar offers you all GNSS relevant functionality.
Select to open the Data settings dialog.
See also:
Settings: Data
Select to open the Processing Strategy dialog.
See also:
Settings: Strategy
☞ You can also define the data settings and the processing strategy through **File** > **Info & Settings** > **Data Processing** > **GNSS**.
Select to open the GNSS Manager.
For further details on different processing strategies, see GNSS Manager.

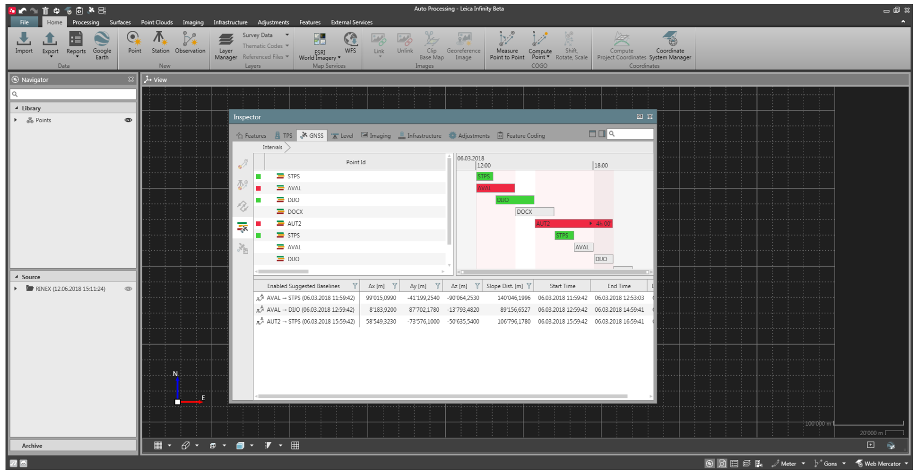
Manual processing

To post-process the imported raw data manually, proceed as follows. You see that directly after import the role of each point is navigated. When you have successfully processed the data, it is GNSS post-processed.

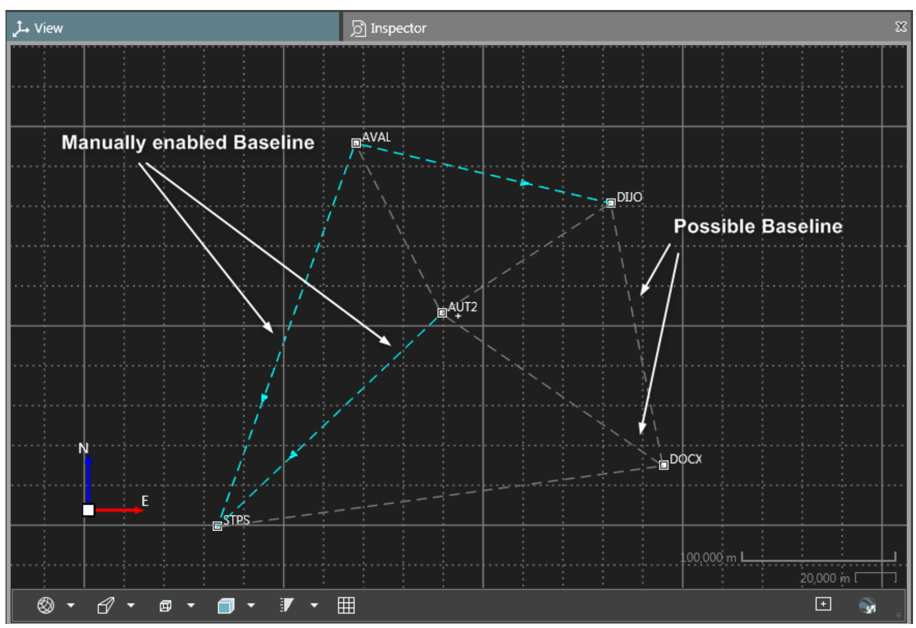
1. Go to the Inspector GNSS tab.
Select the **Intervals** to display the list of all imported intervals.
Select**Start Time** in the header to sort the intervals by time.
Select the option next to the Search field to see a graphical representation.

2. Be sure to have assigned the correct point roles that you use as a reference. To assign as control, right-click on the interval and select **Assign Control Point**.

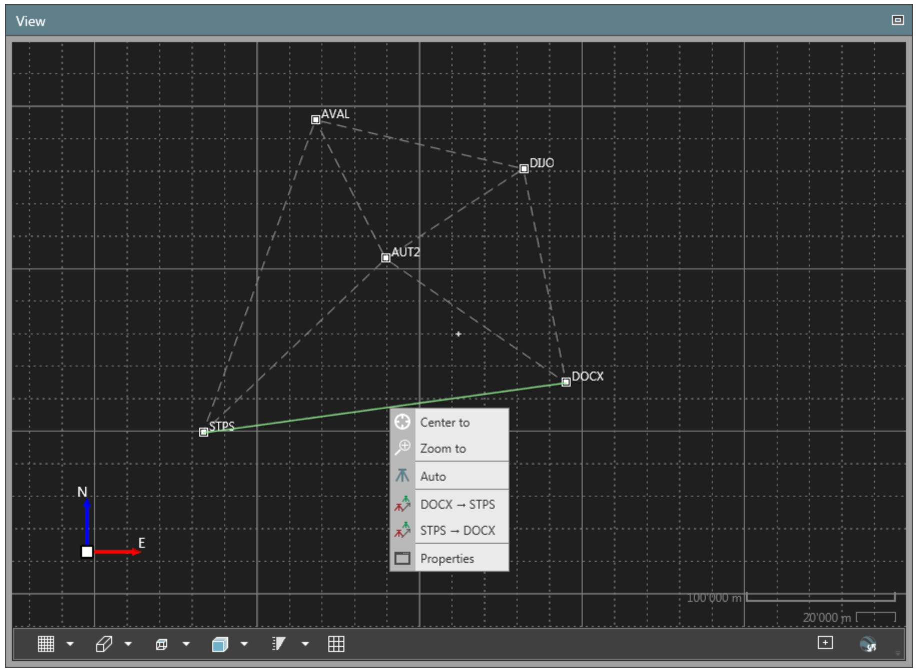
3. Specify the reference and the rover intervals. To do so right-click on an interval in the Inspector and select either **Reference** or **Rover** from the context menu.



☞ After setting your reference and rover station, the enabled baselines are listed in the **GNSS** > **Inspector** > **Enabled Suggested Baselines**. If there is a coordinate system attached to the project, the manually enabled baselines are also visible in the graphical view (before processing). The line colour of the manually enabled baseline is taken from the GNSS observation layer. Line width and line style of the manually enabled baselines are taken from the GNSS suggested baseline layer.



☞ You can also enable a baseline by selecting it from the graphical view (GNSS suggested baseline layer) and setting the direction from the context menu. After this action, the interval view is updated to reflect the changes.



☞ Intervals multi-selection is possible.

☞ Reference station data can also be downloaded from the Internet.
See also:
GNSS Manager

4. When you have selected a reference and all its rover intervals select in the ribbon bar to process the data.

   ☞ Short-cut functionality is available here to change the antenna calibration set before starting the processing run.

5. Highlight the rover and its intervals again in the Inspector and select **Reset** from the context menu to remove the reference and rover flags.
   Specify the next reference and its rovers.
   Select the **Processing** option again in the ribbon bar.

6. Repeat Step 4 until all baselines are processed.

7. In the Inspector, the Results tab has become active. Drill into each processing run by selecting the little arrow right next to it and inspect the results for each baseline.

   ☞ Select in the ribbon bar to generate a report on the currently selected element. You can generate reports on single baselines, single tracks as well as on whole processing runs.

8. Select the results to be stored and select 💾 in the ribbon bar.
You can store single baselines, single tracks as well as whole processing runs.

☞ When you store points that have been computed with two or more different reference stations, then the point average is computed regardless of the solution type. This means that if a phase fixed solution and a code solution shall be averaged then the average is computed in any case and it is left to the user to decide whether the code solution shall be ignored or not.
When you store tracks that have been computed with two or more different reference stations, then the track average is computed to the solution type. This means that if a phase fixed solution and a code solution shall be averaged then automatically the phase fixed solution is taken into account while the code solution is ignored.

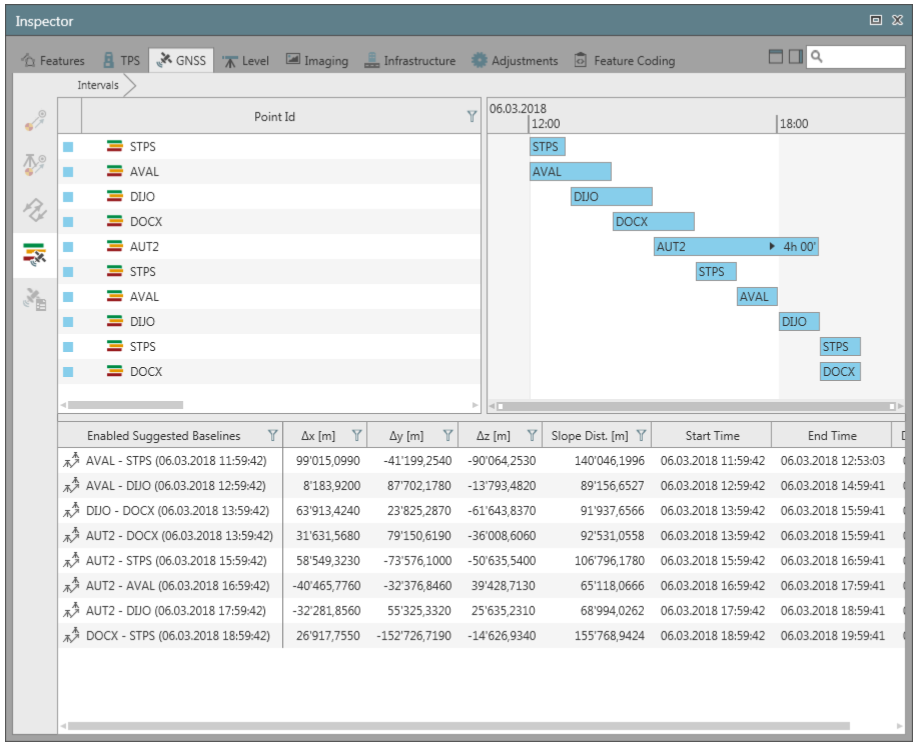
☞ Only stored results are displayed in the graphical view.

Automatic processing

To find the possible baselines automatically and post-process the imported raw data, proceed as follows.

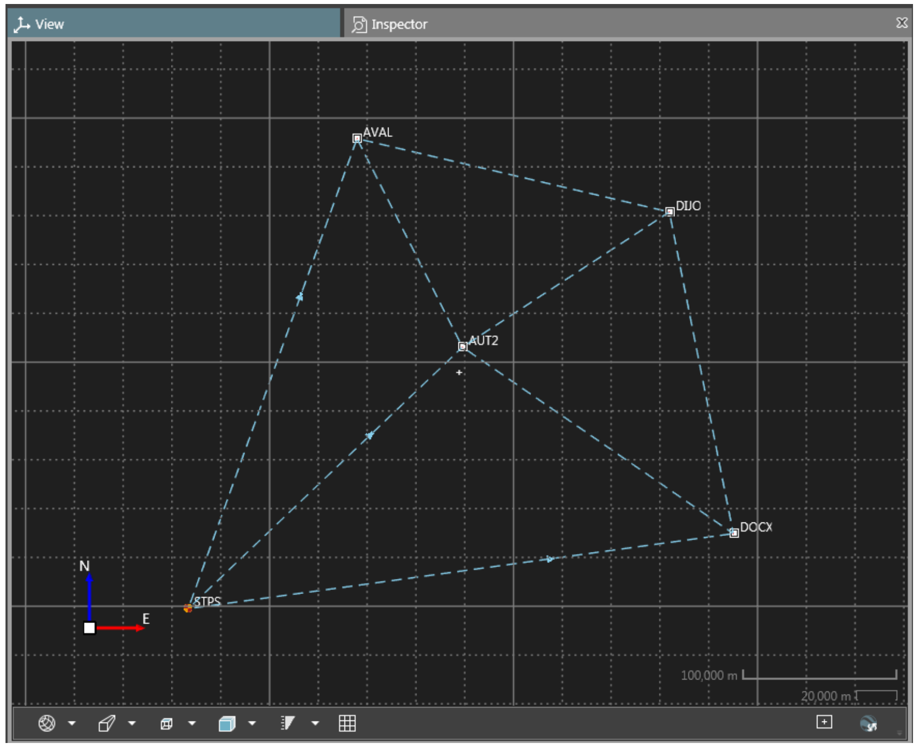
1. Go to the Inspector GNSS tab.
2. Be sure to have assigned the correct point roles for your control stations. To assign as control, right-click on the interval and select **Assign Control Point**.

3. Highlight all the intervals you want to consider for the baselines creation. Set the intervals to automatic from the GNSS ribbon bar or from the interval context menu.

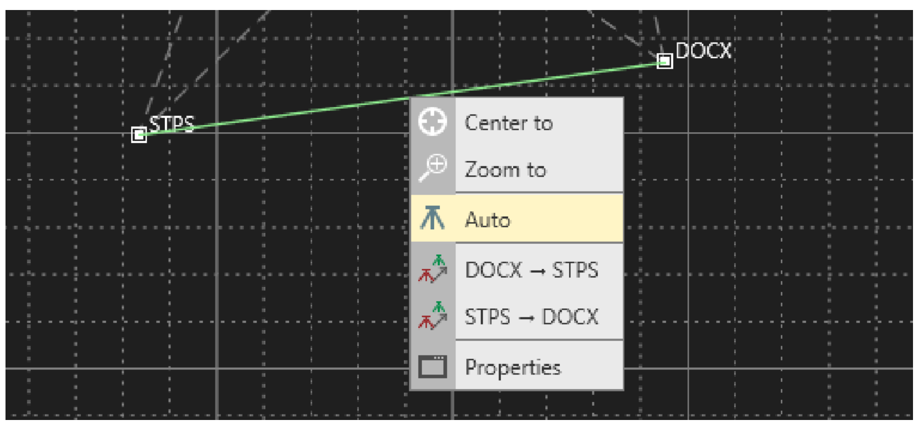


☞ After setting your intervals to automatic, the enabled possible baselines are listed in the **GNSS** > **Inspector** > **Enabled Suggested Baselines**. If there is a coordinate system attached to the project, automatically enabled baselines are also visible in the graphical view (before processing). The direction of the baseline is not always known before processing, but it is determined once that the automatic algorithm is started.

☞ The line colour of the automatically enabled baseline is blue by default. Line width and line style of the automatically enabled baselines are taken from the GNSS suggested baseline layer.

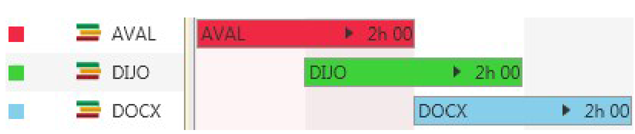


☞ You can also enable a baseline by selecting it from the graphical view (GNSS suggested baseline layer) and selecting **Auto**.

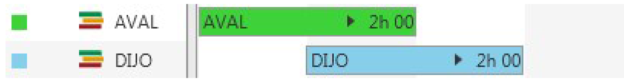


Manual processing and automatic processing can also be used together. In such a setting, the manual processing has the priority to the automatic processing in the baselines computation order. Moreover, results of manually selected baselines can be used to feed the automatic processing.

For example, in the following setting, the baseline AVAL -> DIJO is computed first. Therefore the coordinates of DIJO have been established and can be used to solve the baselines DICO -> DOCX.

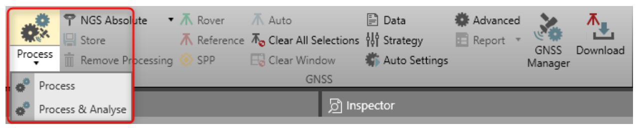


A lonely interval set to reference (without rover), or a lonely interval set to rover (without reference) is not combinable with an interval set to automatic. In the setting below, no processing is performed.



Processing

To launch the processing, select the **Process** or **Process & Analyse** option in the GNSS ribbon bar.



The processing options work as follows:

- **Process:**
  Run the GNSS data processing and show in the results basic data quality information, such as satellite tracking, DOP, ambiguity statistics. Additional data analysis products (observations residuals, position residuals) are not available in the results so to reduce the processing time.
  Choose this option to process a large amount of data.
- **Process & Analyse:**
  Run the GNSS data processing and output additional products to analyse the results. By default, Infinity outputs observation residuals (for all kinds of datasets) and position residual (for static dataset).
  The behaviour of the option can be modified from the GNSS advanced settings.
  Select this option to make a complete and deep data analysis.

3.2.3 Create a Virtual Reference Station (VRS)

Create a Virtual Reference Station (VRS)

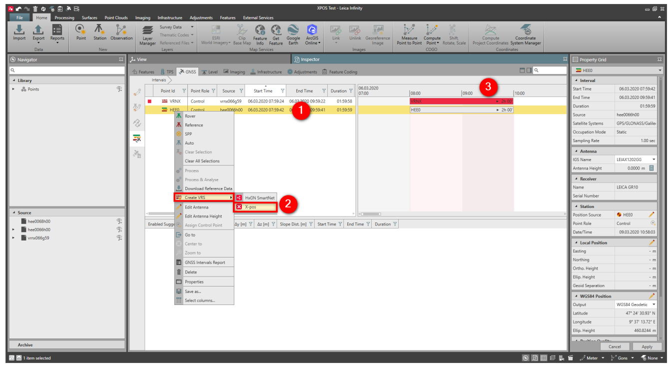
A Virtual Reference Station (VRS) is created nearby the rover position by exploiting corrections, tropospheric and ionospheric modelling coming from a network of physical reference stations.

Infinity creates VRS exploiting the HxGN SmartNet GNSS reference stations. Rover data from short baselines can be processed resulting in an accurate final VRS position.

Requirements:

- Valid subscription for HxGN SmartNet or X-pos.

How to create a VRS:



1. From the GNSS interval view, select the intervals that you want to post-process using a VRS.
2. Select **Create VRS** from the GNSS ribbon bar or from the context menu.
3. Infinity connects to the SmartNet network to download the RINEX file of the VRS.
   The VRS data is directly imported into the project.
   The additional interval is set to reference automatically.

   ☞ A VRS can only be set as reference. A VRS cannot be set as rover or SPP.

When the VRS is created starting from one interval, one virtual reference station is created in the location of the selected interval.

When the VRS is created starting from multiple intervals, Infinity creates one VRS (in the average position) or multiple VRS (in the location of each selected interval). This depends on the specified advanced settings for the VRS set in Settings: Advanced.

See also:

HxGN SmartNet

X-pos

3.2.4 GNSS Analysis Tools

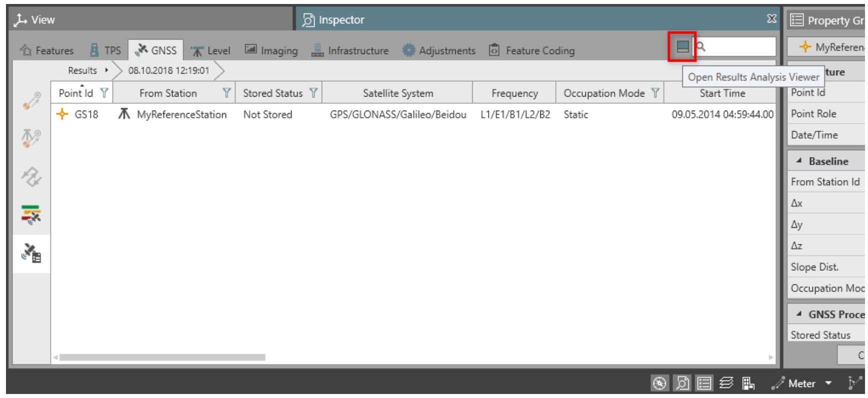
GNSS Analysis Tools

The GNSS post-processing module offers tools to investigate the details of the processing results.

To visualise the charts within the application:

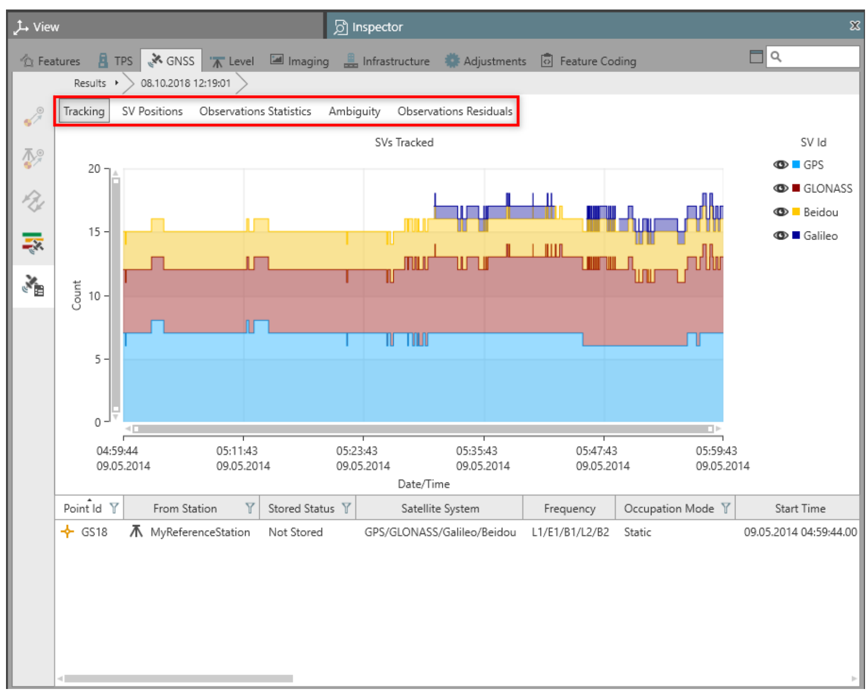
1. Open the GNSS processing results from the **Inspector** > **GNSS** > **Results**.

2. Open the **Results Analysis Viewer**, by selecting the option situated in the top right of the GNSS Inspector.



3. The **GNSS** > **Inspector** > **Results**, is split into two parts.
   - Bottom part
     Has the processing results.
   - Top part
     Has the charts with information about: Tracking (with used/rejected satellites and used/rejected observations per epoch), SV Position, Observations Statistics, Ambiguity and Observations Residuals.

   You can navigate within the charts by selecting the chart menu at the top of the plots.



Position Residuals Charts

Position residuals come from the kinematic processing of static data.

The charts are available by:

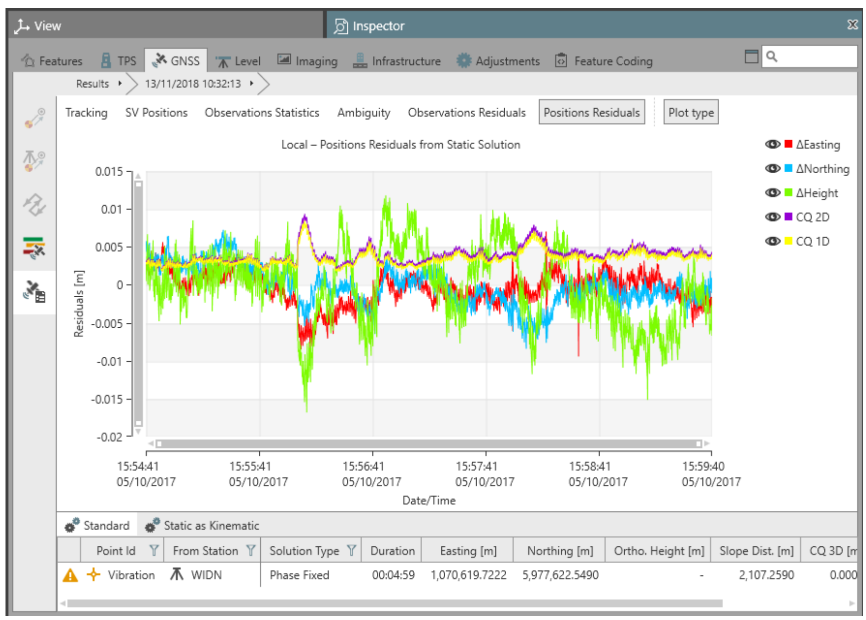
- Selecting the **Process & Analyse** option.
- Setting **Process & Analyse Output** to position residuals or observation and position residual in the GNSS advanced settings, see Settings: Advanced.

Position residuals are computed with respect to:

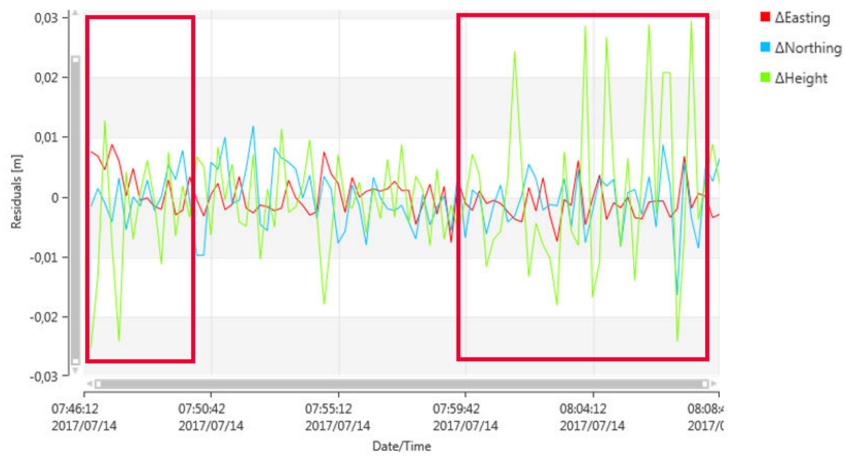
- Static Solution: The coordinates of the computed point (from the static processing) are taken as reference to compute the residuals.
- First Epoch: The coordinates at the first epoch of the kinematic processing are taken as reference to compute the residuals.
- Previous Epoch: The residuals at each epoch (t) are computed with respect to the solution of the kinematic processing at the previous epoch (t-1).
- User Entered: The residuals at each epoch are computed with respect to a point existing in the project. You can select the point through the edit pencil.

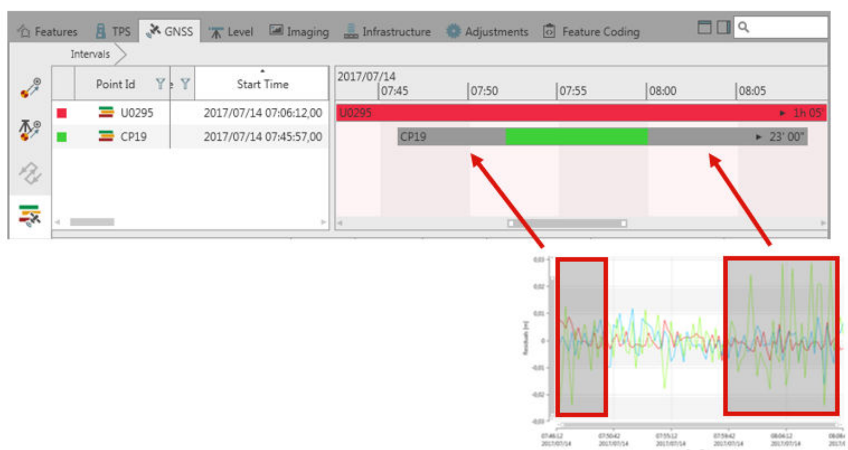
In the **Inspector** > **GNSS** > **Results** > **Static as Kinematic** tab, the statistics over the position residuals are available.

☞ The position residuals plot and statistics, can be used to evaluate the internal reliability of the data set and of the static solution. Residuals with a small dispersion around the zero, are indicator of good reliability.

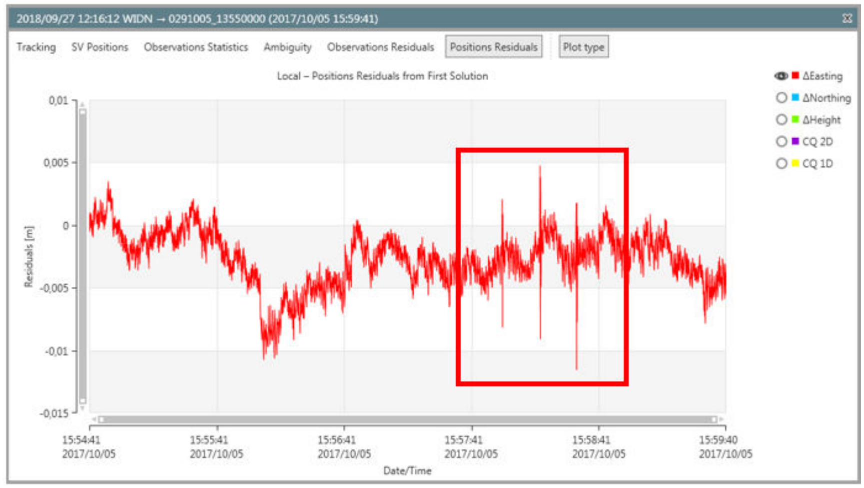


☞ Position residuals can help to identify possible issues in the solution, and at the same time could suggest a possible workaround. If you realise that part of the residuals are particularly noisy, it is suggested to exclude that range of data from the processing, by applying exclusion windows. In the following example (the chart represent the position residuals from the previous epoch), it is suggested to manually exclude from the processing the first and the last part of the data set.

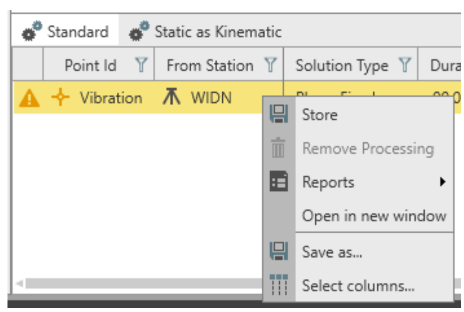




☞ The positions residuals plot can also be used to identify displacement of the antenna. The following picture, shows residuals of an antenna subjected to vibration.



☞ By selecting **Open** in a new window from the **Inspector** > **GNSS** > **Results** > **Advanced**, a different type of plot can be opened and compared in a new window.



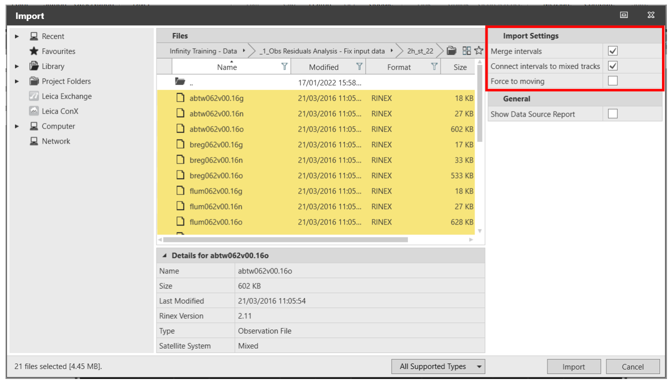
3.2.5 RINEX Import

RINEX Import

The standard and most common format to store and share GNSS raw data is the Receiver Independent Exchange Format (RINEX). RINEX data can be easily imported in Infinity from the import dialog or using drag and drop.

☞ Infinity also supports the import of the GNSS data in proprietary format, such as Leica (.mdb), NovAtel (.gps and .dat) and u-blox (.ubx).

Import dialog:



Import Settings

Merge intervals:

The merge intervals functionality is normally used when importing data from permanent reference stations. If more than one GNSS interval is contained in the data then this function merges any observation intervals that conform to the following conditions:

- The gap between any two consecutive intervals is less than 30 minutes.
- The point ID are identical for consecutive intervals.
- The antenna type is identical for consecutive intervals.
- Coordinates are the same for consecutive intervals.

Connect intervals to mixed tracks:

This setting is used for consecutive measuring sessions of static and moving GNSS raw data.

Check this box to create a unique GNSS interval, containing static and moving data (Mixed Track).

Force to moving:

Check this box to import static data as moving data.

This checkbox helps to overcome storing issues happened on the field (for example, you collected moving data, but the receiver stored the data as static).

3.2.6 Settings: Data

Settings: Data

Here you can select the settings that apply to the data used for GNSS post-processing.

Cut-off angle:

The system default is 10°.

Increase the cut-off angle to avoid the noisier low elevation satellites. This would be used if there may be problems in solving of ambiguities to arrive at a phase fixed solution.

Sampling Rate:

The system default is to use all recorded data.

Increase the sampling rate to use only sub-sets of all recorded data.

Decrease the sampling rate when using long observation times to reduce the processing time.

Used Satellites:

The system default is to use all available satellites.

Select **Manual selection** and then on the option to deselect unhealthy satellites.

☞ Inactive satellites are disabled by default.

☞ To completely enable/disable entire constellations go to **Info & Settings** > **Data Processing** > **GNSS** and select/deselect the satellite systems in the data section.

Ephemeris Type:

The system default is to use broadcast ephemeris.

In order to use precise ephemeris, import the required NOAA/NGS *.sp3 to the project.

See also:

GNSS Manager

☞ The date of the *.sp3 file must correspond to the date of the data to be processed, else Infinity switches back to broadcast automatically.

☞ Download the required *.sp3 file, see Precise Ephemeries.

Antenna Calibration Set:

The system default is Leica relative.

Additional calibration sets for non-Leica antennas are offered for selection but can also be downloaded from http://www.ngs.noaa.gov/ANTCAL/. You can do so through the GNSS manager.

See also:

GNSS Manager

3.2.7 Settings: Strategy

Settings: Strategy

Solution Type:

Set the solution type, the processing engine should output:

- The system default and recommended setting is phase fixed where processing attempts to fix the ambiguities.

☞ The computed solution can also be a Widelane fixed solution, which is the equivalent of the xRTK solution on Captivate/SmartWorx Viva RTK rovers and applies mainly with difficult data and on typically short (<50 km) baselines.

☞ For baselines where the phase fixed solution is not reliable, the solution type reverts to a float solution automatically.

- Select **Float** when you process long baselines and have long observation times. The system then does not try to solve the ambiguities.

☞ If in the data settings you have chosen to process a single frequency solution then the system processes 'L1 only', 'B1 only' or 'E1 only' float solutions.

- Select **Code** in order to compute a code only solution when lower accuracy results are sufficient.

Solution Optimisation:

Set how the processing engine considers the tropospheric and ionospheric biases.

- The system default and recommended setting is automatic which allows the processing engine to determine when to compute a linear combination and which makes it automatically use the best combination of frequencies.

☞ The input data is evaluated during the processing and considers the effects of the ionosphere. When the processing engine determines the reliability of the solution is higher when using a linear combination, then this is automatically considered for the solution.

☞ For the GPS constellation, the system can compute an iono-free solution over three frequencies (if L1, L2 and L5 are all available). To change the frequencies to use, go to the advanced settings.

- Set to **Iono Minimised** to force the processing engine to attempt a linear combination solution.

☞ For baselines longer than 15 km (system default, can be changed in the advanced settings) or with a noisy ionosphere, the system automatically computes a linear combination iono-free solution.

- Set to **None** when you do not want the processing to consider a linear combination for the solution.

☞ When set to none the processing engine uses the selected Ionospheric model to apply during processing.

Tropospheric Model:

The system default and recommended setting is automatic, which uses the VMF (Vienna Mapping Function) with GPT2 (Global Pressure and Temperature - 2) for baselines whose duration is shorter than 90 minutes. If the data is longer than 90 minutes, a computed tropospheric model gets applied.

Indeed, when enough observations are provided, the use of a computed tropospheric model, can be more effective than an a-priori model (such as the VMF, Hopfield or Saastamoinen, also available in Infinity), because it is based on the current status of the troposphere. The longer the observation period, the better the estimation of the troposphere is.

Iono Model:

The system default and recommended setting is automatic.

☞ If you have at least 45 minutes of static or rapid-static dual frequency data collected on the reference station then the system chooses computed. The computed model suits your observations best. If less than 45 minutes of data are available, then it automatically switches to none.

☞ Only choose none when you can be sure that ionospheric activity is low.

☞ You can also use a global/regional model if available. Ionospheric files can be imported through the GNSS manager. If you select this option but no such file is available, then the system switches to use none automatically.

3.2.8 Settings: Advanced

Settings: Advanced

Advanced Processing Strategy

Frequency:

The system default is automatic.

All available frequencies are used, like if available Infinity makes use of triple frequencies.

☞ The system uses the selected frequencies only for the constellations enabled in the **Info & Settings** > **Data Processing** > **GNSS**, for instance to use Galileo E5a E5b E5ab, Galileo must be manually activated. Select Galileo in the data section under satellite systems.

☞ If multi frequency is not available the system automatically processes a single frequency solution. For users having the single frequency processing option, only 'L1/B1/E1" is used by default.

Frequency to use in Iono Minimized:

This setting is active only if you have set solution optimization to Iono Minimized in the strategy settings.

The system default is automatic. Infinity finds the best combination of frequencies to use in the iono-free computation.

To exclude L2/B2 (GPS, Glonass and Beidou) or L5 (GPS) from the iono-free linear combination, select the corresponding options.

☞ For GPS, a triple frequency iono-free solution can be computed.

Min. Distance for Iono Minimized:

By default an iono minimized solution is only attempted for baselines longer than 15km.

Adapt the value to force the system to compute an iono minimized solution even for shorter baselines.

☞ If in the settings: Strategy Solution Optimization is set to Iono Minimized, then the processing engine always attempts a linear combination solution independent of the baseline length.

Possible Ambiguities Fix up to:

The system default is 300km.

Defines the maximum length of a baseline for which the system shall try to resolve ambiguities. For baselines above the limit a float solution is computed.

If you want to ensure a higher accuracy and reliability you can enter a lower value.

Min. Duration for Float Solution (static):

The system default is 5 min.

Defines the time below which the computation of a float solution for static intervals shall not be allowed, because for short observation intervals a float solution may not be accurate enough. In case ambiguities cannot be resolved the processing engine computes a code-only solution for intervals below the limit.

Allow Widelane fix:

By default this setting is active.

It allows for the computed solution possibly being a Widelane fixed solution, which is the equivalent of the xRTK solution on Captivate/SmartWorx Viva RTK rovers and applies mainly with difficult data and on typically short (<50km) baselines.

Analysis Tool

Process & Analyse Output:

Establish the type of output for the process & analyse mode. The analysis products are shown in the GNSS results analysis viewer.

Observation residuals can be used to analyse the noise of the solution on the observations domain.

Position residuals help to analyse the noise of the static solution on the positions domain. The position residuals are derived by processing static data in kinematic mode.

Select **None** to remove the analysis mode option from the GNSS processing ribbon bar.

Select **Observation & Position Residuals** to make a complete and deep data analysis.

Virtual RINEX Download

From multiple points:

This setting is applicable when the user creates a VRS (Virtual Reference Station) from multiple intervals having different positions.

Select **Create one VRS in the middle** when working with raw data collected in a small area (hundred meters). Infinity creates one VRS, whose location is the average of the points.

Select **Create one VRS for each point** when working with raw data collected in a large area (kilometres). Infinity creates a VRS close to each point.

Name:

Define the name to assign the interval for the VRS.

Sampling Rate:

Define the sampling rate for the observation of the VRS.

By default automatically, Infinity uses the sampling rate of the GNSS intervals from where the VRS is generated.

3.2.9 Settings: Automatic Processing

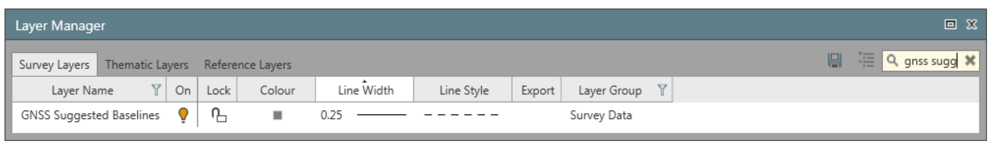
Settings: Automatic processing

The automatic processing settings define the logic Infinity uses to:

1. Identify the possible baselines and draw it in the graphical view prior to processing, through the GNSS suggested baselines layer.
2. Fix the rules to decide which baselines should be processed and in which order.

Suggested Baseline Strategy

These parameters are considered to identify the possible baselines. If the GNSS suggested baselines layer is on, the possible baselines are automatically shown in the graphical view after importing raw data.



Function	Description
Min. Baseline Duration	This sets the minimum amount of time over which simultaneous measurements must be taken at two stations before Infinity try to show the preview and process the baseline between those two stations.
Max. Baseline Length	Sets the maximum length of baselines up to which Infinity shows the baselines in the preview and process.
Re-Compute already computed Baselines	If this option is checked, baselines that have been calculated and stored previously are proposed again as suggested baselines and sent to the processing.
Compute Baselines between Control Points	If this option is checked, baselines between control points role are also processed. This may be interesting if the control points are not kept absolutely fixed in a subsequent adjustment.

Auto Processing Strategy

Function	Description
Baseline Processing	**All**: Infinity processes all possible combinations of baselines that conform to the automatic processing parameters. **Independent sets**: Infinity only processes a set of independent baselines. Between **n** points which are measured at the same time only **n-1** independent baselines exist.
Priority to Baseline with	**Shorter Slope Distance**: The shortest baseline from the first reference point is computed first. Infinity then decides which is the next shortest baseline. This may be from the first point or from the point that was last computed. This line is processed next and then the process is repeated. This is also dependent on the other automatic processing parameters. **Longer Duration**: The baseline with the longest common observation time is computed first. Similarly, to the distance method, Infinity then decides which point has the next longest common observation time and processes that line. The process continues like that. This is also dependent on the other automatic processing parameters.
Session by Session	If this option is checked, Infinity computes all possible baselines from the reference that has been identified according to the selected seeding strategy before proceeding with the next reference. The point with the longest interval is selected as the first reference.
Allow to use Float solution as Reference	Allow points whose solution is float to be used as reference points for further processing.

General

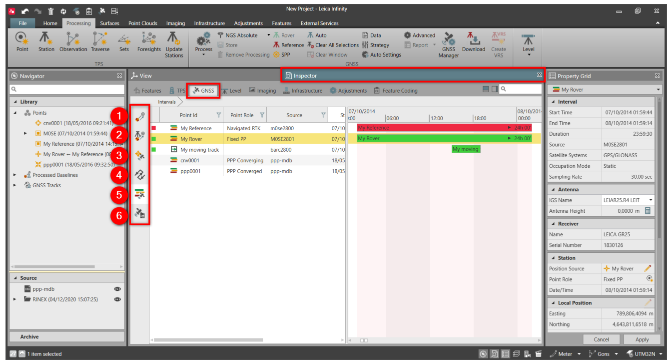
Function	Description
Set Intervals to Auto at Import	If this option is checked, Infinity sets all the intervals to automatic immediately after data import. This setting is suggested in case the user measured a GNSS network with many sessions of reference and rover, and the user wants to process all the possible baselines in one go. If this option is checked, the user should only import all the data and select **Process**.

3.2.10 GNSS Inspector

3.2.10.1 Overview

GNSS Inspector

The GNSS inspector is place where you can visualise and work with all the GNSS data.



No.	Name	Description
1.	**All GNSS Observations**	Lists of all the baselines (RTK or Post-processed) existing in your project.
2.	**GNSS Observations by Station Source**	Lists all the baselines (RTK or Post-processed) organised by reference station.
3.	Point Measured by Precise Point Positioning	List of PPP points measured from Captivate in the field using Smartlink correction service.
4.	GNSS Tracks	Shows the tracks, derived from post-processing of moving intervals.
5.	GNSS Intervals	Shows graphical representation of the raw data to use for post-processing.
6.	GNSS Processing Results	Visualise, analyse and store your processing results.

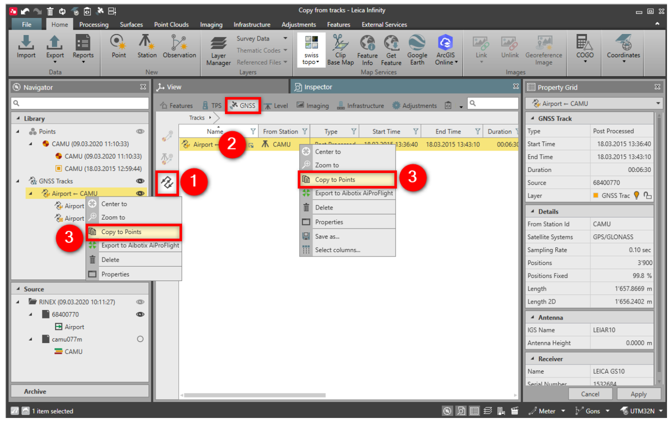
3.2.10.2 GNSS Tracks

GNSS Tracks

The GNSS tracks tab lists all tracks in the project. A track is created using post-processing and storing moving intervals.

When processing kinematic data from two or more reference stations, a reduced kinematic track is computed. Drill in into a track to inspect details and visualise the contributors to the tracks.

To extract discrete points from a GNSS track:



1.	Go to **GNSS Tracks**.
2.	Select the track (entire track or its contributors).
3.	Select **Copy to points** from the context menu or from the Navigator/Library.

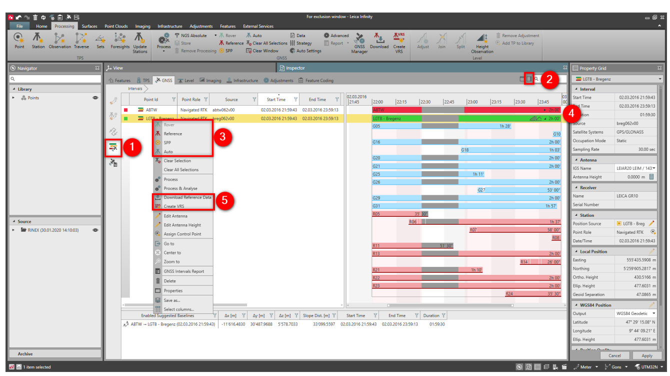
☞ The reduced track, its contributors and the copy to point functionality are also available in the navigator library, for quick access and management of the tracks.

3.2.10.3 GNSS Intervals

GNSS Intervals

GNSS intervals are a graphical representation of the raw data imported into Infinity. They allow you to evaluate the quality of the input data (for example, data completeness and continuity) and to perform some actions on the data set itself (for example, exclude part of the data or download reference station data for data post-processing).

To open or close the interval view:



1.	Go to **GNSS Intervals**.
2.	Select the ☐ option next to the search field.
To set the Interval as a Reference , Rover , SPP or Auto:	
3.	Select the Interval and choose the function from the ribbon bar or the context menu.
To view the details of the Intervals:	

4. Click the arrow ▶ 55' 15" to open details on each satellite in the satellite view.
If you do not see the arrow, zoom over the interval, or expand the interval if you are working with a mixed track.

☞ The default view shows satellites grouped by constellation type and includes display of cycle slips.

Select the , to display the SNR values: Red or Orange data is poor data.
Select , to view the elevation of the satellite.

To download data from a Reference Station or VRS:

5. Select **Download Reference Data** or **Create VRS** from the context menu.

☞ Data to be downloaded matches the date/time of the intervals from permanent reference stations.

See also:

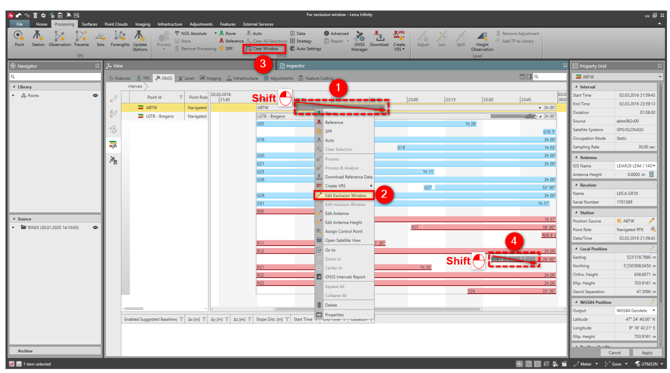
Reference Stations

Create a Virtual Reference Station (VRS)

Exclusion Window

Normally, the whole observation period of an interval is used for processing and Infinity automatically rejects parts of the data which are not beneficial to the results. However, you may want to manually exclude part of the data.

To create an exclusion window:



1. Press ‘Shift’ + ‘left mouse button’ and drag the mouse over the Intervals from left to right.

☞ An exclusion window is created and is not used in the data processing.

2. You can manually edit the start and end time of the exclusion window by selecting **Edit Exclusion Window** from the exclusion window context menu.

3. You can remove the exclusion window by selecting the interval and choosing **Clear Window** from the GNSS ribbon bar.

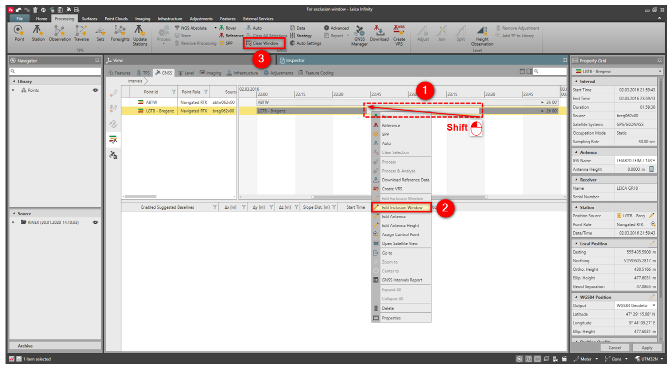
☞ You can remove the exclusion window also working with the mouse, moving it in the opposite direction (from right to left) like what is described in step 1.

4. You can create an exclusion window also for single satellites, by using the same functionality described previously.

Inclusion Window

You may want to create an inclusion window, because you only want to work with part of the data you collected.

To create an inclusion window:



1. Press 'Shift' + 'left mouse button' and drag the mouse over the Intervals from right to left.

   ☞ An inclusion window is created, and the rest of the data is excluded.

---

2. You can manually edit the start and end time of the inclusion window by selecting **Edit Inclusion Window** from the inclusion window context menu.

---

3. You can remove the Inclusion windows by selecting the interval and choosing **Clear Window** from the GNSS ribbon bar.

   ☞ You can also create an inclusion window without working with the mouse, but by directly selecting **Edit Inclusion Window** from the context menu of the intervals. Enter start and end time of the inclusion window and select **OK**.

See also:

Reference Stations

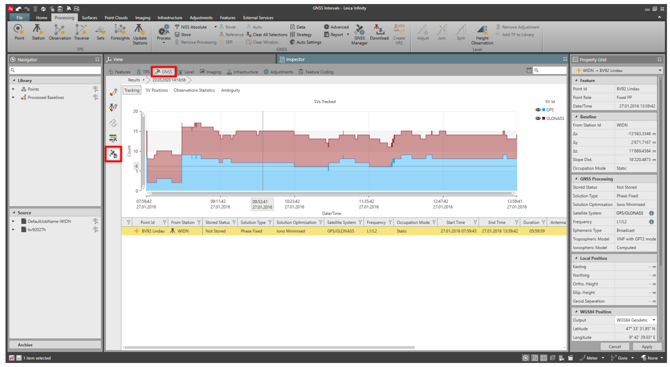
How to Post-Process GNSS Data?

GNSS Manager

3.2.10.4 GNSS Processing Results

GNSS Processing Results

Drill into a result and select the option (next to the search field), to see a graphical representation of the results, for single baselines.



See also:

How to Post-Process GNSS Data?

3.2.10.5 Point Measured by Precise Point Positioning

Point Measured by Precise Point Positioning

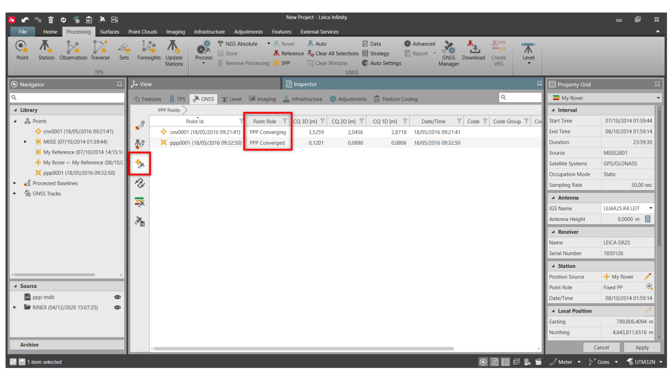
The Precise Point Positioning (PPP) Points tab lists all the points measured in the field with Captivate, using the PPP technology, based on the Smartlink correction service.

Smartlink is a correction service where GNSS correction data, is transmitted by augmentation satellites (no reference station needed). Measuring a point with this technique, is an alternative to the RTK survey in a situation where a local reference network is not available or not accessible.

Using Smartlink, once the augmentation satellites are tracked, the process takes around 10-30 minutes for the coordinate quality to converge to few centimetres in position.

According to the converging process status, the measured point role can be:

- PPP Converged (higher quality).
- PPP Converging (lower quality).



3.2.11 GNSS Manager

3.2.11.1 Overview

GNSS Manager

The GNSS Manager holds all GNSS-related information such as antenna calibration sets, precise ephemeris and iono models. It also contains reference station download and satellite availability prediction tools.

To start the GNSS Manager select from the GNSS Processing ribbon bar or in the Infinity title bar.

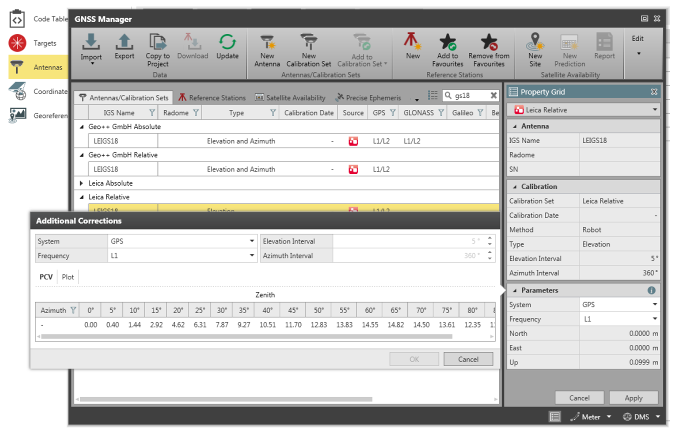
3.2.11.2 Antennas/Calibration Sets

Antennas/Calibration Sets

Here all the antennas considered in the project are listed, grouped by calibration set.

By default, Leica, Geo ++ and NGS calibration set come with the installation package. Leica and Geo++ consist of only Leica antennas; NGS consists of Leica and third parties antennas. The antenna manager allows you to get information about the properties of the antennas.

Select the antenna and open the Info fly-out to display the antenna PCV and offset parameters.



Import antennas (ANTEX format)/calibration sets:

These files are available from various agencies including the International GNSS Service (IGS) or National Geodetic Survey (NGS) and have a list of antenna types by manufacturer with their respective offsets and eccentricities.

☞ When processing baselines with third-party GNSS sensors it is suggested to use the NGS calibration set. If the antenna is not in the selected calibration set, then a NullAntenna is used for both receivers of that baseline.

☞ Antennas can be copied to the Antenna Management through the Copy to Global option.

Add a new antenna:

1. Enter the antenna details in the Add Antenna property grid.
   You can manually enter the antenna offsets and eccentricities when needed.
2. Select **Create**.

The new antenna is added to the calibration set that you have selected in the properties and marked as user-defined.

Create a new calibration set:

Allows you to create user-defined calibration sets, to combine antennas of different agencies into one list and use them for processing.

All antennas must be absolute or of a relative type.

1.	In the Antennas/Calibration Sets tab, select at least one antenna to make the option become active.
2.	Select the **New Calibration Set** option or right-click into the selection and select **Add to Calibration Set**, then **New Calibration Set** from the context menu.
3.	Enter the details in the New Calibration Set property grid.
4.	Select **Create**.

The new calibration set is added and marked as 👤 user-defined.

Add to calibration set:

1.	In the Antennas/Calibration Sets tab, select at least one antenna to make the option become active.
2.	Select the **Add To Calibration Set** option or right-click into the selection and select **Add To** from the context menu. ☞ You can only add antennas to 👤 user-defined calibration sets. You cannot add more than one antenna with the same name (but with different calibration values) to the same user-defined calibration set. ☞ An antenna from an absolute calibration set cannot be added to a relative calibration set and vice versa.

The antenna is added to the selected calibration set and marked as 👤 user-defined.

Delete antennas/calibration sets:

☞ Only 👤 user-defined or 🗋 imported antennas/calibration sets can be deleted.

See also:

Antennas

The tutorial **"How to Manage an Antenna Calibration Set"** https://leica-geosystems.com/-/media/a633447d1f3f4afc8c7abf4a1685be6a.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

3.2.11.3 Reference Stations

Reference Stations

Download GNSS reference station data

See also:

The tutorial **"How to download reference data"** https://leica-geosystems.com/-/media/817c1f0ca2634823b2da45a5359cefec.ashx

☞ Tutorial data can be downloaded in the Localisation Tool.

Use the Internet download function to download and import static GNSS data from continuously operating reference stations.

Stations made available by the following providers are found automatically.

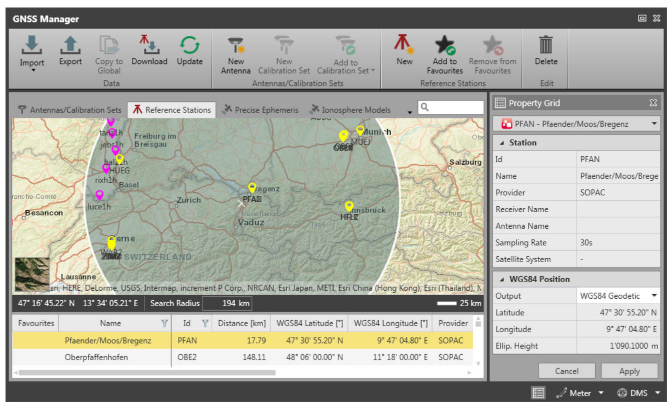
- SOPAC:
  http://garner.ucsd.edu/pub/rinex/
- NGS (NAFTA list of CORS stations):
  https://geodesy.noaa.gov/corsdata/rinex/
- UNAVCO:
  https://data.unavco.org/archive/gnss/rinex/obs/
- RGP:
  ftp://rgpdata.ign.fr/pub/data/
- AG
  https://data.gnss.ga.gov.au/

SmartNet Stations are found when you are logged in to the SmartNet Service.

See also:

HxGN SmartNet

All available stations are shown in a map view:



The list is filtered with the radius of the circle:

Click onto the border of the circle and drag it to shrink or enlarge its radius and find more or less stations.

Click into the circle and shift it to find stations in another region.

☞ By default, the circle is located with a radius of 300 km around your project data.
If the project is empty the circle is by default located with a radius of 300 km around Heerbrugg, Switzerland.
If you have selected sites as favourites, the circle is by default located with a radius of 300 km around your favourites.

Download reference station data from the Internet:

1. Select the stations you want to use and select **Data > Download** in the ribbon bar.
   Alternatively, right-click onto a station and select **Download** from the context menu.
   Next to the Property Grid a Download tab opens up.

2.	Enter a specific date and time for which you want to download data. By default, the files suggested for download are within the time frame of your project data. ☞ If one or more intervals are selected in the GNSS Inspector, the download takes the date/time based on your selection. Else the date/time is taken based on all available data.
3.	Choose another download directory if you want. By default, the download directory is your download directory or the project folder if available.
4.	Choose whether you want to download Precise Ephemeris and Navigation File simultaneously.
5.	Select the **Start** option at the bottom of the tab to start the download.
6.	Follow the download progress in the Status section.
7.	In case of errors, check the availability of the files to be downloaded. Availability can comfortably be checked by clicking the hyperlink on the file name.
8.	When download is complete, the data is automatically imported to your project.

Intervals are automatically merged on import.

See also:

RINEX Import

Precise Ephemeris are added to the list in the Precise Ephemeris tab.

☞ Infinity offers a shortcut to access Internet Download. You can also select the **Download** option in the GNSS Processing ribbon bar or select one or more intervals in the GNSS Inspector and choose **Download** from the context menu. In both cases, the GNSS Manager opens in the Reference Stations tab.

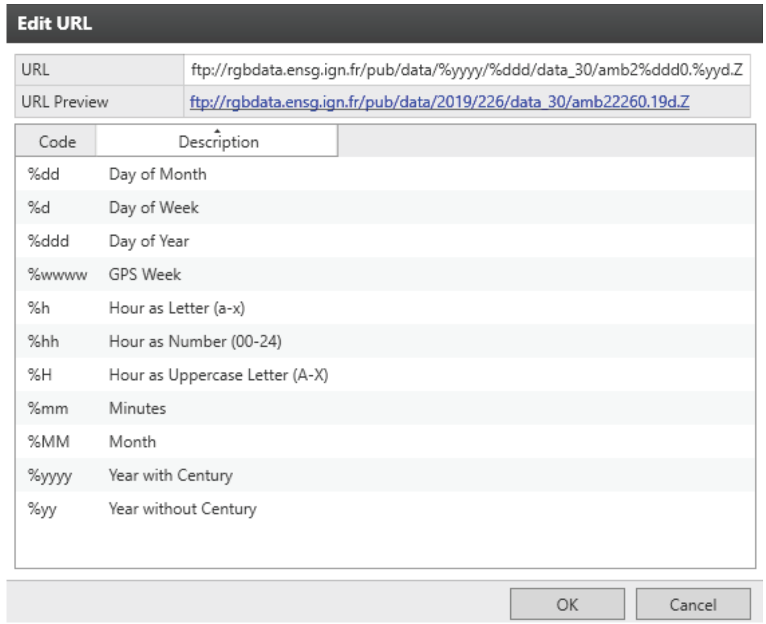
Update reference stations:

To update the list of reference stations, select **Data** > **Update** in the ribbon bar.

New reference station:

1.	To add a new reference station, select **Reference Stations** > **New** in the ribbon bar.
2.	In the Property Grid enter the Station Info and the WGS84 Position of the station.
3.	To define the Connection, select the **Edit** option.

4. In the Edit URL flyout enter the **FTP address**.
For example: ftp://rgpdata.ensg.ign.fr/pub/data/%yyyy/%ddd/data_30/amb2%ddd0.%yyd.Z with %yyyy being the placeholder for a 4-digit year and %ddd being the placeholder for a day of the year.



☞ The URL preview always refers to the current day of the year, but it is just for illustration to show you that you entered path and file name correctly.

☞ When typing the address, beware of case sensitivity.

5. Confirm the address by selecting **OK**.

6. Back in the Property Grid, enter your **User Name** and **Password** if necessary to establish a connection.

7. Select **Create** to create the new, manually entered reference station.

Add to favourites:

To mark a selection of reference stations as favourites, select **Reference Stations** > **Add to Favourites** in the ribbon bar. In the Inspector view, the stations are marked with an asterisk ★.

Alternatively, right-click on a station and select **Add to Favourites** from the context menu.

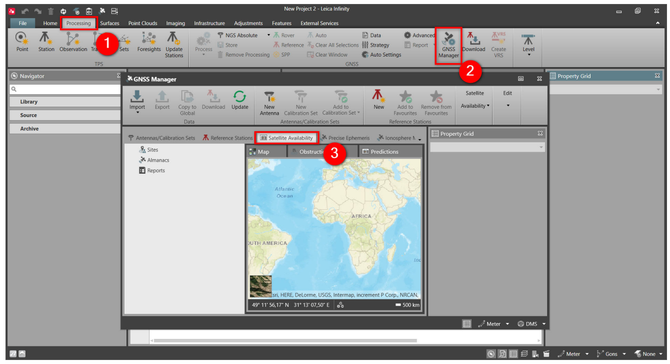
3.2.11.4 Satellite Availability

Satellite Availability

Satellite availability allows you to plan your GNSS field work. It provides you with graphical and numerical information on the satellite constellation for any location (site) at a given time.

You can create site-specific obstructions, to simulate real acquisition conditions to predict the satellites availability for single sites or for a combination of sites. For example, reference and rover receiver at the same time.

To open the satellite availability from inside a project:



1.	Select the **Processing** tab.
2.	Select **GNSS Manager** from the Processing ribbon bar.
3.	Select the **Satellite Availability** tab from inside the GNSS Manager.

To predict satellite availability for a site:

1. Right-click in the map view where you want to create a site and select **New Site** from the context menu.

   ☞ The new site Property Grid opens up next to the regular Property Grid. The coordinates are derived from the map.

   You can also select **New Site** from the ribbon bar, enter the coordinates in the Property Grid and select **Create**, to create the site.

2. Give the site a Name and select **Create** in the Property Grid.

   ☞ The site is added to the map view and to the list of sites in the Navigator.

3. Select the site and select **New Prediction**, either from the context menu or from the Satellite Availability tab.

   ☞ The new prediction Property Grid opens up next to the regular Property Grid.

4. In **Settings**, select a Start Date and an End Date for the prediction.

5. In **Almanac**, select **Automatic** to automatically download and import an almanac file.

☞ You can also import an almanac manually and select it from the drop-down list.

☞ Supported almanac file formats are *.mdb, *.yum and *.alm.

---

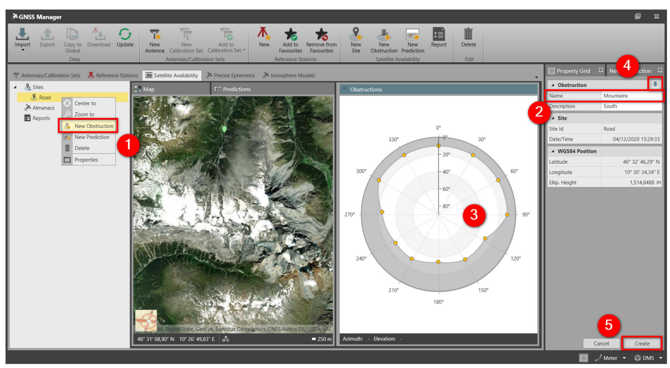
6. In **Obstructions**, select the obstructions you want to consider in the prediction.

☞ Only available, if you have already created obstructions for the specific site. See **To create new obstructions for a site:**.

---

7. Select **Create** at the bottom of the Property Grid.
Details of the new prediction are shown underneath the map view.
You get a skyplot plus information on:
- The satellite vehicles count.
- The satellite vehicles visibility.
- The DOP.
- The elevation values.
- The azimuths.
- The skyplot with the obstructions used in the computation.

To create new obstructions for a site:



1. Select the site, then select **New Obstruction**, either from the context menu or from the Satellite Availability tab.

☞ A new window opens to allow you to draw the obstructions.

---

2. Enter a Name for the obstruction.

---

3. Design your obstructions by drag and drop of the yellow points in the obstructions plot.

---

4. As an alternative, you can also import an obstruction file, by selecting **Import** in the Property Grid.

---

5. Select **Create** at the bottom of the Property Grid.

☞ The obstruction is added in the Navigator, under the site.

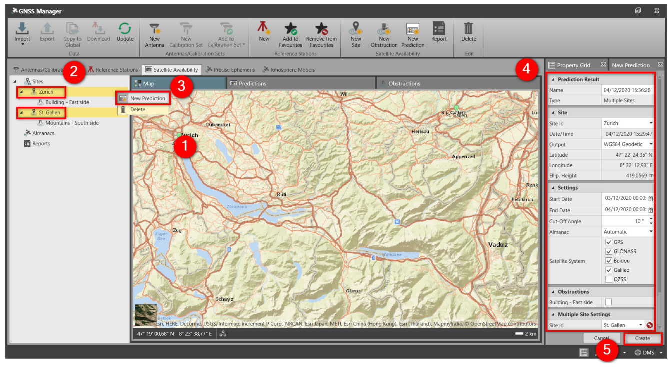
☞ Multiple obstructions can be created for a site. Repeat the steps to create new obstructions.

☞ To use one or more obstructions in the prediction computation, enable the related checkbox, as explained in step 6 of **To predict satellite availability for a site:**.

Predicting satellite availability on multiple sites

To compute the baseline between a reference and a rover receiver, only the common satellites at both sites are considered. In this respect, especially for long baselines, it is useful to have a tool which predicts the satellite availability at multiple sites.

To make a new prediction considering multiple sites:



1. Create the sites and the obstructions you want to consider.
2. Highlight all the sites you want to consider in the prediction.
3. Select **New Prediction** from the context menu.
4. In the Property Grid, select the sites where you want to compute the prediction. Go through the settings and obstructions to use for all the sites.
5. Select **Create** at the bottom of the Property Grid.
   ☞ The results are added in the Navigator.

See also:

The tutorial **"How to predict Satellite Availability for a site"** https://leica-geosystems.com/-/media/599ca3a5148d41bb8821e3d02d1a674a.ashx

☞ The tutorial can be downloaded in the Localisation Tool.

3.2.11.5 Precise Ephemeries

Precise Ephemeries

Manage Precise Ephemeris

Import Precise Ephemeris:

Precise ephemeris files must be in the following format:

- NGS/NOAA SP3-P (Position) format.
  The SP3-P format is an internationally accepted standard ASCII format for precise ephemeris.

There are several services that provide precise ephemeris data, like for example:

- IGN Global Data Center.
- IGS International GPS Service for Geodynamics.

Download Precise Ephemeris from the Internet:

Alternatively, you can make use of the Internet download function to download and import precise ephemeris data. Downloaded data is imported automatically.

1.	Go to the Precise Ephemeris tab.
2.	In the ribbon select the Download option. Next to the Property Grid a Download tab opens up.
3.	Select the date and time that fits your data.
4.	Choose another download directory if you want. By default, the download directory is your Downloads directory or the project folder if available.
5.	Select the **Start** option at the bottom of the tab to initiate the download.
6.	Follow the download progress in the Status section.
7.	In case of errors check the availability of the files to be downloaded. Availability can comfortably be checked by clicking the hyperlink on the file name.
8.	When the download is complete the data is automatically imported and listed in the Precise Ephemeris tab.

☞ Infinity offers a shortcut to access Internet downloads.
You can also select the Download option in the GNSS Processing ribbon bar. The GNSS Manager opens in the Reference Stations tab.

3.2.11.6 Ionospheric Model

Ionospheric Model

Manage Ionospheric Models

Import Ionospheric Models:

Ionospheric model files must be in the *.ion format.

There are several services that provide ionospheric models, like fo example:

- AIUB (Astronomical Institute University of Bern).
- IGS International GPS Service for Geodynamics.

Download Ionospheric Models from the Internet:

The daily ionospheric files from the University of Bern can be downloaded from: http://ftp.aiub.unibe.ch/CODE/.

To download and import the files automatically proceed as follows:

1.	Add a new reference station with URL: http://ftp.aiub.unibe.ch/CODE/%yyyy/COD%wwww%d.ION.Z with %wwww being the placeholder for a 4-digit GPS week and %d being the placeholder for the day of the week (Sunday = 0, Monday = 1 for example). On how to add a new reference station see Reference Stations.
2.	Once defined select the station and select **Data > Download** in the ribbon bar any time you need ionospheric files for a specific day.

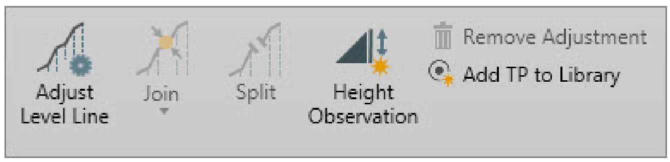
3.	In the Download tab define the period for which you want to download data and select the **Start** option at the bottom of the tab to initiate the download.
4.	When download is complete the data is automatically imported to your project. The ionospheric models are added to the list in the Ionospheric Models tab.

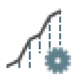
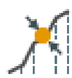
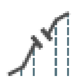
3.3 Level-Processing

3.3.1 Overview

Level-Processing

Level-processing is a purchased option. With this functionality, you can edit and process the level lines but also perform 1D network adjustments in support of level networks.



	Adjust Level Line opens the level line processing dialog to adjust the heights for the line.
	Join allows you to connect two or more level lines to a single level line. The lines are added in order of selection.
	Split allows you to divide a level line into two individual lines. This is important when considering level loops are not participating in a least squares adjustment.
	Height Observation is used to enter observed height difference between two points.
	Add Turning Points (TP) to Library is used for adding turning points to be used for other project data.
	Remove Adjustment resets the heights of the level line

Editing Level Lines

The height of a level line start or end target can be edited.

See also:

The tutorial **"How to process level data"** https://leica-geosystems.com/-/media/e1114ca40fda4640a2afc19292063ea8.ashx

The tutorial **"Working with level lines"** https://leica-geosystems.com/-/media/0e8524beb85c40ab926eec63b1ecc226.ashx

☞ Tutorial data can be downloaded in the Localisation Tool.

3.3.2 Split Level Line

Split Level Line

In some cases its required to split a level line. When considering the data for network adjustments, a level line which starts and ends on the same point must be split to be at least two lines.

To split a level line, select the turning point from the booking sheet or the level line profile view:

1.	From the ribbon bar select the **Split** option.
2.	From the right-click context menu select **Split**.

3.3.3 Join Level Line

Join Level Line

In some cases its required to join level lines.

To join a level line, select the lines to be joined.

1.	From the ribbon bar select the **Join** option.
2.	From the right-click context menu select **Join**.

Multiple lines can be joined at the same time. The lines are joined by order of selection, with the second and subsequent lines added in selected order.

Remember, when considering the height observations for network adjustments, a level line which starts and ends on the same point must be split to be at least two lines.

3.3.4 Height Observation

Height Observation

You can manually enter a level height difference observation in the **Processing** ribbon bar > **Level** > **Height Observation**.

The entered height observation is displayed from the source under the manually entered node.

The height difference together with its absolute standard deviation and relative standard deviation can be used for network adjustments.

3.3.5 Remove Adjustment

Remove Adjustment

For any level line that has been processed, it is possible to remove the height adjustment that was applied to the turning points.

Select the level line and with context menu or ribbon bar select **Remove Adjustment**.

3.3.6 Level Line Data

3.3.6.1 Overview

Level Line Data

You have the ability to import level data, edit the starting height and create reports in a booking sheet format.

Importing Level Data

The Infinity projects support levelled heights data, which can be imported from GSI, LEV or HeXML data formats.

All level jobs imported to a project are listed from source in the navigator. Each job indicates the level line data or level measurements that was imported by that job. Each job can hold many level lines or measurements.

Level Lines

If the level data includes position and height information, then the level line is displayed in the graphical view.

From the Level tab/Level lines subtab of the Inspector, you find all data from level group. From the level group, you can view the list of all level line data. Each level line can be drilled in to, which displays the level data in a booking sheet format.

Level Measurements

From the Level tab/Measurements subtab of the Inspector, youl find all measurements. Measurements are listed by imported jobs. Drill into display measurement booking sheet.

Level Data in a Project

Symbol	Description
	Unprocessed level line.
	Processed level line.
	User entered start point - Indicates the starting height of the level line as entered on a level.
	Control Point start point - Indicates the user selected a fixed control point as the height to start the level line.
	Level measured - These are the turning points of the level line or level measurements. By default, these points are not displayed in the points list. To convert a turning point or measurements to be available in the project, you need to convert that point to a library point.

3.3.6.2 Level Lines

Level Lines

Level data can be imported to a project. Its possible for you to edit the name of a level line, as well as the starting height.

Reports in a booking sheet format can be made for the level lines.

Level Line List

Level Line Variable	Description
Level Line ID	Identification of the lines contained in a job.
Method	Observation method used with a line. The available methods are: BF: BS-FS BFFB: BS-FS-FS-BS BBFF: BS-BS-FS-FS BFBF: BS-FS-BS-FS aBF: alternating BS-FS aBFFB: alternating BS-FS-FS-BS aBBFF: alternating BS-BS-FS-FS aBFBF: alternating BS-FS-BS-FS
Staff One/Two ID	Identification of the two staffs that have been used for measuring the level line.
Start Point	The point ID for the first measured point of the level line.
End Point	The point ID for the last measured point of the level line.

Level Line Variable	Description
Stations	Number of stations used in the level line.
Observations	Total number of observations used in the level line.
Delta Height	The height difference between start and end height.
Length	The length of the level line.
ProcessingMethod	When the line has been processed, which error distribution method was used.
Misclosure	The calculated height misclosure.
Height Error/Point	The height misclosure devided by the number of stations.
Total Dist Balance	Total distance balance considering all stations.
Total Station Diff	Indicates the sum of all station differences as computed at each station in the level line.

3.3.6.3 Level Line Details

Level Line Details

The level lines are shown from the inspector in a booking sheet format.

Level Line Details

Level Line Details	Description
Point ID	The measured point ID.
Type	The type indicates what sort of data the booking sheet line refers to - Backsight, intermediate or foresight measurements.
BS	The Backsight Staff (BS) reading. Depending on the observation method you have one or two backsights for one instrument setup. The starting BS measurement for a level line is always a library point and can be edited.
IS	Intermediate Staff (IS) readings. These are the points not part of the level line where a height value should be measured. Intermediate points are staff setups which are not part of the line but included in the line processing. The IS measurements are always library points and can be renamed.
FS	Intermediate staff readings. These are the points not part of the level line where a height value should be measured. Intermediate points are staff setups which are not part of the line but included in the line processing. The IS measurements are always library points and can be renamed.
Rise	When the BS - FS is positive.
Fall	When the BS - FS is negative.
Hz Dist	Distance between the level instrument and staff 1/2. Ideally the distances should be the same to cancel errors due to curvature and refraction.

Level Line Details	Description
Azimuth	For some XML imported level data there can exist preliminary coordinates and azimuth directions from the level stations to the BS-FS.
Easting	If the Easting has been recorded it can be displayed in the booking sheet.
Northing	If the Northing has been recorded it can be displayed in the booking sheet.
Height	The height of a point as calculated in relation to the start height. Control heights are fixed, while measured heights are updated when a level line is processed.
SD	This column shows the Standard Deviation (SD) of an observation recorded in repeat single, mean, mean+s or median measure mode. To edit one or more SD highlight one or more staff reading and select **Edit Standard Deviations** from the context menu.
Mean SD	The Mean Standard Deviation (Mean SD) is the mean of all the standard deviations calculated in the level line up to that point. It can be used for finding the average variation in the measurements, rather than the variation for each observation.
Measurements	Is the number of measurements taken by the instrument for that observation. This is displayed for measurement modes repeat single, mean, mean+s and median.
Spread	The spread is the difference between the highest and lowest observations in repeat single, mean, mean+s or median measure modes.
Mode	The measure mode indicates how the observations were taken. Most levels have single, repeat single, mean, mean+s and median measure modes.
Diff BS1-BS2	For BF sequence measurements the difference between the first and second backsights is recorded. Ideally the difference should be 0.
Diff FS1-FS2	For BF sequence measurements the difference between the first and second foresights is recorded. Ideally the difference should be 0.
Station Diff	The Station Difference (Station Diff) is used for BFFB (and aBFFB) sequence measurements. It is the sum of the differences in the backsights and foresights.
Total Station Diff	The Total Station Difference (Total Station Diff) is a running total of the station differences for every station in the level line up until that point.
Dist Balance	The distance balance is the difference between the distance to the backsight point and the foresight point.

Level Line Details	Description
Total Dist Balance	The total station difference is a running total of the station differences for every station in the level line up until that point.
Earth Curv Corr	Indicates whether the Earth Curvature Correction (Earth Curv Corr) has been applied to the observations.
Expansion Coeffecient	The expansion coefficient is the a value used when applying staff corrections during line processing. This value varies depending on the type of staff used.
Calibration Temperature	This is the temperature that the staff was calibrated.
Temperature	The temperature during measurement is the T value in the formula. Enter the actual temperature determined while you were observing the level line to be processed.

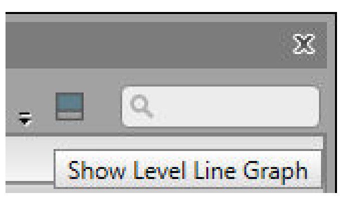
3.3.6.4 Level Line Graph View

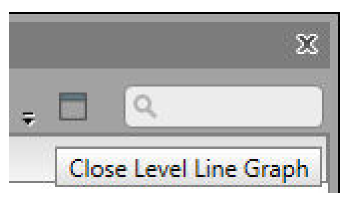
Level Line Graph View

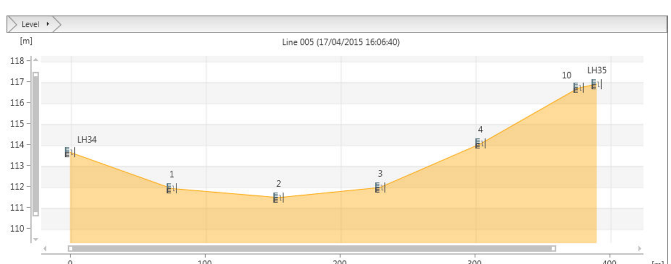
The level line view is a graphical representation of the data given in the booking sheet. The profile of the selected level line is visualised, for instance the rises and falls resulting from calculated point heights. Since the distances between the instrument setups and the staff setups are known a proportionally correct overview on the line structure may be provided.

From within the inspector the level lines list has by default the graph view.

To view the graph view of a level line when drilled into a line, use the level line graph toggle at the top right view:





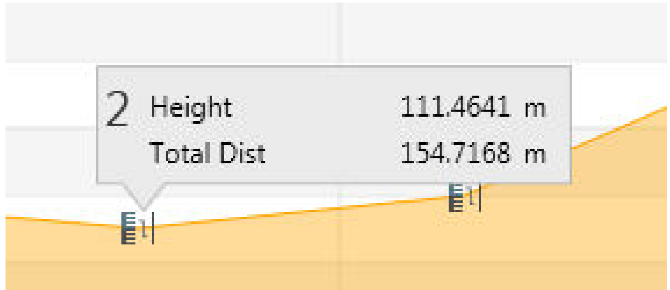


Navigating

You can zoom in/out and pan in the graph view. The mouse uses the same settings as when working in the main graphical display.

Selecting and viewing heights

Using the mouse you can hover and or select a turning point from the display.



3.3.6.5 Level Measurements

Level Measurements

Level measurements that are not part of the level lines are imported to the level tab/measurements subtab of the inspector. Level measurements belong to the imported job and you can drill into see the booking sheet.

To view and post-process the attributes of the imported data, go to the level tab/measurements subtab of the inspector.

Measurement details	Description
Point ID	The measured point ID.
Type	For the level measurements the type is always measure only.
IS	Intermediate staff readings.
HZ Dist	Distance between the level instrument and staff.

3.3.6.6 Using Turning Points

Using Turning Points

By default all turning points imported with the level lines are visible in the project data but they are not listed from the library points. In most cases the level lines are very long and in the case you need to use turning points for other project data, you can convert these points to become library points.

The start, end and all Intermediate Sideshots (IS) are by default library points.

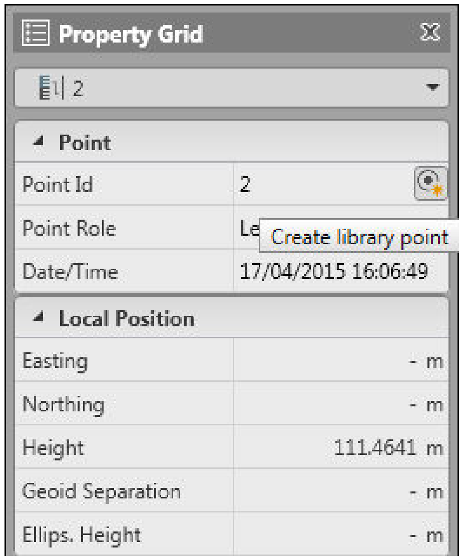
Create Library Point

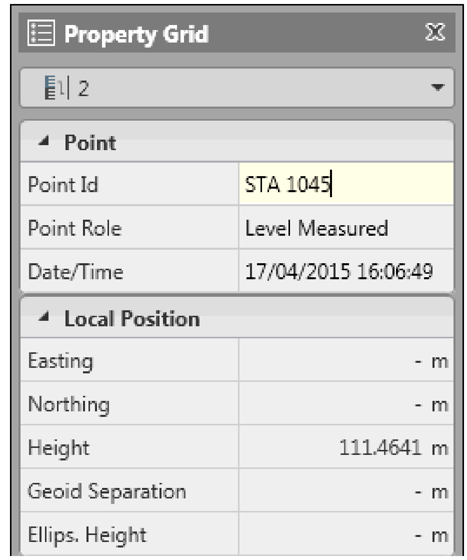
For any turning point that you want to use in the project, such as for the height of a TPS station, do the following:

1. Locate and select the point to view the data in the Property Grid.
2. Select the **Create library point** option in the Point ID field.
3. Select **Apply**.

Now you can see the point ID and it can be edited - If necessary, you can rename the point to use with other project data.

☞ When converting a turning point to a library point and you edit the point ID to be an existing point in the project, this level measured becomes part of the point averaging of the point.





3.3.6.7 Edit Level Standard Deviations

Edit Level Standard Deviations

With each level staff reading a standard deviation is stored and can be displayed in the booking sheet. Editing the standard deviation modifies the standard deviation of the levelled point height and the standard deviation of the height difference observation of the total level line.

To modify the standard deviation of level staff readings:

Highlight one or more lines in the booking sheet indicating level height readings and select **Edit Standard Deviations** from the context menu. Enter the value that shall be applied to the selected staff readings and select **OK**.
The standard deviations for the point height and for the total level line are updated.

3.3.7 Adjust Level Line

3.3.7.1 Overview

Adjust Level Lines

The calculation of level lines is done in a process level line window, that lets you set and verify the results before storing to the project.

To open the process level line wizard select Adjust Level Line:

1. From the ribbon bar Processing tab.
2. Right-click from the mouse context menu.
3. Select the option from the level line properties.

3.3.7.2 Process Level Line Settings

Process Level Line Settings

This page shows you the basic line information including a profile view.

The first step to process a level line is to:

1. Set the process method for how the height misclosure is distributed to the turning points.

2. Set if its needed to use Staff Corrections.

3. Define the line tolerances you want to flag the data for the quality control.

3.3.7.3 Process Level Line Calculate Information

Process Level Line Calculate Information

This page shows the entire line in a booking sheet format.

The second step to process a level line is to:

1. Define the start and/or end heights by selecting the point ID from the project data.
2. Select the ✏ option from the level line starting point ID to choose the starting height.
3. Select the ✏ option from the level line ending point ID to choose the ending height, if necessary. Scroll to the bottom of the line to do it.

When using an ending height, then the level line misclosure and height error/ point are shown on this page.

Select **Next** to continue to view and verify the results.

3.3.7.4 Process Level Line Results

Process Level Line Results

This page shows you the detail level line information including a list of the new computed heights that are stored when finishing the processing.

The last step to process a level line is to confirm the results are within expectations and review the data before storing.

3.4 Surfaces

3.4.1 Overview

Surfaces

Surfaces that have been created on the instrument can be imported into Infinity together with the measured raw data. Surfaces can be managed in the surfaces tab.

The surface tab allows you to:

- Create new surfaces by triangulation.
- Calculate volumes on the basis of existing surfaces.

In Infinity, the surfaces computation is performed using the 3DReshaper kernel licensed to Leica Geosystems AG by Technodigit SARL, 69400 Gleize, France. For detailed information, see https://www.3dreshaper.com.

Creating new surfaces:

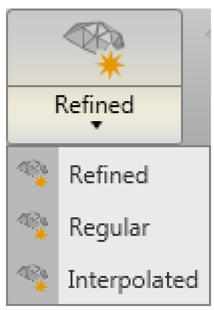
1. Select the entities from which you want to create a surface either graphically or from inside the Inspector or the Navigator.
   Surfaces can be created from Points or Point Clouds or Point Cloud Groups.

   ☞ There are two ways how you can select entities graphically. Either press **Shift** and **drag a rectangle** with your mouse to enclose a selection of entities or press **Ctrl** and draw a polygon (polygon selection). You cannot pick disconnected elements.

   ☞ In the case of the graphical view selection, only the highest point role is added. TPS or GNSS setup points, as well as deleted points are not added.

---

2. Select **New Surface** from the Surfaces ribbon bar.
   The surface type created is indicated in the ribbon icon.



☞ You can also create an empty surface and add points or point clouds to it later on.

The new surface is created and added to:

- The Surfaces section in the Library of the Navigator. If the Surfaces subsection does not yet exist, it is created.
- The Surfaces section in the Surfaces subtab of the Inspector.

☞ Surfaces are always created with a default name. To change the name, click onto a surface in the Navigator or in the Inspector and adapt its name in the Property Grid.

☞ To see only the surfaces triangles in the graphical View, you can switch off the underlying point cloud groups through the **eye** in the Navigator.

To add or remove points or point clouds to or from a surface:

1. In the Inspector, go to the **Features** tab and select the points or the point clouds or point cloud groups to be added to or removed from a surface.
   In the Navigator, points can be selected from the Library section.
   Point clouds can be selected from inside the Source section or from the Library section if they belong to a point cloud group.
   Alternatively, you can also select points or point clouds in the graphical view.

---

2. Select **Add to** from the Surfaces ribbon bar and select a **surface** from the drop-down menu or select **Add to Surface...** from the context menu.
or:

Alternatively, go to the **Surfaces** tab and select **Remove from** from the ribbon bar to select a **surface** from the drop-down menu or select **Remove from...** from the context menu.

☞ Points can also be removed from inside the Surfaces subtab of the Inspector. Drill down into the surface from which one or more points shall be removed and further down into points. Select the **points** to be removed.

☞ Only points identified by point ID (so-called Library points) can be selected to be added or removed from a surface. Cloud points are by default not identified by any point ID and can thus only be selected to be added or removed from a surface individually if either library point have been created from them before or when they belong to a line or area. You can select a subset of cloud points, though, graphically and add or remove it to or from a surface.

Adding and removing lines/areas to or from a surface:

In Infinity you can add lines as breaklines and areas as exclusion areas or to an existing surface. A single area can also be added as a boundary, with the effect that the border of the area becomes the new boundary of the surface.

Any line/area in your project, for example a line defining the sideline of a road or the top of a ridge or an area defining the contours of a building, can be added to a surface as a breakline, exclusion area or boundary.

This forces the triangulation of your terrain model to be rebuilt by taking the line/area into account. The line/border lines of an area become triangle edges. As a result, there is not any incorrect interpolation across the resulting breakline, exclusion area or boundary.

To add a line/area to a surface:

1. Right-click onto the line/area to be added to the surface either in the graphical view or in the Navigator or in the Inspector.

2. For lines, select **Add to Surface as Breakline** from the context menu.

   For areas, select either **Add to Surface as Exclusion** or **Add to Surface as Boundary** from the context menu.
   If more than one surface exists in your project, select the surface to which the line/area shall be added.

   Alternatively, go to the **Surfaces** tab and select **Add to** from the ribbon bar to select a **surface** from the drop-down menu. For areas, decide whether to add the area as Exclusion or Boundary.

☞ It is possible to add only one outer boundary to an existing surface.

☞ It is not possible to add different objects at the same time to a surface.

To remove a line/area from a surface:

1. Right-click onto the line/area to be removed from the surface either in the graphical view or in the Navigator or in the Inspector.
2. Select **Remove from...** from the context menu. If the line/area has been added to more than one surface, select the **surfaces** from which the line/area shall be removed.

   Alternatively, go to the **Surfaces** tab and select **Remove from** from the ribbon bar to select a **surface** from the drop-down menu.

3.4.2 Surface Types

Surface Types

In Infinity you can create surfaces from point sources. Working with point cloud data, the results of surfaces are directly effected by the properties of the point cloud data - Point density and the location from where scan measurements are made from can result in holes and effects where the surface routine cannot determine the above or below surface of the triangles to be computed. To manage the situation of cloud point properties in the best way, Infinity offers four triangulation methods to better determine which method provides the best results.

The three surface types are:

- Refined.
- Regular.
- Interpolated.
- 2.5D.

The surfaces methods are based on the 3DReshaper engine. There are many parameters that are already considered and defined with the three methods that Infinity uses.

Refined

Use this method:

- For clean point cloud data that is consistent in density.
- Uses the meshing parameters and then refines the mesh to generate the most details.

This surface method is created with following steps:

Determine initial mesh:

- Minimum triangle size (regular sampling) example used 0.08 m.
  - Consider that if set to zero, all points are used.
- Maximum triangle size (hole detection) example used 3.0 m.
  - Sets the size for which the maximum triangle is created to fill gaps.
  - A hole remains when the points are not geometrically consistent (for example, the surface does not know which direction it should follow, change of direction is high).

Hole filling:

- Not adding new points, or curvature filling.

Refine the mesh:

- Takes the points of the cloud and compares to existing mesh.
- Fit the mesh adding triangles where the deviation is greater than 0.01 m.
- Remove points that are too far from the mesh.

Remove spikes:

Regular

Use this method:

- For clean scan data but where point density is less consistent, like for long distance between near and far cloud points.
- Uses the meshing parameters min/max triangle size, and appears most evenly.
- This surface can be edited in the Properties - Change the min/max values and apply.

This surface method is created with following steps:

Determine initial mesh:

- Minimum triangle size (regular sampling) example used 0.08 m.
    - Consider that if set to zero, all points are used.
- Maximum triangle size (hole detection) example used 3.0 m.
    - Sets the size for which the maximum triangle is created to fill gaps.
    - A hole remains when the points are not geometrically consistent (for example, the surface does not know which direction it should follow, change of direction is high).

Hole filling:

- Not adding new points, or curvature filling.

Remove spikes:

Interpolated

Use this method:

- For scan data that is noisy and not consistent.

This surface method automatically calculates the meshing parameters to interpolate a best surface.

2.5D

Use this method:

- For points that are distributed on the ground and not representing a 3D object.
- Ensure that every point is used as a vertex of the surface.

This surface is created with the following steps:

- If point cloud points are part of the selection: Use the minimum triangle size from the surface parameters in order to reduce the point cloud to respect a minimum distance between points of this value. A value of 0 would result in all point cloud points to be included in the surface.
- Create a 2.5D surface with all library points and, if applicable, with all the remaining points left over from the previous step.

☞ The same method used in Infinity v1.1. Projects with surfaces from the previous versions can be used in Infinity v1.2. It is also possible to generate the same surface as previously possible.

3.4.3 Surface Properties

Surface Properties

In surface properties you can:

- Change the name (ID) of the selected surface.
- Assign the surface to a different layer.
- Change the styling independent of the overall layer style.

In the details section you are informed about:

- The total number of triangles, triangulation points and triangle edges.
- The total number of breaklines and exclusion areas that have been added to the selected surface.
- The height of the topmost triangulation point (Max. Height) and the height of the lowest triangulation point (Min. Height).
- The sum of all triangle areas (Area).
- The area covered by a surface projected to the ground plane (Area 2D).
- The perimeter as defined by the outer boundary of the selected surface (Perimeter).
- The perimeter as projected to the ground plane (Perimeter 2D).

3.4.4 Surface Contours

Surface Contours

It is possible to create contours for a selected surface.

- Select a **surface**.
- From the ribbon bar, select **Contour**.
- From the Create Contour window, edit the settings to define the contour display.

Create Contour

When creating a set of contours for a surface the following settings must be considered:

Surface

- Displays the selected surface including the height details.

Contours

Set the values to determine the drawing of the contour:

- **Minor Interval**, defines the distance between minor contours that are drawn.
- **Major Frequency**, defines when the major contour is drawn. Usually this setting would be set at 5 or 10.

With a minor interval of 10 and a major frequency of 5, the contours are draw every 10 metres but at every 50 metres a major contour line is drawn.

- **Min. Length**, defines the minimum contour line length to be displayed. Any shorter lines are not displayed.
- **Smoothing**, allows the contour lines to be smoothed with different intensity values.

Contour Styling

- Define the line style and colour settings.

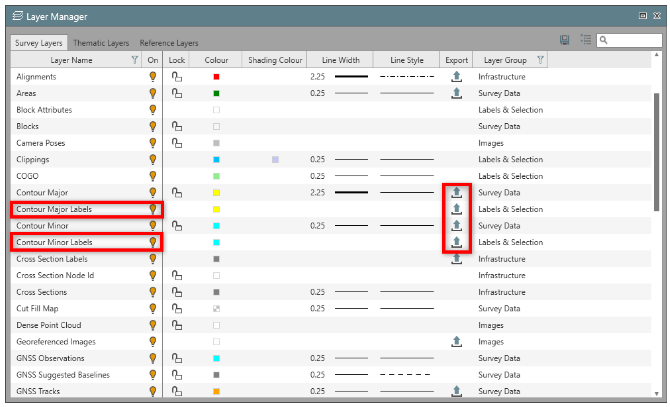
Contour Labels

To display the labels, you must switch them on in the Layer Manager.

Exporting Contours

Contours can be exported to DXF/DWG and to XML.

Set the export flag in Layer Manager to determine if the contours are written to the DXF/DWG file.



3.4.5 Volume Calculations

Volume Calculations

For calculating volumes a reference surface, a ground surface and, depending on the selected calculation method, an intersecting plane must be defined.

Reference surfaces:

A reference surface is the surface of the stockpile/cavity for which a volume shall be calculated. Cavities are negative stockpiles.

☞ If you have selected a surface before in the Inspector or from inside the Library section of the Navigator then the selected surface is automatically taken as the reference surface for which you want to calculate the volume.

Ground surfaces:

A ground surface for simple stockpile computations is the surface at the bottom of the selected stockpile which is created with triangulation among the points which define the outer boundary of the stockpile. For cavities the ground surface is the surface covering the cavity. It is computed in the same way as for stockpiles. If you choose to compute a Surface to Surface volume then the ground surface is the second selected surface (for example, the surface that you select under To Surface in the Surface to Surface method, for further information on this method read the following).

Intersecting planes:

To define an intersecting plane for a volume calculation choose a point from your current project or define a height through which a horizontal plane shall be calculated for intersecting the surface. Volumes are calculated above and below the intersecting plane.

Stockpile:

1. Select the **Stockpile** option in the **Surfaces** ribbon bar. A window opens called Volumes: Stockpile.
2. In the Input section select the option to select the **Reference Surface** for the calculation.
3. Select the **Calculate** option at the bottom of the Volumes: Stockpile window to compute the Stockpile Volume. The result is shown to you in a Results section.

To Point:

1. Select the **To Point** option in the **Surfaces** ribbon bar. A window opens called Volumes: Surface to Point.
2. In the Input section select the option to select the **Reference Surface** for the calculation.
3. In the From Point section select a **point** in your project that shall serve as the reference point for calculating the intersecting horizontal plane.
4. Select the **Calculate** option at the bottom of the Volumes: Surface to Point window to compute the volume above and below the height of the selected reference point. The result is shown to you in a Results section.

To Height:

1. Select the **To Height** option in the **Surfaces** ribbon bar. A window opens called Volumes: Surface to Height.
2. In the Input section select the option to select the **Reference Surface** for the calculation.
3. In the Height section enter a **height** that shall serve as the reference height for calculating the intersecting horizontal plane.
4. Select the **Calculate** option at the bottom of the Volumes: Surface to Height window to compute the volume above and below the selected height. The result is shown to you in a Results section.

Surface to Surface:

1. Select the **Surface to Surface** option in the **Surfaces** ribbon bar. A window opens called Volumes: Surface to Surface.
2. In the Input section select the option to select the **Reference Surface** for the calculation.
3. In the To Surface section select the option to select a **surface** from your current project that shall serve as the ground surface for the calculation.

4. Select the **Calculate** option at the bottom of the Volumes: Surface to Surface window to compute the volume between the selected surfaces. The result is shown to you in a Results section.

For all four methods you can:

Select a **different Input** data and run another volume calculation within the same Volumes window. The result is added on top of the results computed before. Each Results section can be collapsed or closed individually.

Select the **Report** option at the bottom of the Volumes window to generate a report on the Results.

3.4.6 Comparison Maps

Comparison Maps

Comparison maps is a surfaces function that allows the comparison of a reference feature with a measured feature. The differences are displayed in the graphical view, as well as in the property grid of the calculated comparison map.

Reference features can be:

- A surface.
- One point cloud.

☞ Measured type must be point cloud/group as well.

- A point cloud group.

☞ Measured type must be point cloud/group as well.

- A plane:
  - Horizontal.
  - Vertical.
  - Free.
- One or more Industry Foundation Classes (IFC) objects (solids).
- An area.
- A closed line.

Measured features can be:

- A surface.
- One or more point clouds.
- A point cloud group.
- One or more library points.

Differences can be calculated along the height axis or in 3D, perpendicular to the reference object.

The differences are then colour-coded and shown in a so called comparison map. The comparison map is saved to the database as a Infinity object. By default, it shows in blue volumes, distances and areas that are above the reference feature. Volumes, distances and areas that are below, are indicated in red.

☞ The settings can be changed any time before and after the creation of the map in the Styling settings.

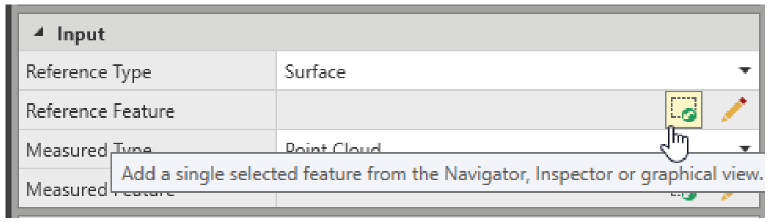
How to calculate a comparison map:

1. Go to the **Surface** tab and select **Comparison Map** from the Surfaces ribbon bar.
   The Create New Comparison Map window opens up next to the regular property grid.

2. Select the Reference Type you want to compare against (design). Once set, you can choose the feature itself.

☞ Not every feature allows a multiple selection.

☞ The selection can be done using a pencil flyout or by selecting the **Add current selection** option. In order to use it, select the required features from the Navigator, Inspector or graphical view, before selecting the option.



3. Select the Measured Type that represents the captured data. Afterwards, choose the required measured features in the same way as the required reference features.

☞ The order of adding the data is not relevant.

☞ For library points, it is also possible to select multiple points within the pencil flyout. Deselecting points can be done by keeping the Ctrl key pressed and a left-click of the mouse.

4. Under Settings, you can choose to compute only differences or check your data against tolerances.
If you select tolerances, specify the upper and lower tolerance values and define a minimum segment distance. Through the segment distance value, you can simplify the tolerance lines.

☞ Tolerance lines indicate the areas in the map where material has to be added or removed. These lines can be exported for stakeout, but if they are too detailed the controller might not be able to handle them.

5. Also under the Settings, you can specify the inspection direction - If you want to calculate distances along the height axis or perpendicular to the reference feature. Then the direction is variable and depends on the shape of the reference feature.

6. If the data set you are comparing has large extends, you can limit the range by activating the ignore distances larger than option. Define the maximum distance between the reference and the measured feature.

☞ You can also limit point clouds by clipping data or performing a manual selection in the graphical view.

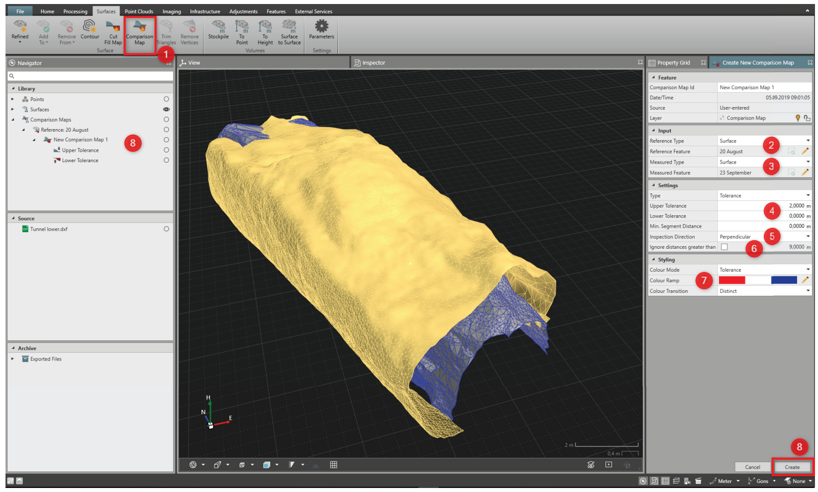
7. Under Styling, define which colours shall be used for indicating above/below values.

☞ Default values for Settings and Styling can be defined in the surfaces Parameters option.

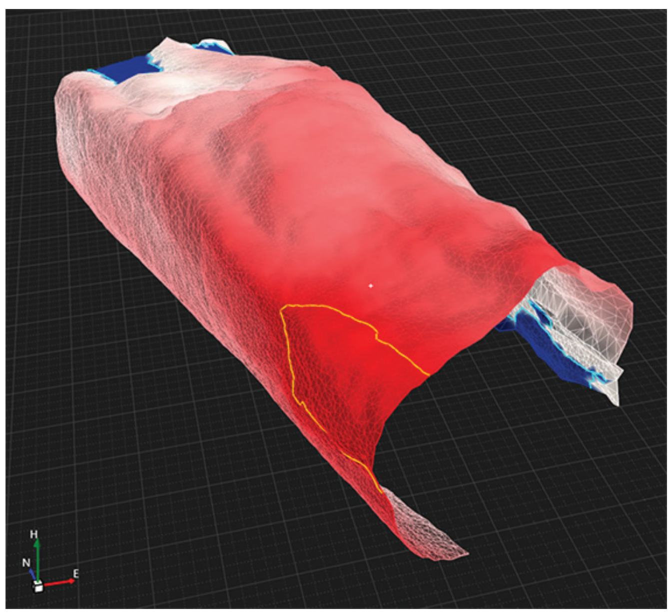
Select **Parameters** from the Surfaces ribbon bar to open.

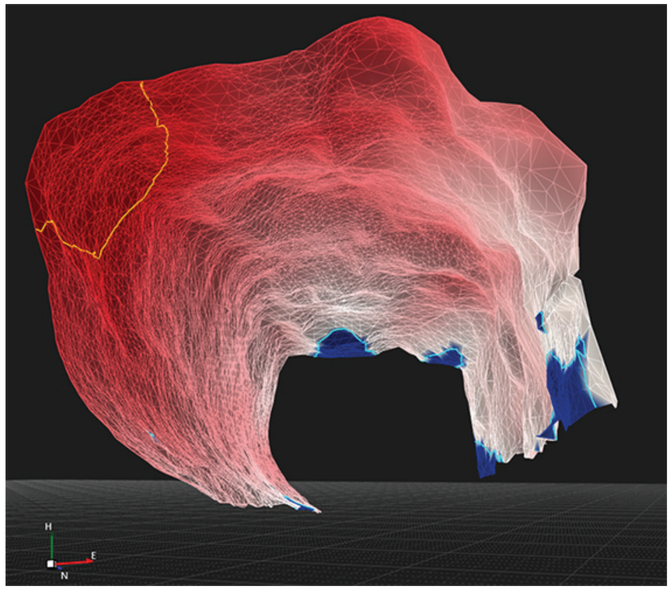
☞ These settings are shared with the cut fill map.

8. Select the **Create** option to create the new comparison map.
A new comparison map object is created and visible in the graphical view, as well as the Navigator, Inspector and Property Grid.



Comparison map tolerance mode results including tolerance lines at 0 difference (cyan lines) and -2m difference (orange line):





Using Point Cloud as Reference

One reference point cloud or group can only be compared to other point clouds or groups. Comparing the same point cloud with itself or a part of it is not allowed.

Inspection Direction set to Along Height Axis: Infinity displays values below (negative) and above (positive) the reference point cloud/group.

☞ The comparison map might have a low point density, when the reference or measured point cloud is dense in some areas and sparse in other areas. In this case, the result can be improved, if the measured point cloud is windowed in the graphical view.

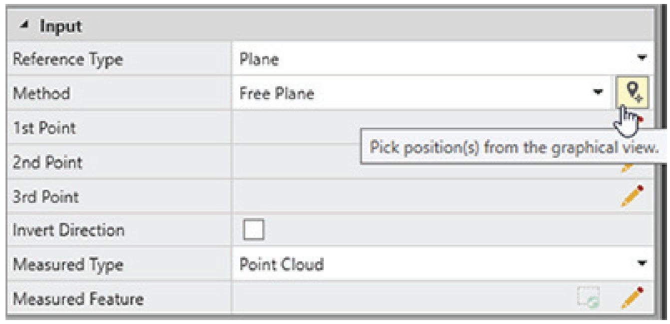
Inspection Direction set to Perpendicular: Infinity displays absolute distances between the two-point clouds/groups. All distances are positive, independent of being above or below the reference.

Using Planes as Reference Features

There are different methods of creating a plane:

- Horizontal: Select a height of a point or enter a height yourself. The inspection direction is fixed along the height axis.
- Vertical: Select two points to define the vertical plane. The inspection direction is fixed to the perpendicular.
- Free: Select three points to define a 3D plane.

Points can be chosen from the flyout or directly from the graphical view. The plane is created only once all required points are selected. It is possible to edit the plane as often as you like, until you select the create option.



☞ The inspection direction can be inverted with the checkbox. You can define which values shall be positive or negative.

After the creation of the comparison map, the plane also appears in the graphical view and navigator. You can turn it off by using the visibility eye or switching off the comparison map reference planes layer from the layer manager.

Inspecting the Comparison Results

Depending on the measured feature selected, you see the distances in the graphical view in the shape of a surface, point cloud or lines (library points).

In the Property Grid select the ⓘ option to see a histogram of the calculation results.

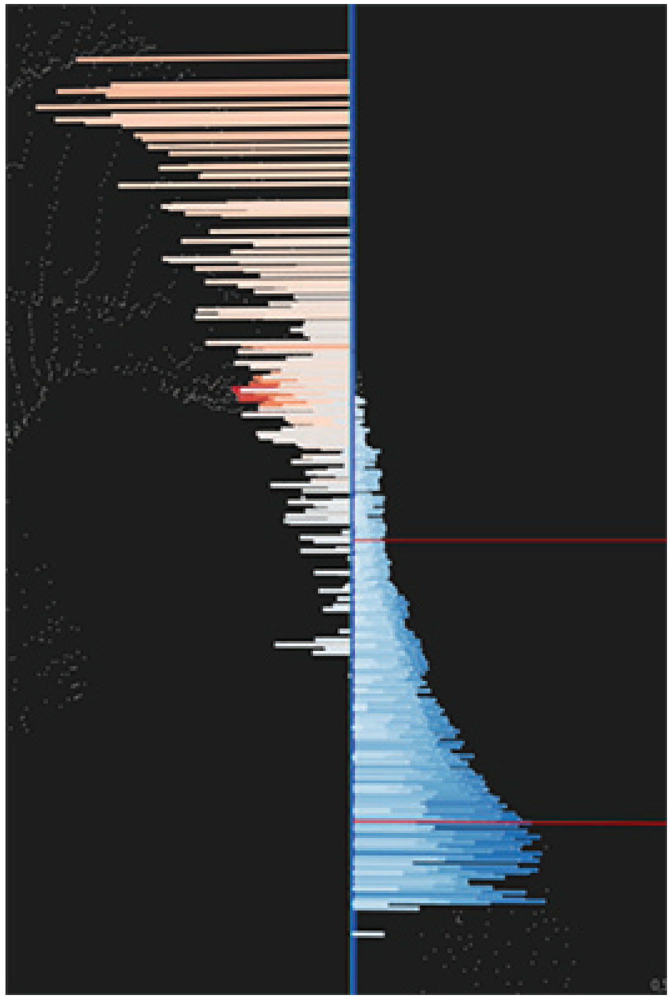
To generate a report, on one or multiple comparison maps, select the maps in the Navigator or the Inspector and either select: **Reports** > **Comparison Map Report** from the Home ribbon bar or right-click and select **Comparison Map Report**.

For best viewing in the graphical view, it is recommended to turn off the visibility of the measured and reference feature. This allows to focus on the comparison map itself.

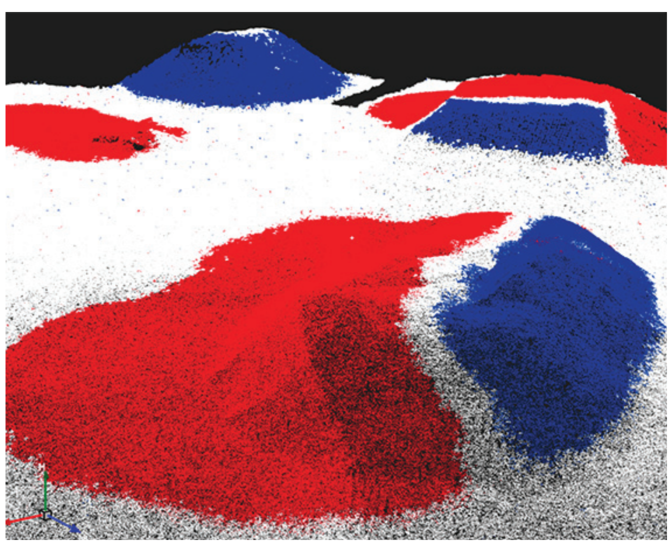
☞ If you choose a plane as a reference, then you can also turn off its visibility from the navigator.

☞ In the Layer Manager, you can change the width of the lines for the representation of distances between a reference feature and library points.

Measured Type = Library Points:



Measured Type = Point Cloud:

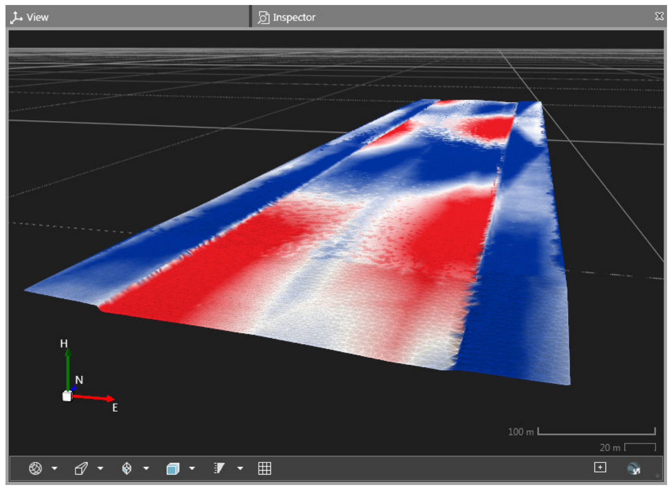


3.4.7 Cut Fill Maps

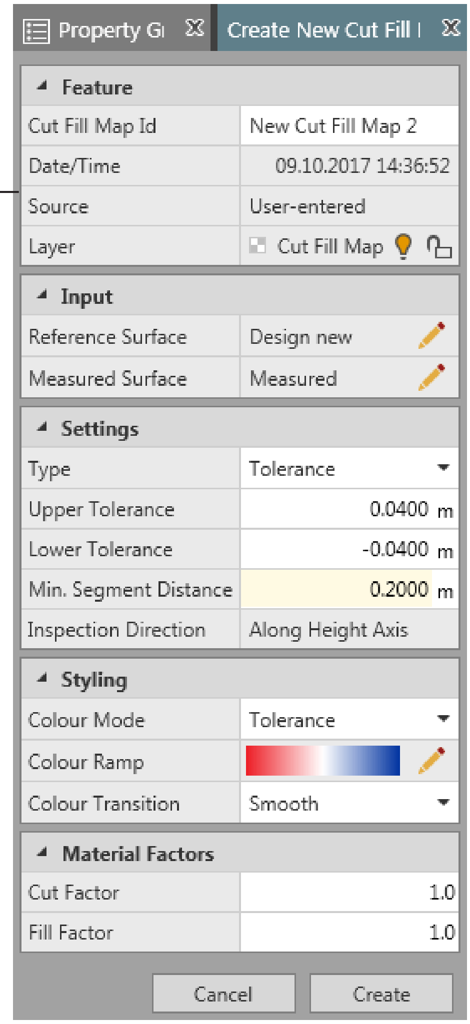
Cut Fill Maps

Cut fill maps is a surfaces function that allows you to compare a measured surface to a design surface, called reference surface in Infinity. Height differences are colour-coded and shown in a so-called cut fill map. The cut fill map is saved to the database as an Infinity object. By default, it shows in blue volumes that need to be cut to reach the level of the reference surface. Volumes that need to be filled are indicated in red.

Sample of a road design:



How to calculate a cut fill map:



1. Go to the **Surfaces** tab and select **Cut Fill Map** from the Surfaces ribbon bar.
   The Create New Cut Fill Map window opens up next to the regular property grid.

2. Select a **Reference Surface** (design) and a **Measured Surface** as input to the cut/fill calculation.

   ☞ You can also first select a Reference Surface (design) and a Measured Surface in the Navigator or the Features Inspector and then select **New Cut Fill Map** from the context menu. In this case the property grid opens up and reference and design surface is already selected. The surface you select first is identified as the reference surface. The second surface you select is automatically identified as the measured surface.

3. Under Settings select whether you want to compute only height differences or height differences with respect to given tolerance values.
   If you select Tolerance, specify the upper and lower tolerance values, too and define a Min. Segment Distance. Through the Min. Segment Distance value you can simplify the tolerance lines.

   ☞ The tolerance lines indicate the areas in the map where material has to be cut or filled. These lines can be exported for stakeout, but if they are too detailed the controller might not be able to handle them.

4. Under Styling define which colours shall be used for indicating cut/fill areas.
By default, blue is used for volumes that need to be cut and red is used for volumes that need to be filled.

   ☞ Default values for Settings and Styling can be defined in the surfaces Parameters option.
   Select **Parameters** from the Surfaces ribbon bar to open.

   ☞ To reset Settings and Styling to the factory defaults go to the **File** tab > **Info & Settings** > **Surfaces & Contours** and select the **Defaults** option.

5. Under Material Factors you can enter factors by which the cut/fill volumes shall be multiplied. The material factors are used to better estimate the volume of material to move.

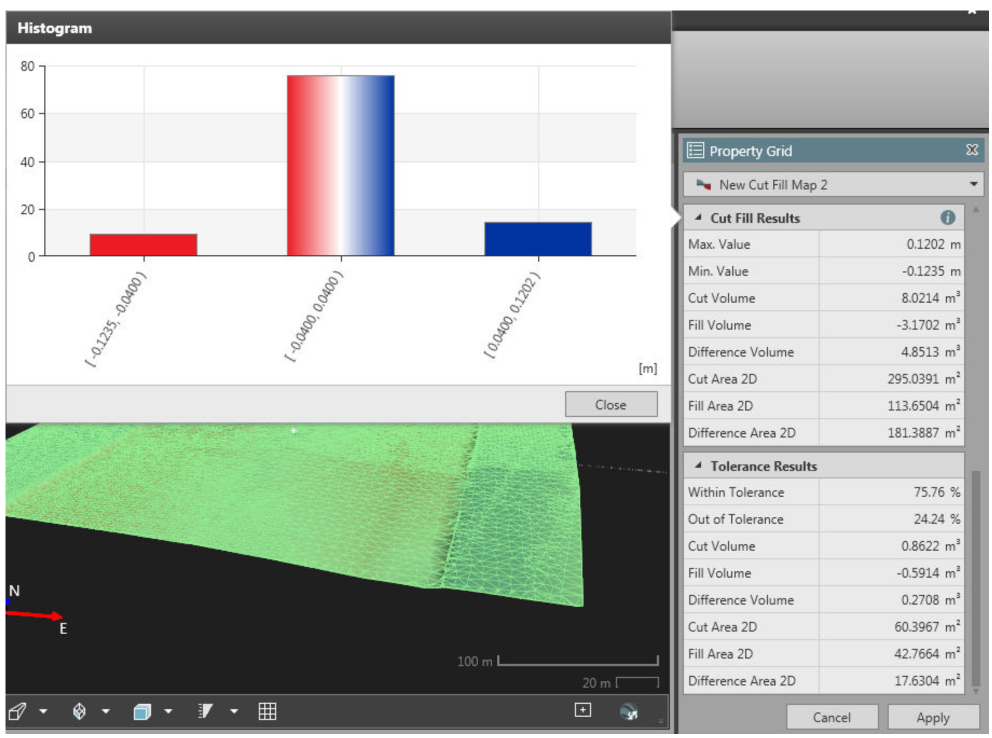
6. Select **Create** at the bottom of the property grid to create the new cut fill map.

Cut Fill Results and Tolerance Results

If tolerances have been used for the calculation, they are added as properties of the cut fill map to the Property Grid and the Features Inspector.

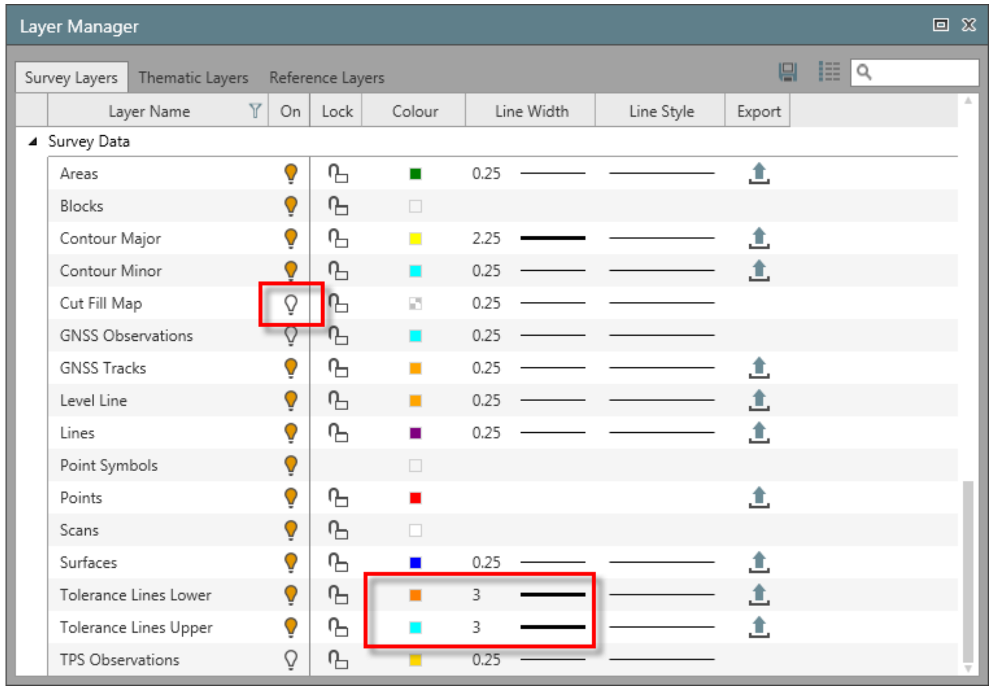
Inspecting cut fill results:

- In the Property Grid select the ⓘ option to see a histogram of the calculation results.



- In order to generate a report on one or more cut fill maps, select the maps in the Navigator or the Inspector and either select **Reports** > **Cut Fill Map Report** from the Home ribbon bar or right-click and select **Cut Fill Map Report** from the context menu.
- In order to focus the graphical view on just showing the cut fill map make other objects invisible by selecting the 👁 option in the Navigator.

- In the Layer Manager you can select different colours for the so-called Tolerance Lines Upper and Tolerance Lines Lower to enhance their visibility in the graphical view. You can also adapt the line width or change the line style. If you switch off the Cut Fill Map itself, you can focus on the tolerance lines in the graphical view.

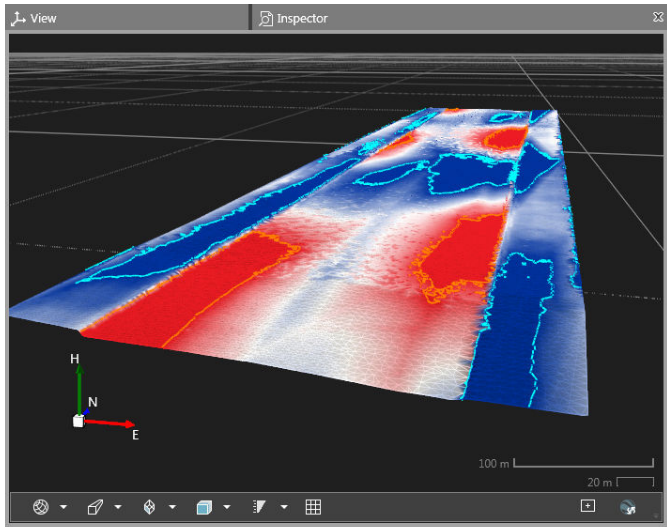


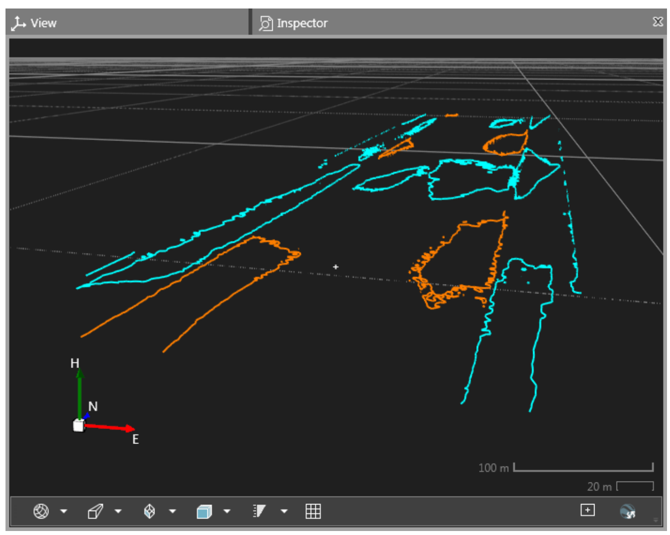
Workflow

A typical workflow making use of cut fill maps would be:

1. Measure, for example, a road that is being built based on a given design.
2. Import the measured data and the design surface from CAD to Infinity.
3. Calculate the surface resulting from the measurements.
4. Compare the surface to the design by calculating a cut fill map. Here you can make use of tolerances by entering the values. The cut fill map only shows areas and volumes to be cut or filled that are beyond the entered tolerance values.
   The areas where material needs to be moved are indicated by contour lines called Tolerance Lines Lower and Tolerance Lines Upper. These contour lines can be exported to DXF.
5. Use the exported upper and lower tolerance lines for stake-out of the out-of-tolerance areas in the field.

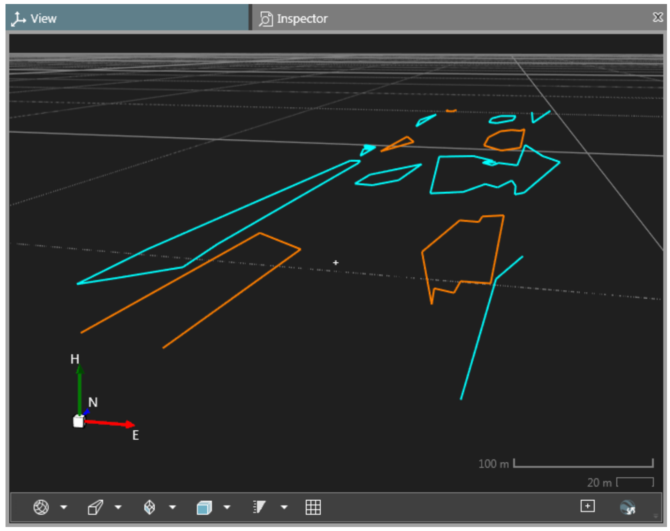
Illustration showing the tolerance lines upper and lower where material still needs to be moved to reach the design surface:





The styling of the tolerance lines has been modified in the Layer Manager to enhance their visibility against the cut fill map itself.

To use the lines for stake-out in the field make use of the Min. Segment Distance to simplify the lines:



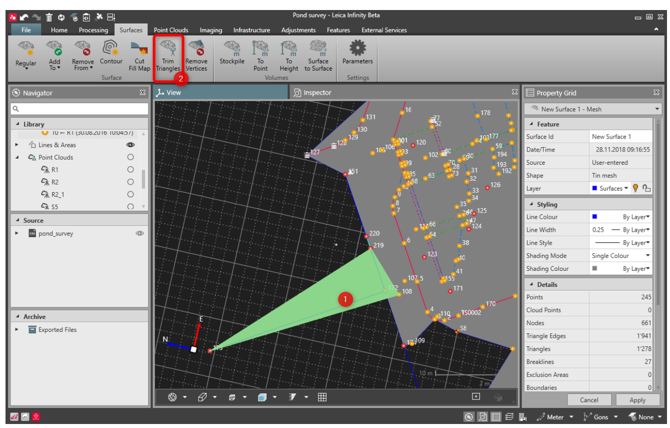
3.4.8 Trim Triangles

Trim Triangles

It is possible to remove the triangles at the outer border of a surface.

☞ Removing the triangles does not create holes.

How to trim triangles:



1. Select at least one triangle at the outer border of a surface.

   ☞ All existing selection shortcuts are available.

2. Select **Trim Triangles**, either from the ribbon bar or from the context menu.

See also:

Graphical View

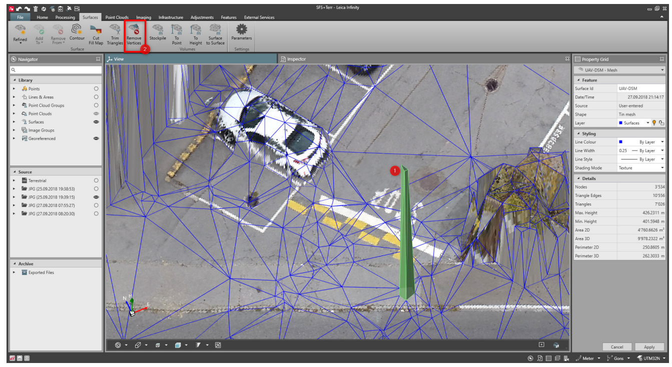
3.4.9 Remove Vertices

Remove Vertices

It is possible to remove vertices from a surface when spikes or areas of triangles have been created but not wanted.

☞ Removing the vertices does not create holes.

How to remove vertices:



1.	Select all triangles that contribute to a vertex.
2.	Remove vertices either from the ribbon bar or from the context menu. ☞ The triangles are locally reorganised.

3.4.10 Fill Holes

Fill Holes

It is possible to fill holes in the surfaces when the holes in the surface are created but the surface should be continuous.

☞ Fill holes fills the closed shape holes.

How to select the surface for the fill holes tool:

1.	Select the surface from the graphical view, the inspector or the navigator.
2.	Select **Fill Holes** from the surface ribbon bar.
☞	The selected surface is automatically used in the fill holes tool.

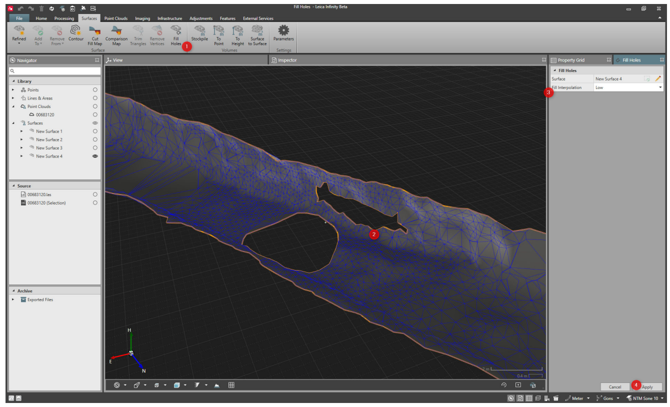
Alternatively:

1.	Select the surface and right-click to open the context menu from the graphical view, the inspector or the navigator.
2.	From the context menu, select **Fill Holes**.
☞	The selected surface is automatically used in the fill holes tool.

Alternatively:

1.	Select **Fill Holes** from the surface ribbon bar.
2.	Select the **pencil** icon and then the surface from the list or select the surface in graphical view and select the **Add current selection** icon.

How to fill the holes:



1.	Select the surface and launch the fill holes tool as explained previously.
2.	Select highlighted holes edge which should be filled.
3.	Select the **Fill Interpolation** parameter.
4.	Select **Apply**.

☞ Multiple holes can be selected in one hole filling event. The hole is filled in the same fill interpolation parameter within the same fill holes event.

Fill Interpolation Parameter

This setting defines how holes in the surface are filled. The level of interpolation considers the surrounding surface geometry to determine how to fill the holes and how well to fit to the surrounding surface. With no interpolation, straight triangles between hole vertices are created, with no interpolated vertices. When applying levels of interpolation results in more vertices within the holes to be filled.

3.5 Point Clouds

3.5.1 Overview

Point Clouds

When a job contains point cloud data, that is imported into Infinity, this special kind of data can be managed in the Point Clouds tab.

If you scanned buildings, industrial plants or other huge objects you probably had to take your point clouds on several stations all around the object and it becomes necessary to group the single point clouds as belonging together, defining one object. Especially, when you scanned more than one object in a job and when you had to scan from several stations around each object, it becomes necessary to group the scan data. The grouped data also gives you a solid basis for future surface calculations.

Creating new point cloud groups

To group point clouds into a point cloud group:

1.	In the Inspector, go to the **Features** tab and open the **Point Clouds** section.

2. Select the **point clouds** to be grouped and select **New** from the **Point Clouds** ribbon bar.

The new point cloud group is created and added to:

- The **Point Cloud Groups** section in the **Library** of the Navigator. If the Point Cloud Groups subsection does not yet exist, it is created.
  For each point cloud group, the single scans belonging to it are listed.
- The **Point Cloud Groups** section in the **Features** tab of the Inspector.

☞ Point cloud groups are always created with a default name. To change the name, click onto a point cloud group in the Navigator or in the Inspector and adapt its name in the Property Grid.

☞ You can also create an empty point cloud group and add single point clouds to it later on.

Adding and removing point clouds from a point cloud group

To add point clouds to a point cloud group:

1. In the Inspector, go to the **Features** tab and open the **Point Clouds** section.
2. Select the **point cloud** to be added to a point cloud group and select **Add to** from the **Point Clouds** ribbon bar.

or:

1. In the Source section of the Navigator, go to the station setup containing the point cloud to be added.
2. Select the **point cloud** and select **Add to** from the **Point Clouds** ribbon bar.

☞ If there is more than one point cloud group available in your project, select one from the drop-down list.

To remove point clouds from a point cloud group:

1. In the Inspector, go to the **Features** tab and open the **Point Clouds** section.
2. Click on the arrow next to Point Cloud Groups to open the category.
3. Click on the arrow next to the point cloud group from which one or more point clouds shall be removed.
4. Select the **point clouds** to be removed and select **Remove from** from the **Point Clouds** ribbon bar.

or:

1. In the Library section of the Navigator, go to **Point Cloud Groups** and expand the tree for the point cloud group from which one or more point clouds shall be removed.

2. Select the **point clouds** to be removed and select **Remove from** from the **Point Clouds** ribbon bar.

Cleaning points

If your point clouds contain random points, like points that have accidentally been taken while scanning a main object, then these random points can be identified by Infinity and cleaned from the point cloud data in your project. As a consequence, you only see the relevant point cloud data in the graphical view.

To clean points:

1. Select a **point cloud** or a **point cloud group** either in the Inspector or in the Navigator.

2. Select **Clean Points** from the ribbon bar to clean accidentally taken points from your point cloud data.

In the Property Grid of the point cloud and/or the point cloud group, you see that the total number of points decreases to just count the relevant points.

Deleting points

To delete point clouds:

Select **Delete** to delete the selected point clouds.

In the graphical view, select the point clouds to be deleted.

Colouring Modes

Select between:

- **RGB** colouring (natural colours).
- **Intensity**.
- **SNR** (Signal-to-Noise Ratio).
- **Single Color**.

For Intensity and SNR you can select a **Color Scale** from the drop down list. By default, MultiHue is set.

For Single Color representations you can select whether the representation shall be **By Layer** (default) or **by a user-selected colour**.

For all Colouring Modes the last used settings are remembered when you open another project. Select the same option to use the same settings again. The icon indicates which settings have last been used. By default, the project data is displayed in RGB mode.

Filtering points

To reduce the point density in the graphical view:

Select **Filter** from the ribbon bar and select a percentage to which the point density shall be reduced from the drop-down menu.

The last used setting are remembered when you open another project. Select the option again to filter the points using the same percentage. The icon indicates which percentage has last been used. By default, the project data is displayed with 100%.

3.5.2 Point Cloud Properties

Point Cloud Properties

In the point cloud properties you can:

- Edit the Point Cloud ID and enter a description.
- Select a **Colouring Mode** from the drop-down list:
  For Intensity and SNR choose a colour palette, for Single Colour choose a colour or decide on the layer colour.
- Select to select a different **Position Source**.
- Select to select a **Panorama image** to be assigned to the point cloud.

☞ You can only assign panoramas that have been taken on the same station as the point cloud.

You are informed about:

- The panorama assigned to the point cloud.
- The station on which the point cloud has been taken.
- The settings (point cloud method and mode, resolution, min. and max. distance).
- The station coordinates.

3.5.3 Reduce Point Clouds

Reduce Point Clouds

General

The reduce point clouds tool allows you to reduce the number of points within selected point clouds. There are three different methods to choose from. They work differently, but result in a point cloud with less than the original number of points.

☞ In order to run this feature, at least one point cloud or point cloud group must be selected.

☞ When applied to multiple point clouds, each point cloud is treated individually. When applied to a point cloud group, the group as a whole is reduced to the specified value.

Short description of each method:

Method	Description
By Percentage **By Count**	The user can decide what percentage or number (count) of points to keep after the reduction. Use one of these methods when the number of remaining points should be under control. The operation automatically tries to remove points in more dense areas first. There is no equal spacing between points after the reduction.
By Distance	An average distance between points is entered. That defines the size of an octree that is projected onto the point cloud. Within each of the cells, the point closest to the centre of gravity is kept. The result is a dense point cloud that is grid-like with an approximate spacing of the average distance setting. ☞ It does not create an exact grid out of the point cloud. ☞ Unlike the other methods, there is no control over the remaining number of points.

3.5.4 Clipping

Clipping

Clipping allows the user to hide the cloud points with defined clipping objects like plane, slice or box. Clipping does not delete the points. Multiple clips are permitted. Use zooming, rotating and panning when creating a clip for better data visualisation in 2D or 3D.

Once a clip has been created, you can remove the clipping and restore the visibility of all point cloud data by selecting **Reset Clips** from the ribbon bar.

Plane

Plane hides points beyond a single plane.

A single plane can be created along a specified axis or defined by three points.

To hide points by plane:

1.	Select **Clip** from the ribbon bar and then **Plane** from the drop-down menu. An additional window opens called Clip.
2.	Select the **Single point**, **Two points** or **Three points** method to create the plane. For the Single point method define the Easting, Northing or Height axis for the plane orientation.
3.	To create the plane with the Single point method, a temporary plane representing the clip appears in the graphical view and follow the cursor. Click once to position the plane. Points on one side of the plane are hidden from the view. Use **Invert view** to change the points visibility between plane sides.

4.	To create the plane with the Two points or Three points method, select two or three points in the graphical view to define and orientate the plane. Points on one side of the plane are hidden from view. Use **Invert view** to change the points visibility between plane sides.
5.	You can move the plane along the current axis by selecting the red indicator and holding the left mouse button.
6.	Select **Create** to save the current clip.

Slice

Slice hides points outside of two parallel planes.

Two parallel planes can be created along a specified axis or defined by three points plane and slice width.

To hide points outside of a slice:

1.	Select **Clip** from the ribbon bar and then **Slice** from the drop-down menu. An additional window opens called Clip.
2.	Select the **Single point**, **Two points** or **Three points** method to create the Slice. For the Single point method define the Easting, Northing or Height axis for the slice orientation.
3.	To create the slice with the Single point method, a temporary plane representing the clip appears in the graphical view and follow the cursor. Click once to position the first plane. Now the second temporary plane follows the cursor. Click again to position the second plane. Points outside of the slice are hidden from view.
4.	To create the slice with the Two points or Three points method, select two or three points in the graphical view to define the first plane. Now the second temporary plane follows the cursor. Click again to position the second plane. Points outside of the slice are hidden from view.
5.	You can resize the slice by selecting the red indicator and holding the left mouse button or change the slice width in the Clip window. To move the slice, hold down the Ctrl-key and left-click the mouse at the same time.
6.	Select **Create** to save the current clip.

Box

Box hides points outside a 3D box.

To hide points outside of a box:

1.	Select **Clip** from the ribbon bar and then **Box** from the drop-down menu. An additional window opens called Clip.

2. Select a method:
Fixed on ground
The first box face is defined by three points at the height of the first selected point. The height of the box is defined by a fourth point.
Free
The first box face is defined by three points. The height of the box is defined by a fourth point.
Easting
The first box face is defined three points around the Easting axis. The height of the box is defined by a fourth point.
Northing
The first box face is defined by three points around the Northing axis. The height of the box is defined by a fourth point.
Height
The first box face is defined by three points around the height axis. The width of the box is defined by a fourth point.

3. Select a point in the graphical view to represent the first corner of the desired 3D limit box.
A temporary line is attached to the cursor.

4. Select a second point representing the opposite corner of the desired limit box.
The resulting rectangle is projected onto the work plane.

5. Select a third point to determine the last box dimension.
Points outside the boundaries of the box are hidden from the view.

6. You can resize the box by selecting the red indicators and holding the left mouse button or by changing dimensions in the Clip window.
To move the box, hold down the Ctrl-key and left-click the mouse at the same time.
You can rotate the box by selecting the purple indicators and left-click the mouse.

7. Select **Create** to save the current clip.

☞ If all points are hidden after applying the box clip, it is likely that the limit box is defined in an incorrect position. In this case, use **Cancel** and try again.

3.6 Point Cloud Registration

3.6.1 Overview

Point Cloud Registration

The point cloud registration module allows for the import of BLK360, RTC360 and RTC360 LT data into Infinity. Individual scan setups can then be registered together, optimised and stored to create a Unified Point Cloud (UPC). Scanning objects in Infinity are accessible from the navigator, inspector, graphical view, property grid and the register tool.

Infinity currently supports registration through visual alignment and by the matching of targets. Targets can be matched to Infinity points (for example, TPS or GNSS observations) to translate the UPC onto a coordinate system.

If necessary, the UPC can be exported from Infinity using various formats for work in third-party applications.

Requirements:

The following modules are required to utilise point cloud registration functionality in Infinity:

- Point Cloud Registration.

Module Features

- Up to 100 setups per Infinity project.
- Import BLK360 data.
- Import RTC360 data.
- Visual alignment.
- Match targets.
- Apply control.
- Optimise.
- Store and create UPC.

☞ The BLK data manager application is required when importing BLK360 data.

☞ To ensure the best performance, it is recommended that you install the latest graphics card drivers from the website of the manufacturer.

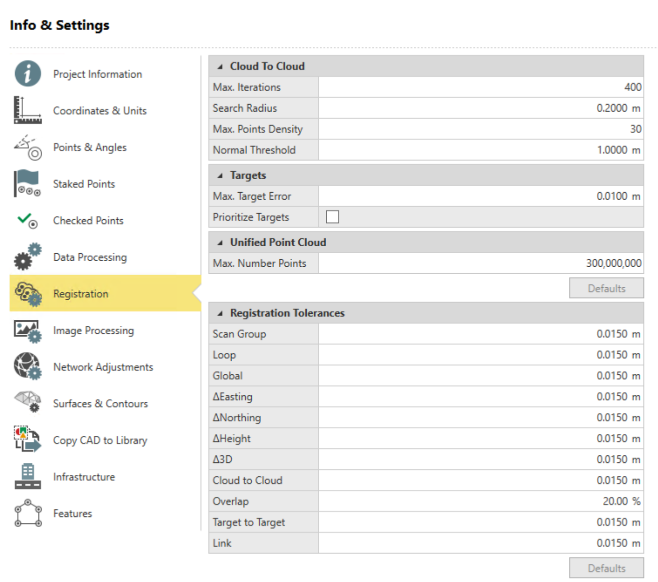
3.6.2 Point Cloud Registration Settings

Point Cloud Registration Settings

The point cloud registration settings are configured through the Infinity **File** > **Info & Settings** > **Registration**, the import window and in the register tool itself.

Registration Settings

The following settings can be configured:



Cloud-to-Cloud

- **Max. Iterations**: The maximum number of times the algorithm tries to fit the point clouds together.
- **Search Radius**: The radius in which the application searches for common matched surfaces.
- **Max. Point Density**: The number defines how many points in the cloud are used for cloud-to-cloud registration.
- **Normal Threshold**: This value controls the threshold for how valid the surfaces are in the cloud-to-cloud registration.

Targets

- **Max. Target Error**: Maximum allowed error between targets to be included in the registration.
- **Prioritise Targets**: The targets receive a higher weight than the cloud-to-cloud constraint during the link creation process.

Unified Point Cloud

- **Max. Point Number**: The maximum number of points which are created for the unified point cloud.

Registration tolerances

- **Scan Group**: Overall Scan Group error for all links in the Scan Group.
- **Loop**: The loop error shows how much the links have to move to be globally optimised. After the scan group optimisation, the loop error is 0.000 as it is contained within the globally optimised error.
- **Global**: The value that the setups have to be moved when included in a loop. A high global error value may indicate a problem with a link or scan group.
- **ΔEasting**: The difference between the Easting coordinate of the control point and the assigned target.
- **ΔNorthing**: The difference between the Northing coordinate of the control point and the assigned target.
- **ΔHeight**: The difference between the height coordinate of the control point and the assigned target.
- **Δ3D**: The difference between 3D coordinates of the control point and the assigned target.
- **Cloud-to-Cloud**: The error in the cloud-to-cloud constraint between the setups.
- **Overlap**: The percentage overlap between the two setups forming the link.
- **Target to Target**: The average distance or error between matching targets in a link or scan group.
- **Link**: The overall error for the link which can be composed of cloud-to-cloud error, target to target error and overlap.

Import Window

The following settings can be configured in the import window when BLK360 (.blk) files or RTC360 (project.rtc360) files are selected:

- **Auto Cloud**: The auto cloud routine is performed on the selected setups at import. This process attempts to align the setups and join them together using cloud-to-cloud.
- **Auto Black & White Target**: Black and white targets are extracted at import. The match targets routine then attempts to join setups with matching targets.

Register Tool

The following settings can be configured in the links subtab of the register tool tab:

- **Lock Links**: The lock check box locks the link so that the link does not move during global optimisation. You may choose to lock a link when the global error of the link is too high. Locking the link prevents the link from being affected (moved) by global optimisation.

Property Grid

The following setting can be configured in the scan group property grid (error measurements flyout):

- Lock Links.

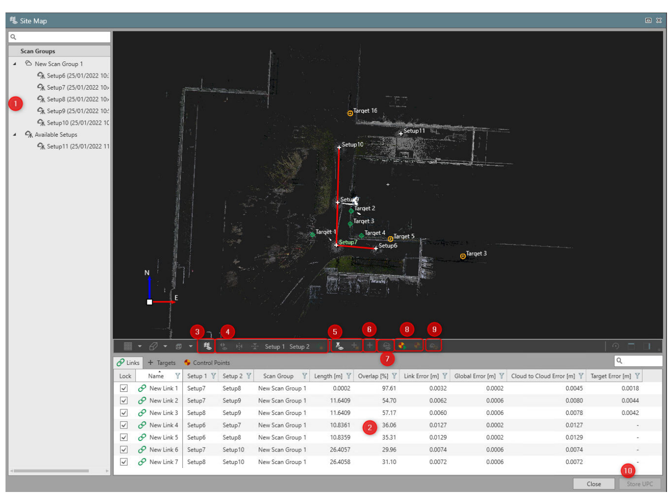
The following setting can be configured in the targets property grid:

- **Target ID**: The Target ID can be renamed or matched to an existing target.
- **Assigned Point**: The target can be assigned to an existing Infinity point (for example, TPS or GNSS point).

3.6.3 Register Tool Workflow

Register Tool Workflow

The register tool has several different elements to facilitate the registration workflow.



No.	Name	Description
1.	**Registration Tool Navigator**	View all the imported scan setups organised in scan groups or available setups.
2.	**Register Tool Bottom Menu**	View links, targets and control points.
3.	**Site Map View**	View and organise scan data in 2D top down view.
4.	**Visual Alignment**	Display two selected setups and create link between them.
5.	**Scan Setup View**	View scan setup from scanner position to view the data and create targets.
6.	**Match Targets**	Create link based on targets.
7.	**Scan Group View**	View scan group in 3D view.
8.	**Georeference the Scan Group**	Auto assign control and apply control.
9.	**Optimise**	Optimise the scan group.
10.	**Store UPC**	Store Unified Point Cloud (UPC) in the Infinity project.

Register Tool Navigator

This window allows you to select the setups and align them to build the scan group.

The data can be organised in:

- **Scan Groups**: The scan groups show all scan groups in the current Infinity project.
- **Available Setups**: The available setups shows all setups that are not part of a scan group.

Register Tool Bottom Menu

This window displays all the information about the scan group:

- **Links Subtab**: Shows all links in the current Infinity project.
- **Targets Subtab**: Shows all targets in the current register view.
- **Controls Subtab**: Shows all controls in the current register view.

Site Map View

The Site Map view is a 2D-only view to select and move setups to organise and make it easier to register.

The following controls are available:

- **Shift Setup or Scan Group**: Left-click mouse on a setup to shift a setup or scan group.
- **Rotate Setup or Scan Group**: Left-click mouse + Shift key on a setup to rotate a setup or scan group.
- **Multi-Object Selection**: Left-click mouse + Ctrl key.
- **Fence Selection**: Left-click mouse + Shift key on the open space.
- **Context Menu**: Right-click mouse on an object for the context menu options.

Visual Alignment View

The visual alignment view displays two selected setups to perform a cloud-to-cloud registration.

The following controls are available in top view:

- **Shift Setup**: Left-click mouse to shift the active setup.
- **Rotate Setup**: Left-click mouse + Shift key to rotate the active setup.

The following controls are available in front/back/left/right views:

- **Shift Setup Along Z-axis**: Left-click mouse on a setup to shift the active setup along the Z-axis.
- **Tilt Setup About View Axis**: Left-click mouse + Shift key to tilt the active setup about the view axis.

Setup View

The setup view displays the setup cloud of the selected setup.

The setup view can be used to:

- View and inspect the setup cloud using existing Infinity view controls.
- Mark user targets.

Match Targets

Match targets creates the link between two or more scan setups based on a minimum of three matching targets.

Where matching target labels exist, the match targets algorithm attempts to align the setups based on the labels.

Where no matching labels exist, the match targets routine attempts to align the setups by target geometry.

Scan Group View

The scan group view displays the scan group cloud of the selected scan group. The scan group view can be used to interrogate the scan group cloud using existing Infinity view controls.

Georeferencing the Scan Group

Use the controls assigned to targets to bring the scan group to project coordinate system.

- **Auto Assign Control Points**: Match target and control point with the same point ID if not manually matched.
- **Apply Control Points**: Assign the control points to the scan group to transform it to the project coordinate system.

Optimise

Final optimisation of the scan group, updating all links and scan setup positions before storing the UPC.

Store Unified Point Cloud

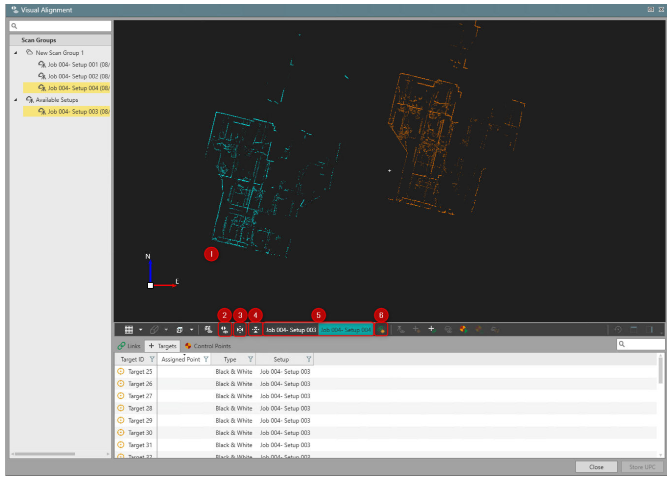
Store the UPC to be used in the Infinity project.

Register Strip

The register strip is at the bottom of the register tool view and provides access to the functions provided by the registration module.

3.6.4 Registration Using Visual Alignment

Registration Using Visual Alignment



No.	Name	Description
1.	**Visual Alignment View**	Display two scan setup point clouds which are used in the visual alignment.
2.	**Visual Alignment**	Open the visual alignment mode.
3.	**Top View**	Switch the visual alignment graphical view to top view.
4.	**Side View**	Switch the visual alignment graphical view to side view.
5.	**Setup Selector**	Switch between two setups opened in visual alignment. Active one can be moved.
6.	**Join**	Create the link between two setups opened in visual alignment.

The visual alignment view displays two selected setups in different colours to align and join.

The following controls are available in the top view:

- **Shift Setup**: Left-click mouse to shift the active setup.
- **Rotate Setup**: Left-click mouse + Shift key to rotate the active setup.

The following controls are available in the front/back/left/right views:

- **Shift Setup Along Z-axis**: Left-click mouse on a setup to shift the active setup along the Z-axis.
- **Tilt Setup About View Axis**: Left-click mouse + Shift key to tilt the active setup about the view axis.

The active setup can be toggled by selecting the setup name from the register strip.

Once the setups are visually correct they can be joined:

- **Join**: Uses the overlapping points between setups to join them together through cloud-to-cloud. A global optimisation is then performed on the scan group. If it detects setups that can also be joined, you are notified and asked to or not to proceed with this additional step.

☞ In the case when between setups, there is not enough overlap to create a valid link and optimise it, the user-defined link is created.

☞ The user-defined link has a lock state as default.

3.6.5 Registration Using Targets

Registration Using Targets

This article covers the process of registration using targets in Infinity.

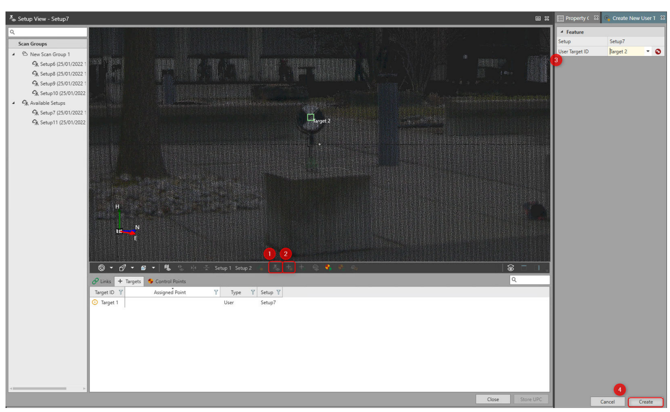
Target Creation

Targets can be created in Infinity in a number of ways:

- **Black & White Target Extraction**: Ensure that the Auto Black & White Targets checkbox is ticked in the import window. This option also attempts to align and join scans with matching targets as part of the process.
- **User Target Creation**: When in the setup view, select **Create User Targets** from the register strip. You can then mark user targets on the setup cloud by a left-click of the mouse.

☞ Any part of the setup cloud can be used to mark a user target, facilitating the ability to join setup clouds using objects that are not necessarily the targets.

User target creation:



1.	Open the scan **Setup View**.
2.	Select **Create User Target**.
3.	Mark the **User Target**. ☞ The tool follows the same logic as create new point. It is possible to relabel the target during target creation. The next created target takes the name after the previous one and follows the numbering.
4.	After all the targets are created, select **Create**.

Target Labelling

Targets can be labelled in Infinity in a number of ways:

- By entering a defined text string in the Target ID field of the targets Property Grid.
- By selecting a matching target from another setup using the pencil icon in the Target ID field of the targets Property Grid.

☞ See Applying Control for further information on applying control to targets.

Matching Targets

To join setups with a minimum of three matching targets:

- Select two or more setups from the register tool navigator or register tool.
- Select **Match Targets** from the register strip.

Where matching target labels exist, the match targets algorithm attempts to align the setups in the following order:

- By label.
- By Target geometry.

Where no matching labels exist, the match targets routine attempts to align the setups by target geometry.

☞ Once the setups have been aligned using targets they are joined using cloud-to-cloud.

Targets State:

1. ⊙ - Not used: Target is not part of the link. Target does not constrain with any targets on other setups.
2. ⊕ - Used: Target is part of the link. Target constrains with targets on other setups.

3.6.6 Applying Control

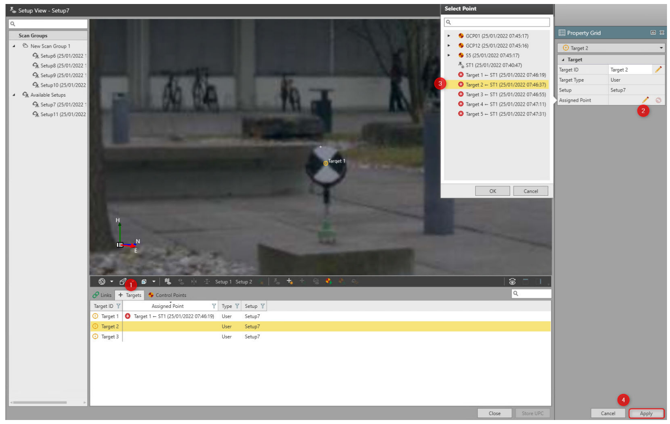
Applying Control

Black & white targets extracted at import and user targets created after import, can be assigned to Infinity points for translating the scan group on to a coordinate system.

Assign Control to Target

Infinity points can be assigned to targets by:

- Selecting an Infinity point using the pencil icon in the Assigned To field of the targets Property Grid.
- When the Target ID Matches the Infinity Point ID: Selecting **Assign Control** from the register strip.



1.	Select the target from the target subtab or the graphical view.
2.	Select the pencil icon in the Property Grid to open the flyout with Control Points from an Infinity Project.
3.	Select from the flyout the control point which should be assigned to the target.
4.	Select **Apply**.

Apply Control to Scan Group

With a minimum of three targets with assigned Infinity points in a scan group, select **Apply Control** from the register strip. The transformation is computed and relevant error information for each assigned point is reported in the controls subtab of the register tool.

☞ GNSS points must have a local position to be assignable.

☞ No scale factor is applied in the apply control process.

3.6.7 Optimise Scan Group, Store and Create Unified Point Cloud

Optimise Scan Group, Store and Create Unified Point Cloud

Once the required links between setups have been created the scan group can be optimised. Following this, the scan group can be stored and a Unified Point Cloud (UPC) can be created for working with in the Infinity graphical view.

Optimise Scan Group

To perform a global optimisation of a scan group:

- Select the scan group to be optimised from the library, inspector or the register tool side tab.
- Select **Optimise** from the register strip.

Store Scan Group and Create Unified Point Cloud

To store the optimised scan group and create the UPC:

- Select the scan group to be optimised from the library, inspector or the register tool navigator.
- Select **Store UPC** from the bottom right corner of the register tool.

The scan group state changes to stored and the UPC is visible in the graphical view.

☞ The number of points contained within the UPC is controlled by the file tab setting Max. Number Points.

☞ All setups in the Infinity project must be part of the scan group for the Store UPC option to be active.

☞ The scan group must be optimised for the Store UPC option to be active.

☞ Unstoring a scan group removes the UPC from the Infinity project.

3.6.8 Reports & Exports

Reports & Exports

The data source report and the scan registration report are available for BLK360 and RTC360 data. The stored Unified Point Cloud (UPC) can be exported to the existing point cloud formats supported by Infinity.

Data Source Report

To create the data source report for BLK360 or RTC360 data:

- Select the required source from the source pane in the navigator.
- Select the file/reports/data source report.

Scan Registration Report

To create the scan registration report for a stored scan group:

- Select the stored scan group from the library, the inspector or the register tool.
- Select the file/reports/scan registration report.

UPC Export

To export the UPC to a supported point cloud format:

- Select the UPC from the library or the inspector.
- Select **File**, then **Export**.
- Select **Point Clouds** from the Export window.
- Select the desired point cloud format from the Export side menu.
- Select **Export**.

3.6.9 Importing BLK360 and RTC360 Data

Importing BLK360 and RTC360 Data

Import Settings

Scanning related import settings shown in the import window include:

- **Auto Cloud**: The auto cloud routine is performed on the selected setups at import. This process attempts to align the setups and join them together using cloud-to-cloud.
- **Auto Black & White Target**: Black and white targets are extracted at import. The match targets routine then attempts to join setups with matching targets by geometry.

☞ Previously applied settings are used and also applied when importing data using the following drag and drop methods.

Importing BLK360 data

Infinity is able to import BLK360 data from the .blk files created using the BLK data manager application.

To import BLK360 data:
- Download and install the BLK data manager.
- Create the necessary .blk files for each setup in the BLK data manager.
- Import the necessary .blk files through:
  - The import window (selecting Home, then Import and then the scan data).
  - Drag and drop the necessary .blk files to the source pane.

Importing RTC360 data

Infinity is able to import RTC360 data using several methods:
- The Import window (selecting Home, then Import and then the scan data) using the project.rtc360 file.
- Drag and drop the necessary folder to the source pane.

☞ Field360 links and VIS links are imported as part of this process.

3.7 Imaging

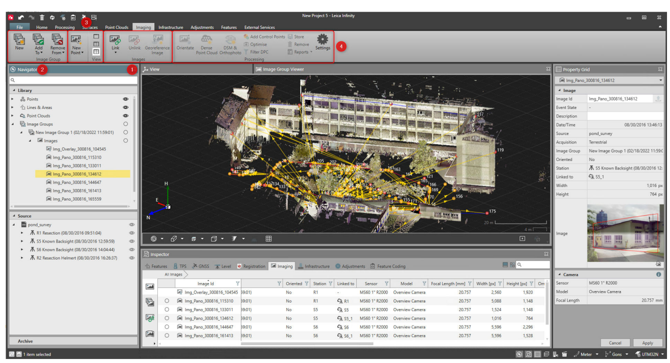
3.7.1 Overview

Imaging

Images are a useful way to add context to your project data. Infinity supports working with images in a project for documentation, along with computing points from images where measured data may have been overlooked during field collection. In Reality Capture, images can be used for generating dense point clouds, providing a detailed 3D perspective to a project area and allowing you to extract or create 3D features and data deliverables.

Infinity supports the import of images from SmartWorx and Captivate field jobs, Aibot UAV, other fixed and rotary wing drones and general JPG images. It is also possible to import georeferenced images to display as background maps.

In imaging you get access to:



No.	Group Name	Description
1.	Images	Base licence needed: • Import, export and view images. • Link or unlink images, to or from points, lines or areas. • View computed image points. • Georeference images.

No.	Group Name	Description
2.	Image Group	Imaging option needed: • Create image groups that allow you to work with related images easier. • Compute points from the grouped images.
3.	Image Point Computation	Imaging option needed: • Image Group Viewer - View images and points computed from images (image points), or compute points from a selected image group. • New Image Point - Calculate new points from images.
4.	Image Processing	Point clouds from images option needed: • Dense point clouds. • Digital surface models. • Orthophotos.

See also:

The video **"Leica Infinity - Imaging Module - How to work with GS18 I data"** https://www.youtube.com/watch?v=1TwomI2lxw0

The video **"Leica Infinity - Imaging Module - Points from Images"** https://www.youtube.com/watch?v=PDR4ROLrKbA

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

The video **"Leica Infinity - UAV Processing Part 1 - Import and orientate Aibot data"** https://www.youtube.com/watch?v=F8OdA9sy6q4

The video **"Leica Infinity - UAV Processing Part 2 - Add GCPs and update orientation"** https://www.youtube.com/watch?v=Y8hLA0RObFA

The video **"Leica Infinity - UAV Processing Part 3 - Create Dense Point Cloud, DSM and Orthophoto"** https://www.youtube.com/watch?v=XPpLS1cxZlk

3.7.2 Images

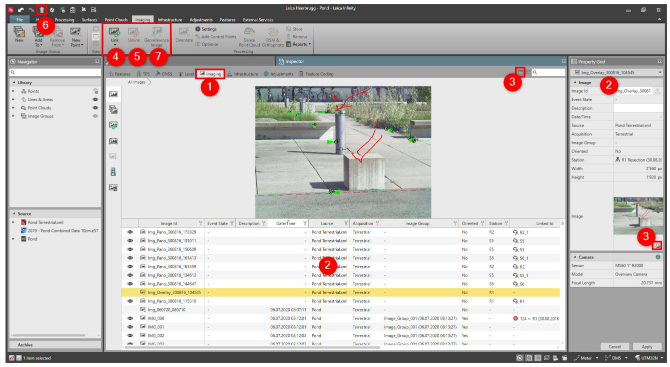
3.7.2.1 Overview

Images

Within images, you get access to the basic functionality of imaging.

Requirements:
- Basic option.

Here you can do the following:



No.	Name	Description
1.	Import Images	Import image data.
2.	Imaging Inspector	Explore imaging-specific project content grouped thematically for detailed investigation.
3.	Image Properties	View the properties of the selected image.
4.	Image Viewer	View the selected image in the image viewer.
5.	Link Images	Link selected image to point/line/area.
6.	Unlink Images	Unlink selected image from point/line/area.
7.	Delete Images	Delete selected images.
8.	Georeference Images	Georeference selected images.

See also:

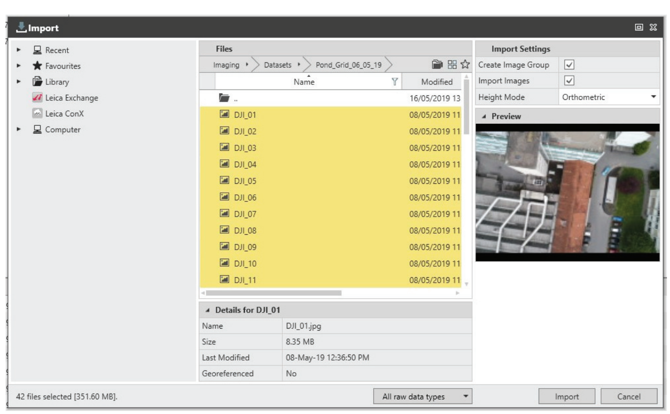
The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

3.7.2.2 Import Images

Import Images

Images can be imported through the Import dialog or by drag and drop option. Currently, JPG, PNG, TIFF and PDF file types are supported for images. JPG, PNG and TIFF file types are supported for georeferenced images.

Import dialog:



1.	Check or uncheck the desired **Import Settings**.
2.	Select **Import** to start the import.
☞	For images imported by drag and drop, the default Import Settings are applied.

Import Settings

When the **Create Image Group** box is checked, an image group is created directly after closing the import dialog. In order to be processed, images must be added to an image group. If no image group is created on import, one can be created after the import and images can be added to it. For further information, see Image Group.

☞ At least two images must be selected for this option to become available.

☞ By default, this setting is checked. However, if unchecked, Infinity remembers the selection at the following import.

When the **Import Images** box is checked the images are copied to the project. If this option is unchecked, the images are not copied into the project but linked to it through the path specified in the import dialog. If, after importing, the images are deleted or moved to a different path, no processing is allowed. A warning message is displayed if the images cannot be found at the expected location.

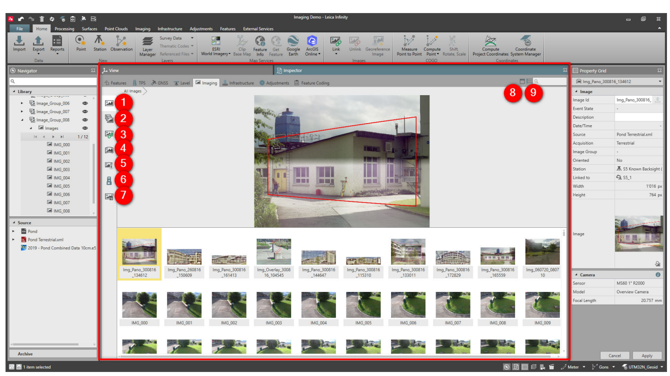
☞ When moving the project to another computer for further processing, the link is no longer valid and no further processing is possible.

☞ By default, this setting is checked. However, if unchecked, Infinity remembers the selection at the following import.

3.7.2.3 Imaging Inspector

Imaging Inspector

From the inspector imaging tab, there are several groups to help you work with images.



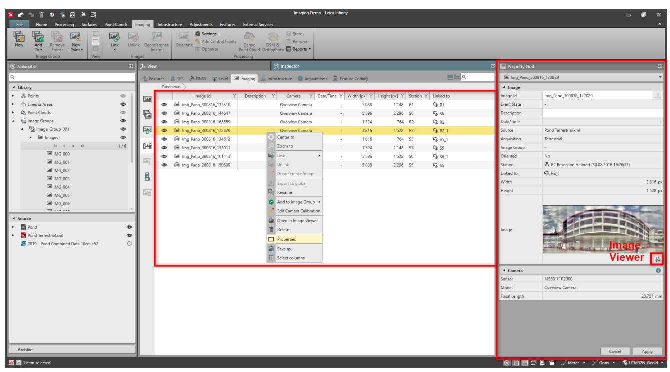
No.	Name	Description
1.	**All images**	View all images in a project.
2.	**Image Groups**	View images by their grouping.
3.	**Linked/Not Linked**	Sort images by point, line, area feature which images are linked to.
4.	**Panoramas**	Using TPS imaging sensors, all imported field acquired panorama images are sorted here.

No.	Name	Description
5.	**Georeferenced**	List of all the images that have been rectified, either by import, using clip base map or from the DSM & Orthophoto processing step.
6.	**TPS Setups**	View all images that are linked to a total station setup.
7.	**Imaging Results**	Access imaging results. The view is split in two panes, the results pane and the task pane.
8.	Image Viewer	You can view the selected image in a resizable image viewer that is part of the inspector or in separate window.
9.	**Show Thumbnails**	Switch between a detailed list or a thumbnail strip of images.

3.7.2.4 Image Properties

Image Properties

Image properties can be seen in the property grid as well as in the imaging inspector.



Name	Description
Image ID	The name of the image.
Event State	This is the GNSS solution type (for example phase fixed RTK or navigated RTK) associated to Leica and DJI UAV images. It indicates the quality of the image position.
Description	Any optionally entered description.
Date/Time	The date and time of when the image was taken.
Source	The name of the image source data.
Acquisition	The acquisition type informs whether the images are from an aerial or from a ground perspective. Could be: • Aerial • Terrestrial • Unknown
Image Group	The ID of the image group which the image belongs to.

Name	Description
Oriented	Is indicating if the image has its orientation computed with photogrammetric techniques.
Station	The ID of the station on which the image was taken. ☞ Station is shown only for TPS images.
Linked to	The ID of the object which the image is linked to. ☞ Linked to is shown only when the image has a link to an object.
Width	Image width in pixels.
Height	Image height in pixels.
Image	The image itself is displayed as a thumbnail. ☞ The image can be opened in a separate image viewer using the icon.
Camera	The details of the camera that the image was taken with.

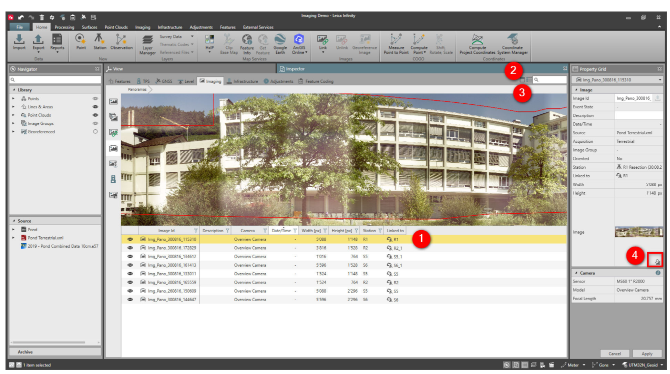
☞ Images are also displayed in the property grid of the objects they are linked to. If you select an object to which one or more images are linked, then the thumbnails of these images are displayed in the properties.

3.7.2.5 Image Viewer

Image Viewer

You can view the selected image in a resizable image viewer that is part of the inspector or in separate window.

To open/close the image viewer:



1. Select an image in the content view.
2. Select **Open Image Viewer** to see the image in a window.
   ☞ The image fits to the window size.
3. To close the image viewer, select **Close Image Viewer**.
4. Alternatively you can open the image viewer in a separate window by selecting the icon in the Property Grid.

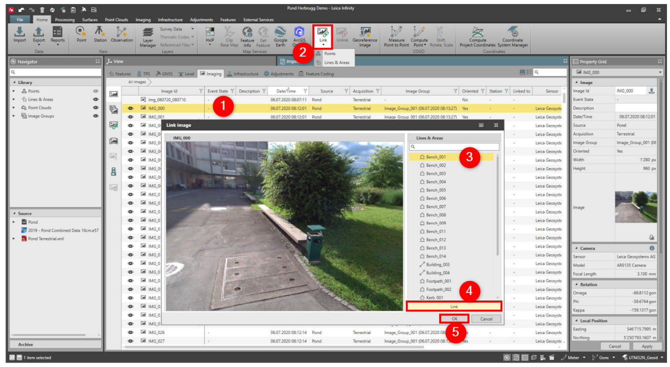
☞ You can shift the separator between the content view and the image viewer to change the size of the display.

3.7.2.6 Link Images

Link Images

Images can manually be linked to points or lines or areas. They cannot be manually linked to point clouds.

To link images to an object:



1. Select the image you want to link, either from the Inspector, Navigator or the graphical view.
2. Select **Link** from the Images ribbon bar.
   Alternatively select **Link** from the context menu.
3. In the Link Image window, select the object to which you want to link the images.
   ☞ You can also search for an object.
4. Select **Link** to link the image.
5. Select **OK**.

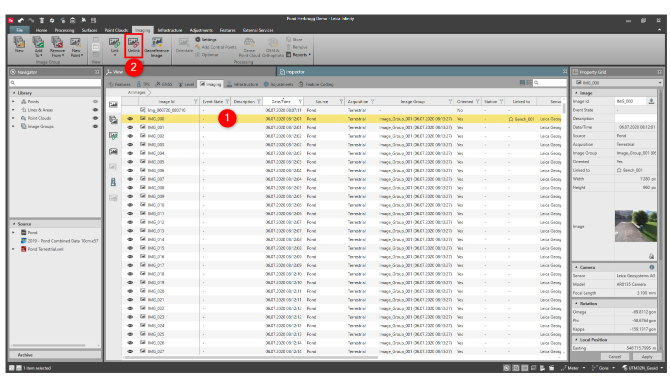
☞ An image can be linked to more than one object and an object can have more than one image linked to it.

3.7.2.7 Unlink Images

Unlink Images

Images can manually be unlinked from points or lines or areas. They cannot manually be unlinked from point clouds.

To unlink an image from an object:



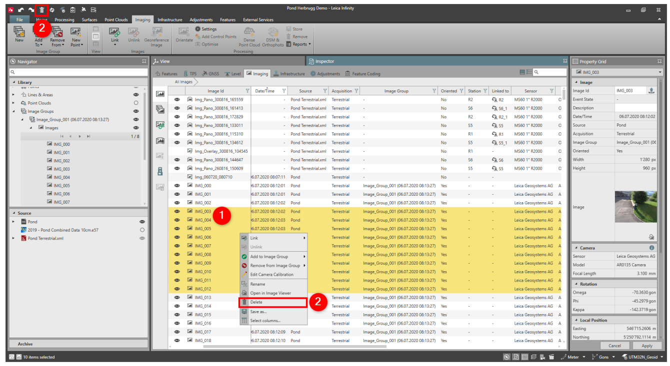
1. Select the linked image, either from the Inspector, Navigator or the graphical view.

2. Select **Unlink** from the Images ribbon bar.
Alternatively select **Unlink** from the context menu.

3.7.2.8 Delete Images

Delete Images

To delete images:



1. Select the images you want to delete, either from the Inspector, the Navigator or the graphical view.
2. Right-click into the selection and select **Delete** from the context menu or select the **Delete** option in the top left corner of the main window.

3.7.2.9 Georeference Images

3.7.2.9.1 Overview

Georeference Images

Infinity supports the display and the creation of georeferenced images. Using these images is a nice way to help visualise, reference and relate your project data.

Georeferenced images are shown as objects in the library and can be set to visible or not visible. It is possible to use many georeferenced images. An existing georeferenced image can be edited and updated with a new transformation.

Georeferenced images are global objects. You can find them under **File** > **Tools** > **Georeferenced Images** ().

Here you can do the following:

No.	Name	Description
1.	Import	Import georeference images.
2.	Export	Export georeference images.
3.	Georeference Tool	Select points and match points to georeference images.

See also:

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

3.7.2.9.2 Import

Import

Images can be imported from PNG, JPG and TIFF format.

Georeferenced images must have a world file, which is a PNW for PNG images, JPW for JPG images, TFW for TIFF images.

Otherwise it is possible to reference these images using the georeference image wizard inside the project or from **File > Tools > Georeferenced Images** ( ).

1.	Locate the images to import.
2.	Define the Import Settings unit for the files.
3.	Select **Import**.

See also:

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

3.7.2.9.3 Export

Export

Georeferenced images can be exported in PNG, JPG, TIFF and geoTIFF format.

☞ Georeferenced images from a project can be exported to be used in other projects.

From the properties, select the **Export to Global** option.

1.	Select the image to export.
2.	Select **Georeference Image**.
3.	Edit the suggested file name.
4.	Define the world file units.
5.	Navigate to the desired export location.
6.	Select **Export**.

☞ All georeferenced images can be viewed in 3D. Enable the 3D terrain from the graphical view tool bar. The default DEM uses a resolution of approximately one arcsecond.

See also:

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

3.7.2.9.4 Georeference Tool

Georeference Tool

Georeference Image: Select Points

To open the georeference image wizard, select an image and:

- Select from the georeference image context menu.
- From the ribbon bar, select **Georeference Image**.

To georeference an image, ground control points are required. These points must be in an Infinity project.

☞ Only points with local grid coordinate values are shown as available.

The first step of georeferencing an image is to select the ground control points to be used to mark the image.

1.	Select the project and then the image control points.
2.	Select the points to move to the Ground Control Points list by selecting ➡ or ➡ to select all points.
3.	Select **Next** to continue to Match Points.

Georeference Image: Match Points

Next step is to match points from the list to the image.

There are two ways to define the picture pixel and the ground control point:

Method 1

Selecting the points manually from the list:

1.	Select the point from the Points list.
2.	With the mouse, navigate to the location in the image and left-click to define this point position.
3.	Select a second point from the list.
4.	With the mouse, navigate to the location in the image and left-click to define this point position.
5.	Repeat until you have computed the residuals and you accept the results.

Method 2

Place the mouse over the pixel of the image and right-click to select from the list the point to mark at this position.

1.	With the mouse, navigate to the location in the image to where the point should be marked.
2.	Right-click with the mouse and from the context menu select the point that defines the pixel.
3.	Repeat until you have computed the residuals and you accept the results.

Select **Finish** to complete the georeference wizard.

The image is stored in the Inspector and in the Navigator.

See also:

The video **"Leica Infinity - Home Module - Georeferencing Images"** https://www.youtube.com/watch?v=m-U29-ipJgs

3.7.3 Image Group

3.7.3.1 Overview

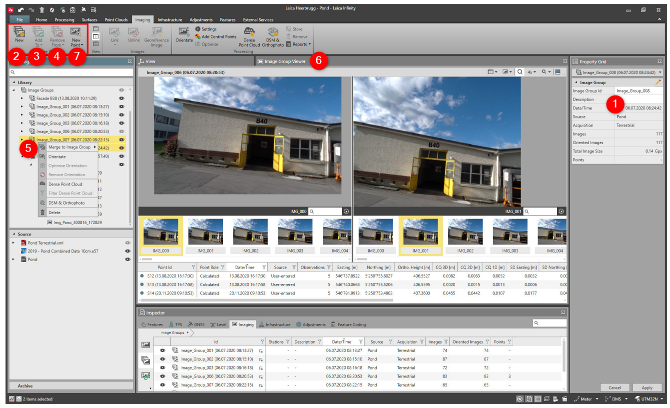
Image Group

To generate data from images, you work with an image group. Image groups are project objects that can be created in Infinity and imported from Captivate.

Requirements:

- Imaging option.

Here you can do the following:



No.	Name	Description
1.	Image Group Properties	Check the properties of an image group.
2.	New Image Group	• New Image Group - Create a new image group. • Add to Image Group - Add images to an image group. • Remove from Image Group - Remove images from an image group. • Merge to Image Group - Merge image groups.

☞ It is important to know, that data computed from image groups are bounded to that image group. If the image group is deleted, any data generated from the image group is also deleted.

See also:

Optional Modules

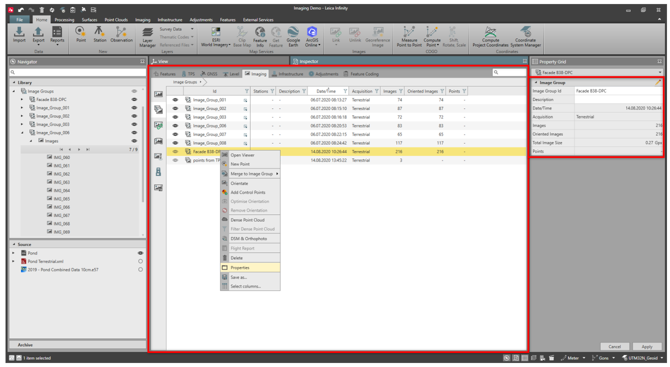
The video **"Leica Infinity - Imaging Module - How to work with GS18 I data"** https://www.youtube.com/watch?v=1TwomI2lxw0

The video **"Leica Infinity - Imaging Module - Points from Images"** https://www.youtube.com/watch?v=PDR4ROLrKbA

3.7.3.2 Image Group Properties

Image Group Properties

Image group properties can be seen in the property grid, as well as in the imaging inspector.



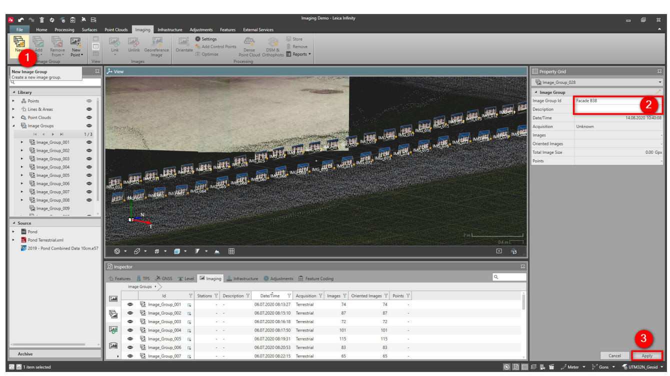
Name	Description
Image Group ID	The name of the image group.
Description	Any optionally entered description.
Date/Time	The date and time of when the image group was created.
Source	The name of the image source data.
Acquisition	The acquisition type informs whether the image groups are from an aerial or from a ground perspective. Could be: • Aerial. • Terrestrial. • Unknown.
Images	Number of images in the image group.
Oriented Images	Number of oriented images in the image group.
Total Image Size	Overall size of all images contained in the image group.
Points	Number of points computed from images.

3.7.3.3 Managing Image Groups

3.7.3.3.1 New Image Group

New Image Group

To create a new image group:



1.	Select **New** from the Imaging tab. Alternatively select images from the Inspector, Library or the graphical view to be grouped and select **New** from the Imaging tab.
2.	Define the Image Group ID and optionally the Description.
3.	Select **Apply**.

The new image group is created and added to:

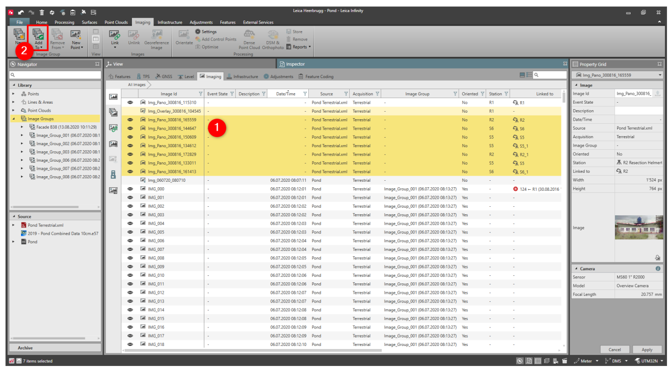
- The image groups section in the library of the navigator. If the image groups subsection does not yet exist, it is created.
- The image groups section in the imaging tab of the inspector.

3.7.3.3.2 Add to Image Group

Add to Image Group

Add to Image Group

To add images to an image group:



1.	Select the Images to be added either from the Library or the Inspector or the graphical view.
2.	Select **Add** from the Imaging tab or from the context menu.

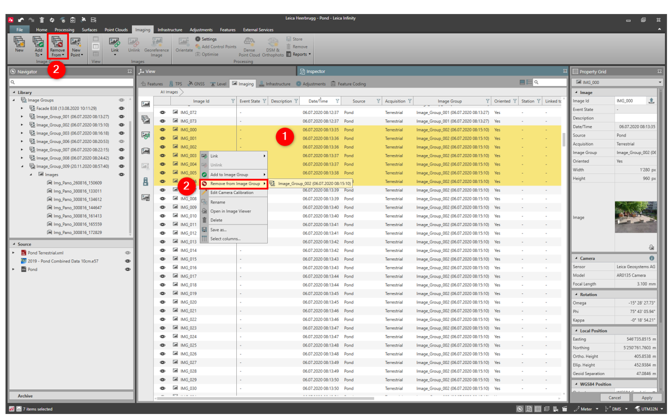
☞ Images are added to a new image group only if they do not belong to any other image group. Only images acquired from TPS devices can belong to multiple image groups.

3.7.3.3.3 Remove from Image Group

Remove from Image Group

Remove from Image Group

To remove images from an image group:



1.	Select the images to be removed either from the Library or the Inspector or the graphical view.

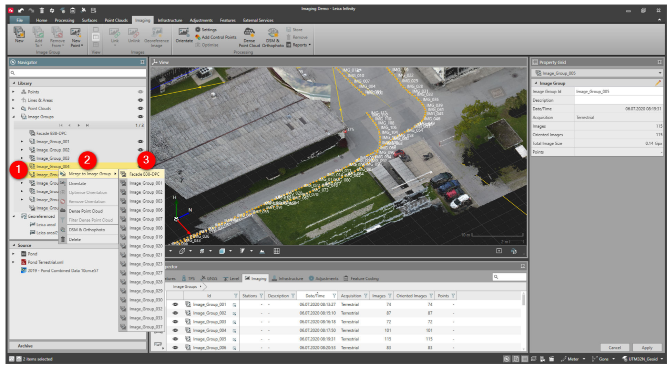
2. Select **Remove** from the Imaging tab or from the context menu.

3.7.3.3.4 Merge to Image Group

Merge to Image Group

It is possible to merge multiple image groups by using the merge to image group function.

To merge to image groups:



1. From the Navigator or the Inspector, select the Image Groups that should be combined into another Image Group.
2. Select **Merge to Image Groups** from the context menu.
3. Select the Image Group to which it should be merged.

☞ It is only possible to merge image groups when the images have the same camera properties and the same acquisition type.

3.7.4 Image Point Computation

3.7.4.1 Overview

Image Point Computation

To generate data from images, an image group must be used. Image groups are project objects and can be created in Infinity and imported from Captivate.

There are several sources of images that can be used to generate points from.

Points can be computed when:

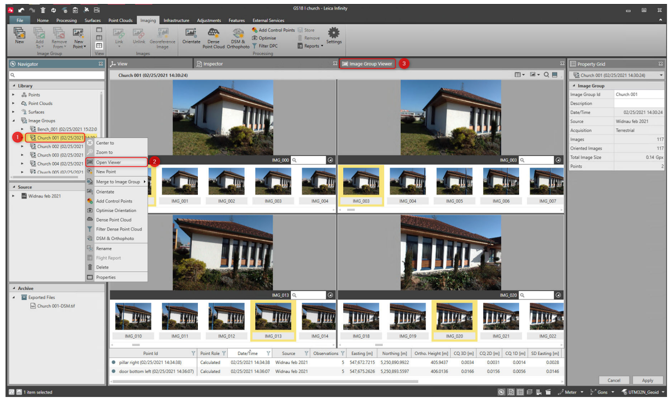
1. Images are taken from TPS image stations with either panorama images or overview images and these images cover the same object.
2. Images with GNSS geotags are acquired from the same camera such as GS18 I or a UAV, and the images have a good overlap.

Using the image group viewer and selecting a common target from the image, the forward intersection from each image position arrives at a 3D intersection point. These points have the point role computed.

3.7.4.2 Image Group Viewer

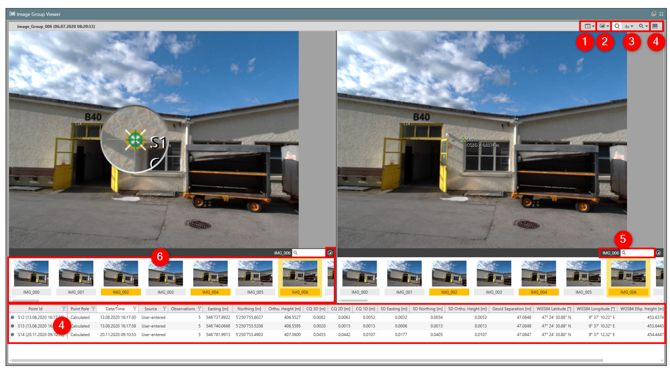
Image Group Viewer

To open the image group viewer:



1. From the Library or the Inspector, select the Image Group and from the context menu select **Open Viewer**.
2. The Image Group Viewer opens in a new window.

In the image group viewer you have the following options:



No.	Name	Description
1.	**Display Windows**	Set the number of working windows.
2.	**Filter**	Filter images to display.
3.	**Magnifier**	Switch magnifier On/Off to help select feature of interest.
4.	**Point Table**	Show or hide point table. Any points that were computed in the field or in the office are listed in the point table of the open image group.
5.	**Search**	Search for image name.
6.	**Image Strip**	Show or hide image strip. • For images measured from TPS, each image view allows selecting the TPS station and lists the images that were taken from that station setup. • For images measured with GNSS, the GS18 I or UAV, you can select which image to view from the image group.

See also:

The video **"Leica Infinity - Imaging Module - How to work with GS18 I data"** https://www.youtube.com/watch?v=1TwomI2lxw0

3.7.4.3 New Image Point

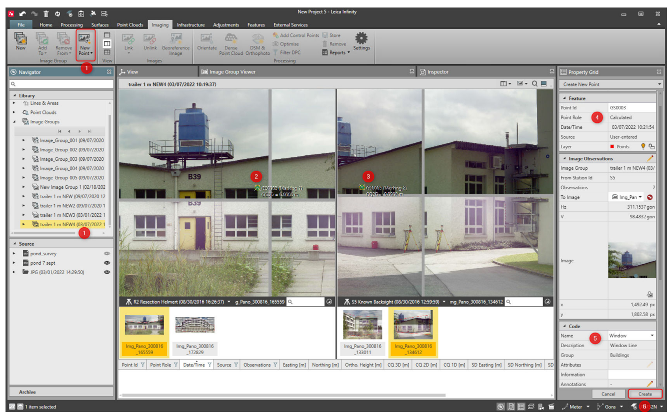
New Image Point

Points can be computed when:

1. Images are taken from TPS image stations with either panorama images or overview images and these images cover the same object.
2. Images with GNSS geotags are acquired from the same camera such as GS18 I or a UAV, and the images have a good overlap.

Using the image group viewer and selecting a common target feature from the image, the forward intersection from each image position arrives at a 3D intersection point. These points from images are stored to the points and have the point role computed.

To create new points from images:



1.	Select **New Point** either from the ribbon bar or the context menu.
2.	Mark the target point in the first image.
3.	Mark the target in the second image.
4.	Once the target has been selected in two or more images the projected point is shown.
5.	Add thematic data if needed.
6.	Select **Create** to store the point to the project.

When working with GS18 I images, the default mode for point creation is the automatic target selection mode. With this method, a single click defines the point. For other cases, the intersection mode is automatically set to manual, and the point can be created after being marked on two or more images.

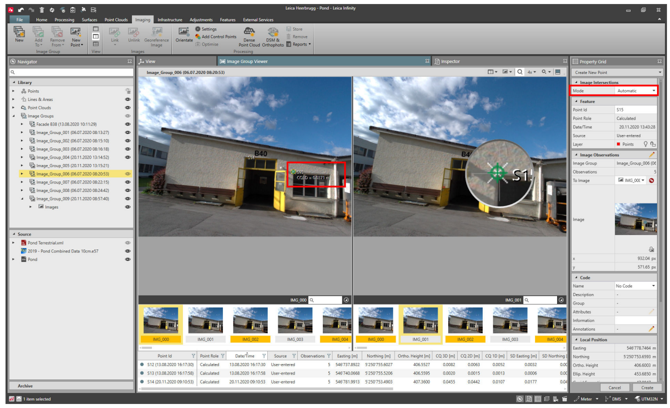


Image Point Settings

The computation of image points is following the same principle as with point averaging. You set a tolerance threshold for which the computed 3D point quality is acceptable and when the results are within this tolerance a position is computed. When the 3D accuracy is outside of that tolerance, then no point is computed.

Set the image point accuracy in the Points & Angles options page.

See also:

The video **"Leica Infinity - Imaging Module - How to work with GS18 I data"** https://www.youtube.com/watch?v=1TwomI2lxw0

The video **"Leica Infinity - Imaging Module - Points from Images"** https://www.youtube.com/watch?v=PDR4ROLrKbA

3.7.5 Image Processing

3.7.5.1 Overview

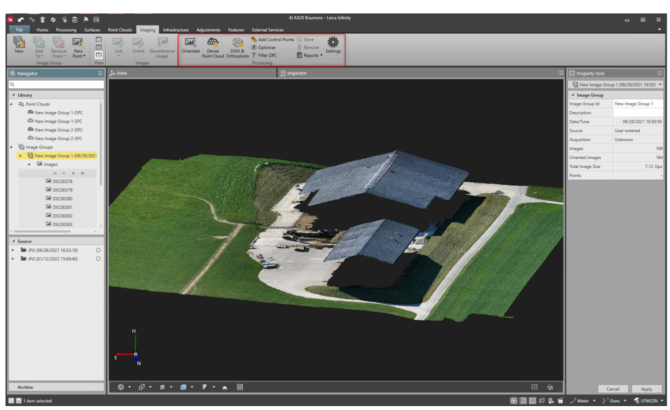
Image Processing

Images can be processed to create quality 2D and 3D data deliverables, such as Dense Point Clouds (DPC), Digital Surface Models (DSM) and Orthophotos.

Requirements:

- Point Clouds from Images option.

Here you can do the following:



No.	Name	Description
1.	Image Processing Settings	Adjust the Image processing settings for achieving best possible results.
2.	Image Processing Workflow	• Orientation of the Images - Calculate the exterior and interior orientation of images. • Marking of Control Points - Mark Ground Control Points (GCP) or Check Points (CP), before or after running the orientation. • Optimisation of the Orientation - Update the exterior and/or interior orientation of images, after changing either the control points (adding, removing or adjusting them) or the camera calibration settings. • Generation of the Dense Point Cloud - Create the DPC. If images are not oriented, an orientation is computed first. • Filtering of the Dense Point Cloud - Update the DPC after changing the DPC settings, to create a filtered improved version. • Generation of the DSM & Orthophoto - Create the DSM and the orthophoto. If the orientation and DPC are not stored, they are computed first.
3.	Image Processing Report	Explore the details.
4.	Task Manager	View the progress of the processing tasks for all the created projects.
5.	Export Image Processing Results	Export deliverables in various format files.
6.	Processing Considerations	Helps with understanding which component has an impact on processing.

The processing procedure includes three core steps:

1.	Orientation of the Images
2.	Generation of the Dense Point Cloud
3.	Generation of the DSM & Orthophoto

☞ Some further processes can be carried out if necessary.

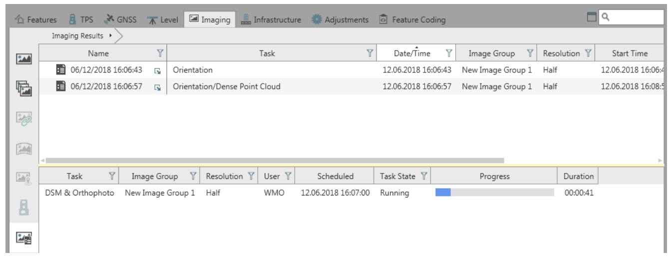
There are two approaches to process image data:

Step-by-Step - Process each task manually, step-by-step, and store it. This allows a better control on the results. Once stored, the output of each step (sparse point cloud, dense point cloud, DSM & Orthophoto) is available in the view and the numerical results in the inspector.

Onestep - Choose the end data result, such as DPC or DSM & Orthophoto, and all individual tasks are computed automatically.

Imaging Inspector

From the **Inspector > Imaging** tab, you can access Imaging Results. The view is split into two panes, the results pane and the Task pane:



See also:

The video **"Leica Infinity - Imaging Module - How to work with GS18 I data"** https://www.youtube.com/watch?v=1TwomI2lxw0

The video **"Leica Infinity - UAV Processing Part 1 - Import and orientate Aibot data"** https://www.youtube.com/watch?v=F8OdA9sy6q4

The video **"Leica Infinity - UAV Processing Part 2 - Add GCPs and update orientation"** https://www.youtube.com/watch?v=Y8hLA0RObFA

The video **"Leica Infinity - UAV Processing Part 3 - Create Dense Point Cloud, DSM and Orthophoto"** https://www.youtube.com/watch?v=XPpLS1cxZlk

---

Image Acquisition Suggestions

In order to get a good quality result, it is of main importance to acquire the data set to be processed correctly. A poor quality data set may lead to poor results or even to failed processes.

The following guidelines are recommended for a good outcome:

- Use digital cameras with a fixed lens, or keep it fixed throughout the acquiring phase.
- Use focal lengths between 20 mm and 80 mm (in 35 mm equivalent).
- Use 70%-80% forward overlap and 65%-80% side overlap.
- Avoid blurry images.
- Avoid low textured, moving or reflective objects.
- Avoid regions with shadows.
- Do not manipulate original images, for example crop or rotate them.
- Use control points to improve the orientation.
- Use check points to assess results accuracy.

☞ Fisheye, ultra-wide and macro lenses as well as hyperspectral images are currently not supported.

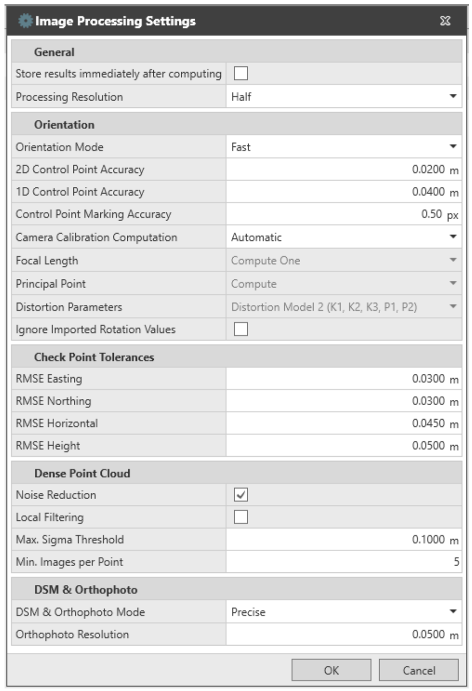
☞ Corrections for the rolling shutter effect are currently not supported.

3.7.5.2 Image Processing Settings

Image Processing Settings

Default settings are intended to be the most suitable solution in the more common cases. Nevertheless, for achieving a better final accuracy or for dealing with specific acquisitions, they have to be often tuned accordingly.

The following settings are available for the image processing:



General

Store results immediately after computing

The results are stored without any manual intervention.

Processing Resolution

The processing resolution is the image resolution, that is used for all the processing steps.

- **Full**, resolution uses the original image size for the processing.
- **Half**, resolution uses the images downscaled from the original size by a factor of 4.
- **Quarter**, resolution uses the images downscaled from the original size by a factor of 16.
- **Eighth**, resolution uses the images downscaled from the original size by a factor of 64.

This setting has to be chosen as a trade-off between the final desired quality and the processing time, considering the original image resolution as well. A full resolution process produces more accurate results, exploiting all the information contained in the acquired images, although requiring more time for generating the results

Orientation

Orientation Mode

The orientation mode allows the choice between two orientation processing options, precise and fast.

Select **Precise**, if the orientation processing computes the co-variance matrix considering the camera pose accuracy of each image within the selected image group.

The precise option provides an improved accuracy of the orientation including the resulting dense point cloud. Consider precise mode when the geometry reconstruction and data accuracy are the important factors of processing results.

Select **Fast**, if the orientation is processed without computing the co-variance matrix. The advantage to the fast option is a quicker orientation result. Consider fast option when accuracy is not the most important factor of processing, such as working with stockpile volume computations or general earth work surfaces.

☞ The precise option requires more hardware RAM during the orientation processing.

☞ When using the Fast orientation mode, there is the option to enable CUDA acceleration from **File** > **Preferences** > **Imaging**. This is available for machines using graphic cards with CUDA capabilities (such as NVIDIA GeForce). The benefit is shorter processing times.

Camera Pose Accuracy

This parameter is used to weight properly the GNSS information of the acquired images during the orientation step. It is related to the accuracy of the position of each image and not to the accuracy of the attitude of the UAS itself:

- Use a higher value when the UAS has a low-end GNSS receiver, no accurate RTK position or no raw data collection for post-processing.
- Use a smaller value when the UAS has an accurate RTK GNSS sensor or when collecting raw data for post-processing.

This setting strongly affects the orientation results and all the following steps. Pay attention with the choice of it.

Poor results (noise in the sparse point cloud) might be due to the current camera accuracy, either bigger or smaller than the one set. For example, when dealing with a low quality RTK position (because of interruption of the connection with the reference station or because of the acquisition environment), poor results can be generated due to a camera pose accuracy lower than expected. In this case, a bigger value is of advantage for a better result.

2D Control Point Accuracy/1D Control Point Accuracy

The use of control points is recommended for:

- Accurate georeferencing of the images and the deliverables that depend on them.
- Calibrating a camera or for improving the existing calibration.

These parameters are used to weight the control point horizontal position (2D) and height (1D) during the orientation step. They should reflect the real accuracy of the measured points.

These settings strongly affect the orientation results and so all the following steps. Pay attention with the choice of it.

These settings are used only for control points that do not have a coordinate quality (CQ 2D or CQ 1D respectively), so when using a point that already has CQ 2D or CQ 1D, the respective setting is ignored for that point.

Control Point Marking Accuracy

This parameter refers to how accurately a control point can be marked on the image. It is used to formulate the weight of the image observations of the control points in the orientation step. If for any reason control points cannot be marked on the images with a subpixel accuracy, for example blurry images, this value must be increased to make it more realistic.

Camera Calibration Computation

This parameter is connected to the interior orientation of the camera.

- **Automatic**: Default setting. When Ground Control Points (GCP) are used, the parameters of the interior orientation (focal length, principal point and distortion parameters) are estimated. If GCP are not marked on the images (both cases when Control Points (CP) only are used, or when no points at all are marked on the images) the camera calibration is not estimated. For images with interior orientation values stored in the metadata, those values are read and used as initial values: they are kept fixed if the GCP are not marked, estimated if the GCP are marked on the images, as described previously. When interior orientation values are not available, the interior orientation is always computed using the default distortion model.
- **User-Defined**: Using the User-Defined option, focal length, principal point and distortion parameters can be set, and the interior orientation parameters are estimated accordingly.

Focal Length

The focal length of the camera is read from the image metadata information (EXIF data). The more accurate the initial parameter is, the better the orientation of the images and the more accurate the scale of the reconstruction is. This value can be optimised during camera calibration and is then replaced with either one or two camera constants. These constants are the adjusted focal length equivalents.

- **Compute Two**: The use of two camera constants compensates for the non-squareness of the pixels.
- **Compute One**: In the other cases, the Compute One option can be used.
- **Do not compute**: Select **Do not compute** if the camera is already calibrated and the existing values are reliable enough.

A scale issue in the object reconstruction might depend on a wrong used value. Either perform a calibration of the camera or carry out the auto-calibration during the orientation step, to get a more reliable focal length value and a better result in the reconstruction.

Principal Point

By default, the principal point it is placed to the centre of each image.

- **Compute**: Select **Compute** to improve the existing value for the principal point.
- **Do not compute**: Select **Do not compute** if the camera is already calibrated and you want to keep the existing values.

Distortion Parameters

This parameter controls the distortion of the lenses used to acquire images. The default distortion model works well for most of the frame cameras. However, for cases when additional information about the lens distortion model is available, further models are available:

- **Distortion Model 1 (K1, K2, P1, P2).**
- **Distortion Model 2 (K1, K2, K3, P1, P2). (Default).**
- **Distortion Model 3 (K1, K2, K3, K4, P1, P2, S).**
- **Distortion Model 4 (K1, K2, K3, Kd1, Kd2, Kd3, P1, P2).**
- **Distortion Model 5 (K1, K2, K3, K4, K5, K6, Kd1, Kd2, Kd3, P1, P2, P3, P4, P5, P6, S1, T1).**
- **Do not compute.**
- **None - Undistorted Images.**

Parameter	Description
Kx	Radial distortion parameters.
Px	Tangential distortion parameters.
S	Skew parameter when using Distortion Model 3.
S1, T1	Shear and skew parameter when using Distortion Model 5.

☞ The focal length, principal point and distortion parameter options are only selectable if the camera calibration computation is set to User-Defined.

Ignore Imported Rotation Values

When this box is checked, the rotation values stored in the metadata of the images are not used in the orientation process.

Use this setting if imported values are wrong, as in, if the images appear incorrectly rotated or upside down in the graphical view. In all the other cases do not use this setting, since it deteriorates the solution.

Check Point Tolerances

Along with control points, Infinity supports the use of check points.

In order to have an independent measure for assessing the accuracy of the image orientation, a tolerance for each of the Root Mean Square Error (RMSE) on control points has to be set. The 3D position of the check points is not used during the orientation, yet the computed 3D position of each check point is compared to its current position. The differences are used to evaluate the overall accuracy of the orientation.

The following criteria, derived from ASPRS guidelines, are used to determine when the analysed check point should be considered as out-of-tolerance:

- If the computed RMSE values for Easting, Northing, horizontal or height exceed the ones entered.
- If the absolute value of the mean of Easting, Northing or height exceeds the 25% of the respective RMSE value entered.
- If the absolute value of ΔEasting, ΔNorthing or ΔHeight for a check point exceeds three times the respective RMSE value entered. In this case, the results are marked in bold red text.

Dense Point Cloud

In the generated dense point cloud there can be some outliers, due to blurry images or poor overlap. A filter can be used to remove them and generate a more accurate dense point cloud.

Noise Reduction

Used to remove inconsistent points above or in front of a reconstructed surface.

Local Filtering

Best used with data sets with high image overlap. Used to remove points that are below or behind a visible surface and to create sharper edges.

Max. Sigma Threshold

The max. sigma threshold is the maximum value of the uncertainty (sigma) that a point can have in order not to be filtered out.

- Increasing this value may lead to a more complete but noisier point cloud.
- A smaller value guarantees more accurate points, although reducing the density of the dense point cloud. If a too noisy dense point cloud is generated, a smaller value can be of advantage.

Min. Images per Point

This parameter defines the minimum number of images that a point must be seen in to be considered valid.

- Increasing this value reduces the noise, although a lower number of points in the point cloud might be computed. If a too noisy dense point cloud is generated, a bigger value can be of advantage.

DSM & Orthophoto

DSM & Orthophoto Mode

Digital Surface Model (DSM) & Orthophoto mode allows the choice between two computation methods, Precise and Fast.

With **Precise**, a 3D DSM is generated that can better model the dense point cloud. This 3D DSM is used for generating the orthophoto. This method provides the best quality orthophoto output, but can take longer to process.

With **Fast**, a 2.5D DSM is generated that approximates the model of the dense point cloud. This 2.5D DSM is used for generating the orthophoto. This method has three options for defining the density of the DSM. Each option affects the output of the orthophoto.

Triangle Size

When using the Fast DSM & Orthophoto mode, triangle size can be chosen according to the size of the details to be reconstructed. Use large triangles for the quickest processing time and when fine details are not of interest. Use medium or small triangles to add more detail to the DSM and improve the output of the orthophoto.

Orthophoto Resolution

This parameter controls the detail level of the orthophoto. Generally, this value can never be smaller than the ground sample distance computed for the image group being processed.

- Decreasing the parameter, increases the resolution of the orthophoto, but also the computation time for its generation.
- Increasing the parameter, decreases the resolution of the orthophoto and reduces the computation time. The orthophoto is smaller in size.

3.7.5.3 Image Processing Workflow

3.7.5.3.1 Orientation of the Images

Orientation of the Images

After the import and the image group creation, the images can be orientated.

To orientate the images, do the following:

1. Set a master coordinate system in the project. See Working with Coordinate Systems.
2. Select the image group from the Navigator or the Inspector.
3. Select **Orientate** from the Imaging ribbon bar or from the context menu.

☞ Images can be oriented only if they are geotagged and have (at least a rough) camera calibration.

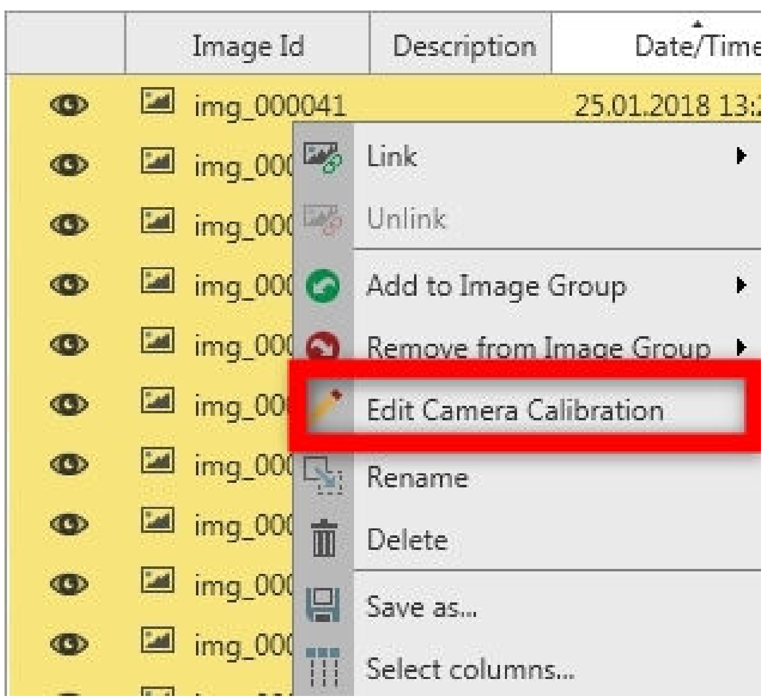
☞ If the previously mentioned requirements for images and coordinate system are fulfilled, images camera poses should be shown in the view. If not, check that the coordinate system used is the appropriate one for the area considered.

☞ Infinity reads information for camera focal length and computes the principal point position based on the information stored in the EXIF data.

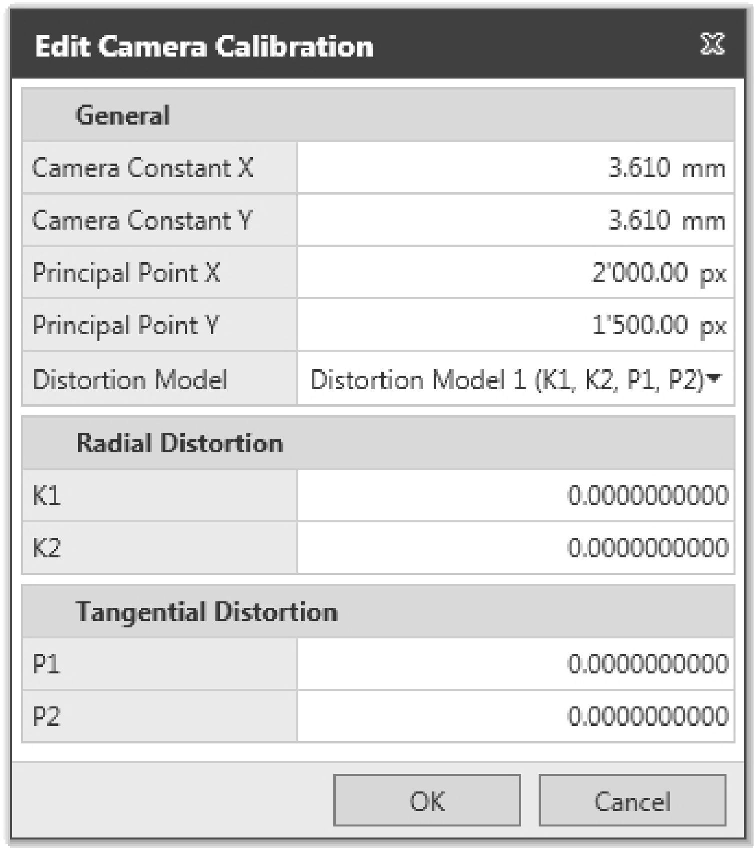
If the camera calibration parameters are provided, for example in the case of a factory calibration, they can be manually entered.

To enter the camera calibration parameters, do the following:

1. Select all the images of the image group from the Inspector or from the Navigator.
2. Use the context menu option **Edit Camera Calibration**.



3. According to the selected distortion model, the calibration values can then be entered.



4. As described in Images, once the orientation is triggered, the processing task appears in the Task pane of the Imaging tab in the Inspector, and in the Task Manager.

5. When the orientation is computed, you can drill in the orientation result to view details about:
   - The exterior orientation.
   - The interior orientation.
   - The control and the check points, if any have been used. For further information, see Marking of Control Points.

6. To store the orientation result, highlight it and select **Store** from the Imaging ribbon bar or from the context menu. The sparse point cloud appears in the View.

☞ Together with the orientation of the images, Infinity computes the 3D position of the common features that were matched and used for orientation, also called tie points. These points form a sparse point cloud. This point cloud shows a rough representation of the area, that is reconstructed during the dense point cloud computation.

☞ After the orientation, the images along with the updated position in the EXIF data can be exported.

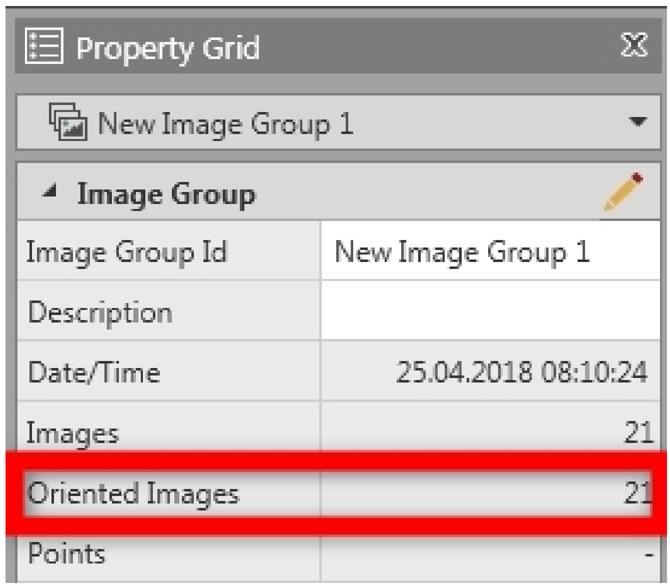
If you realise that the used settings are not compliant with the desired outcome, you can remove the orientation result in order to run a new orientation.

To run a new orientation, do the following:

1. Highlight the orientation result and select **Remove** from the Imaging ribbon bar or from the context menu.
2. You can also select the image group and then **Remove** the stored orientation.

☞ Every time you remove an orientation from an image group, the position and the attitude of each image rollback to the values they had after import.

To see the number of Oriented Images, select the **Image Group** and view the Property Grid. This information is also available in the Imaging tab of the **Inspector** > **Image Group**.



☞ The oriented image groups have a special icon.

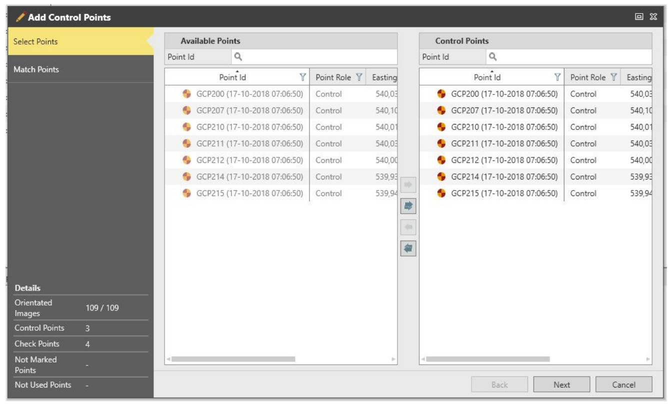
3.7.5.3.2 Marking of Control Points

Marking of Control Points

Infinity supports the use of control points and check points. You can mark them either before running an orientation task or after storing it.

To mark control points, select the image group and select **Add Control Points** from the **Imaging** ribbon bar or from the context menu.

This action starts the Mark Control Points wizard:



Select Points

In this window, you select the points that you would like to mark on the images. Any point from your project (imported or calculated) can be added as control point, as long as it has a 3D position.

1. Select the points you want from the Available Points list, move them to the Control Points list.
2. Select **Next** to proceed to the second page of the wizard.

Match Points

Infinity offers two ways to mark and match control points:

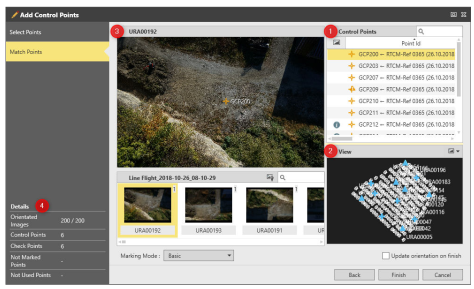
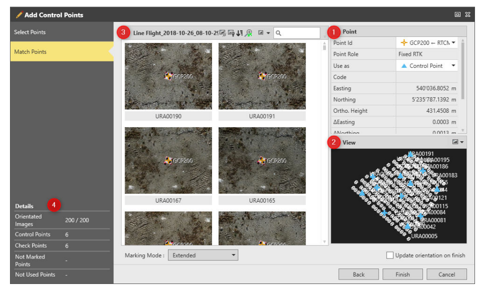
- Basic Marking Mode: Used to mark multiple control points on one image. It is the default mode when there is no orientation result stored.
- Extended Marking Mode: Used to speed up the marking of a selected control point on multiple images. It is the default mode when there is an orientation result stored.

General Description

The main components of both marking modes are:

No.	Group Name	Description
1.	**Point List**	The place to select which point to mark.
2.	**Map View**	Shows the images and the control points that have been added in the list. It helps checking the distribution of marked points. You can use the map view option to hide or show the images. If the image group is already oriented, you can also restrict the images that are shown to those that the selected point can be reprojected on. Once a point has been marked, it is displayed in the map view with one of the following icons: If it is used as a control point. If it is used as a check point. If it is not used at all.

No.	Group Name	Description
3.	**Marking Pane**	Where the points are marked on the images. The two marking modes use this pane in a different way, refer to the following.
4.	**Wizard Navigator**	Shows details about: • The total number of orientated images/the total number of images in the image group. • The total number of control points. • The total number of check points. • Which points have not been marked. • Which points are not used.

Basic marking mode layout:	Extended marking mode layout:
	

The Warning section provides information on possible weaknesses of the marking procedure, such as points marked on one image only or if not enough control points are used.

The points that have been added in the point list can be used as control points or check points. This option is available in the Use as column/property in both marking modes.

The basic marking mode:

Apart from the common components that were described above, the page also comprises a thumbnail list of the images that belong to the selected image group.

The marking window shows the image that is active for control point marking.

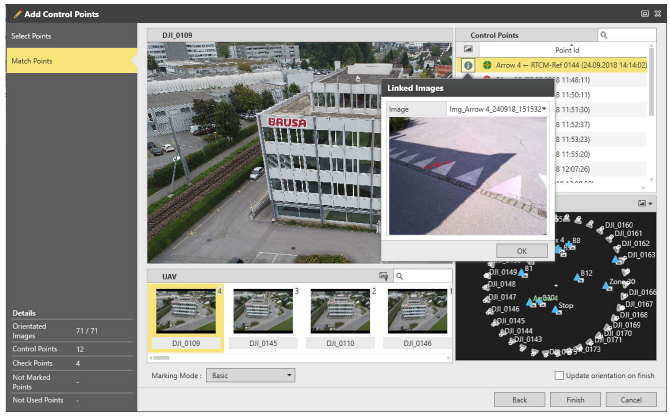
The list of control points includes all the points that can be marked.

To mark a control point:

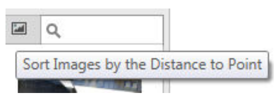
1.	Select the control point from the list or the map view.
2.	Select the image you want to mark this control point on from the image thumbnail list or the map view. The selected image appears in the marking window.
3.	Mark the point on the image.

If another point is found on the selected image and must be marked, navigate to the position on the image and use the context menu to select the point.

If there are images that have been linked to the control points, they are available in the flyout:



To speed up the point marking phase, Infinity can sort the image thumbnail list by the incrementing 2D distance of each image to the selected point. This increases the probability that the selected point appears in the first images of the list. To activate this sorting, select **Sort Images by the Distance to Point**. The same option is available in the Extended Marking Mode.



To activate this sorting, select **Sort Images by the Distance to Point**.
To sort the images by name, select the same option again.

The extended marking mode:

In this point-based view, multiple images with marking suggestions are displayed for a selected point.

To mark a control point:

1. Select the control point from the list or the map view.
2. Mark the point on the image.

If there are images that have been linked to the control points, they are displayed at the bottom of the pane.

The marking pane includes a set of options to:

- Show only images a selected point can be reprojected on (valid only if an orientation has already been stored for the selected image group).
- Sort images by distance to point or by name (the same option is available in the basic marking mode).
- Sort images by ascending/descending reprojection error (valid only if an orientation has already been stored for the selected image group).
- Zoom in/out in all images using the same zoom factor (can be activated/deactivated).
- Show images in small, medium or large size.
- Search for a specific image.

☞ The context menu that is available in the marking image offers short-cuts that can speed up the marking process.
If a point is already marked on an image, place the mouse over the point marking and use the context menu to: remove, use as check/control point or do not use the specific point.
If an orientation is already stored, the control points and the check points that have at least one out-of-tolerance value are indicated with the icon.
You can switch between the basic and the extended marking mode at any time: the markings and the settings are saved when switching from one mode to the other.

To close the wizard and store the markings, select **Finish**.

If you want to change the control points, for example:

- Add or remove control points.
- Improve markings.
- Disable control points.
- Change control points to check points.

You can start the wizard at any time.

☞ If the images are already oriented, the selected point is automatically detected on the selected image. But this marking is not used for updating the image orientation, unless it is confirmed. The suggested marking is displayed with .

☞ When possible, the use of control points acquired at different heights is recommended in order to improve the orientation step.

3.7.5.3.3 Optimisation of the Orientation

Optimisation of the Orientation

Optimisation is the process of updating the exterior and/or interior orientation of images, by making changes to the control points or to the camera calibration values.

The initial stored orientation can be optimised so that:

- Any wrong point marking is manually removed.
- Any control point with wrong 3D position is disabled.
- The camera calibration is fine-tuned to fit the specific image capture.

Since most cameras that are used do not retain a constant internal geometry, the camera optimisation can improve the accuracy of the interior and exterior orientation and its deliverables.

The orientation of an image group may take a considerable amount of time. However, the optimisation only triggers a final bundle adjustment, without recomputing the orientation from the beginning.

The optimisation of the orientation can be triggered:

- By the control points wizard.
- By highlighting the oriented image group and selecting **Optimise** from the Imaging ribbon bar or use the context menu.

Optimisation using the control points wizard:

1. After making a change in the control point wizard for an oriented image group, on the second page there is now an additional check box.

2. Activate the **Update orientation on finish** option to optimise the orientation.
3. Select **Finish** in the wizard to start a new processing task. The orientation is updated and optimised with the changes you made in the control point wizard.

Optimisation of the camera parameters:

To fine-tune the interior orientation of the images, do the following:

1. Open the **Settings** from the Imaging ribbon bar and make the changes you want to the Camera Calibration Optimisation section.
2. Highlight the oriented image group and select **Optimise** from the Imaging ribbon bar or the context menu.

3.7.5.3.4 Generation of the Dense Point Cloud

Generation of the Dense Point Cloud

To create a dense point cloud from an image group, do the following:

1. Highlight the image group.
2. Select **Dense Point Cloud** from the Imaging ribbon bar or use the context menu.
3. Once the dense point cloud process is triggered, the processing task appears in the Task pane of the Imaging tab in the Inspector and in the Task Manager. See Images.
4. To store the result, highlight the result and select **Store** from the Imaging ribbon bar or from the context menu. The dense point cloud appears in the View.

The dense point cloud can be filtered according to the settings described in the Image Processing Settings.

To remove a stored dense point cloud result, do the following:

1. Highlight the dense point cloud result.
2. Select **Remove** from the Imaging ribbon bar or from the context menu.

To create a report for the dense point cloud, do the following:

1. Highlight the dense point cloud result.
2. Select **Reports** and then **Dense Point Cloud Report** from the Imaging ribbon bar or from the context menu.

☞ If the images are not oriented at the specified processing resolution or the orientation is computed but not stored. An orientation is computed first and then a dense point cloud is created, based on the computed orientation.

The Dense Point Cloud (DPC) can be also used to:

- Create surfaces, contours and compute volumes.
- Draw features, like lines and areas.
- Measure distances between objects.

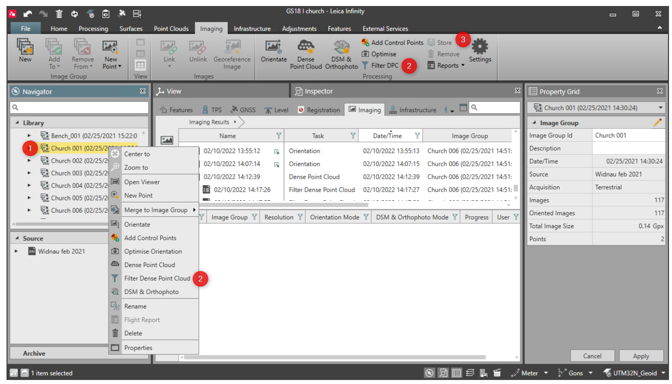
3.7.5.3.5

Filtering of the Dense Point Cloud

Filtering of the Dense Point Cloud

The DPC can be filtered according to the settings described in the Image Processing Settings.

To filter the stored DPC, do the following:



1. Highlight the image group object from which you created the DPC.
2. Select **Filter Dense Point Cloud** from the context menu or the ribbon bar.
3. To store the filtered DPC result and see it in the View: Highlight it and select **Store** from the Imaging ribbon bar or the context menu.

☞ The Filter DPC functionality is active only when a DPC task is already stored.

3.7.5.3.6

Generation of the DSM & Orthophoto

Generation of the DSM & Orthophoto

To create a Digital Surface Model (DSM) and an orthophoto from an image group, do the following:

1. Highlight the image group.
2. Select **DSM & Orthophoto** from the Imaging ribbon bar or use the context menu.
3. To store the DSM & orthophoto result, highlight the result and select **Store** from the Imaging ribbon bar or use the context menu.

To remove a stored DSM & orthophoto result, do the following:

1. Highlight the DSM & orthophoto result.
2. Select **Remove** from the Imaging ribbon bar or from the context menu.

To create a report for the DSM & orthophoto, do the following:

1. Highlight the DSM & orthophoto result.

2. Select **Reports**, then **DSM & Orthophoto Report** from the Imaging ribbon bar or from the context menu.

☞ If the images are not oriented at the specified processing resolution or the previous step is computed but not stored. An orientation is computed first, then a dense point cloud is created, based on the computed orientation. Finally a DSM & orthophoto is created, based on the computed orientation and the dense point cloud. It is equivalent to running a onestep process. For further information, see Image Processing.

☞ The DSM is used to create orthophotos but can also be used to compare between different models.
Orthophotos can be used as background maps in Infinity. They can also be exported to onboard software, to Leica ConX, as well as to third party software.

3.7.5.4 Image Processing Report

Image Processing Report

To create a report for the orientation, select the orientation result and select **Reports > Image Orientation Report** from the Imaging ribbon bar or the context menu.

Project Details

This section includes information about the project, such as the project name, the customer details and the master coordinate system.

Image Group Information

This section includes information about the selected image group:

- The ID.
- The total number of images and their resolution.
- The sensor and the camera model.
- The EXIF focal length.
- The total size of the images in Gpx.

Orientation Settings

This section includes information on the settings that were used to compute the orientation of the images. The settings are described in detail in Image Processing Settings.

The check point tolerances are the manually entered values used to assess the quality of the check points.

Interior Orientation

This section includes information on the parameters of the interior orientation.

Calibration Source:

- **Pre-Calibrated**: The initial values for the parameters are imported to the project, for example from Leica Aibot.
- **User Entered**: The initial values for the parameters are manually edited.
- **Unknown**: The initial values for the parameters were computed using information from the EXIF image metadata.
- **Computed**: The initial values for the parameters were computed in a previous step by self-calibration.

The initial value, the calibrated value and their differences are shown for each parameter of the camera and the lens of the Image group. Normally, the differences are expected to be small, but there is no limitation. Yet, a large difference may indicate instability of the self-calibration or the orientation. In most cases, this issue can be addressed by adding control points as explained in Marking of Control Points.

Exterior Orientation

This section includes information about the exterior orientation of the images.

If control points have been used, the information shown here reflects the absolute position of the images. Otherwise, the information is shown with respect to the relative position of the images.

General Information

General statistics for the exterior orientation are shown in this subsection.

- Check the **Minimum Number of Tie Points Per Image**: If this value is zero, there is at least one image that has not been orientated, due to a not optimal acquisition. For example blurry images, not optimal camera network or flight height.

Absolute Camera Pose Standard Deviations

This table shows the mean and the standard deviation of the mean, for the standard deviations of the camera poses. Depending on the initial orientation estimates, these values are more or less close to zero. A value that is bigger from the rest or from zero, indicates a possible bias of this parameter. In most cases, this can be fixed by adding control points to the orientation.

Control Points

This table shows the residuals in each direction, the mean reprojection errors and the total number of images each control point has been marked on.
The mean reprojection errors are checked with the nMAD (normalised Median Absolute Deviation) criteria. Possible outliers are marked as out-of-tolerance.

- If there are flagged values, review the markings of the specific control points.

The mean and the standard deviation are computed for each column. These values are more or less close to zero. A value that is bigger from the rest or from zero, indicates a possible bias.

In this case:

- Check that the control points that are used have a good distribution.
- Check that the control points have been correctly marked on at least four images.
- Check that there is no typing error in the coordinates of the control points.

More information on control points can be found in Image Processing Settings.

Check Points

This table shows the residuals in each direction, the mean reprojection errors and the total number of images each check point has been marked on. The mean reprojection errors are checked with the nMAD criteria. The Δ values, the mean and the computed RMSE are checked against the tolerances defined in the image processing settings. Possible outliers are marked as out-of-tolerance.

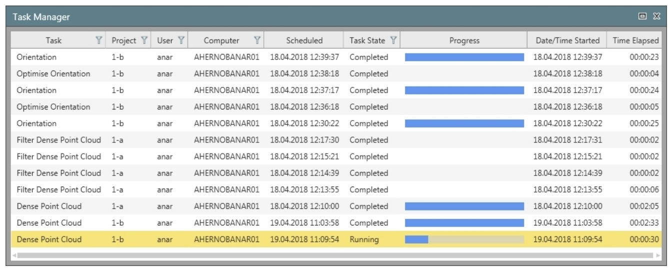
- If there are flagged values, review the markings and the coordinates of the specific check points.

More information on control points can be found in Image Processing Settings.

3.7.5.5 Task Manager

Task Manager

The task manager is launched from the quick access bar. Here you can see all processed tasks, in progress or to be processed when using a worker host machine.



Once you start processing a task, it runs in the background. You can continue working with Infinity or even change projects and submit another task to the queue that starts as soon as the previous task ends. You can view the progress of each task in the Task Manager. If the task is finished in the meantime, the result is moved to the results pane, for you to store.

☞ Processing tasks create an asset directory on the computer where the processing takes place. The default directory is: C:\Users\[USER]\AppData\Local\Leica Geosystems\Infinity\mvs\assets\. The default directory can be modified, as described in Imaging.

3.7.5.6 Export Image Processing Results

Export Image Processing Results

Infinity can export:

- The oriented images with the updated position in the EXIF data.
- The dense point cloud, in the standard formats used for point clouds.
- The DSM and orthophotos, as georeferenced images.
- For more information on exporting, see Data Export.

3.7.5.7 Processing Considerations

Processing Considerations

The number of images that can be processed with Infinity depends on several factors:

- **RAM**: The image group size or Gpx size is related to the amount of system RAM. It is important to consider the size of data sets to be processed to ensure the platform has enough system RAM. Processing times can also be reduced when enabling XMP profiles offering increased RAM speed.
- **CPU**: The processing is directly affected by the speed of the CPU. Consider a high clock speed CPU (greater than 3 GHz) with a high boost speed (greater than 4.5 GHz). A CPU with high core count improves processing times, currently the biggest impact of cores is with the Dence Point Cloud (DPC) step.
- **GPU**: Using NVIDIA GPU is recommended with CUDA capabilities and speeds up the processing steps of the orientation and DPC.

Processing at half resolution is suggested as the best balance between density and completeness of data and time to process. For high-resolution images > 36 MP, quarter resolution is also providing a good balance of data to process time.

When working with large data sets, creating several smaller image groups helps with the processing time.

Considering a 20 megapixel image, 16 GB of RAM is recommended to process 100-120 images at full resolution. The processing limits depend on the total pixel count of all images being processed.

It is possible to process images on the local machine or to use a dedicated worker host machine. When working with larger data sets its recommended to use a dedicated worker host.

Suggested System Requirements

Components	Minimum	Recommended	Recommended Worker Host
Windows Operating System	Windows 8, 10 (64 bit)	Windows 10 (64 bit)	Windows 10, Server 2016 or newer (64 bit)
Processor Intel or AMD	Multi-Core base 2.5 GHz or greater	Multi-Core base 3 GHz or greater	Multi-Core base 4 GHz or greater
Memory	32 GB DDR4 RAM	128 GB DDR4 RAM XMP enabled	128 GB or greater DDR4 RAM XMP enabled
Storage	512 GB SSD	1 TB or greater SSD	1 TB or greater, multiple drives for storage
Graphics	2 GB NVIDIA	8 GB NVIDIA CUDA capable	8 GB or greater NVIDIA CUDA capable

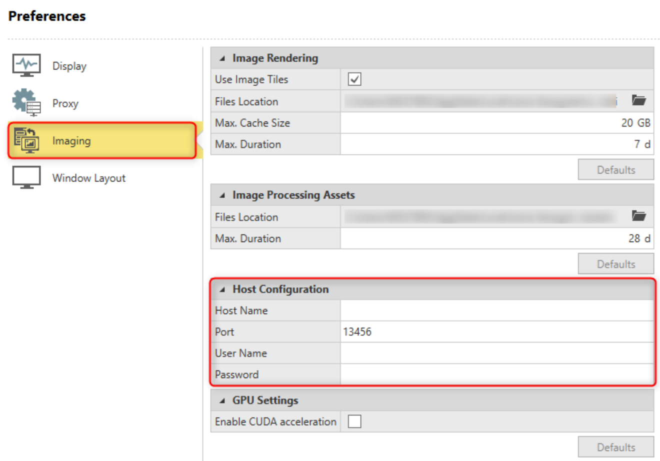
Setup Worker Host

The image processing tasks can be sent to a different computer (host) for processing, as long as the following requirements are met:

- Both computers have an Infinity licence with the point clouds from images option.
- Both computers have the same version of Infinity.
- The Infinity image processing worker host is already running on the computer that to be used for processing.
- Infinity is running on the computer that sends the processing task.
- The computers are working within the same network.

By default, the host for image processing tasks is the local computer, that is the computer where the project is already running.

To define the host, do the following:



1. Select **File** and then **Preferences**.
2. Select the **Imaging** tab.
3. Enter the necessary information regarding the Host Name, the Port, the User Name and the Password.

3.8 Infrastructure

3.8.1 Overview

Infrastructure

The infrastructure module allows you to create basic road designs as well as convert data from external files that already have a design.

When a design is imported from an external file, the object is created directly. However, you can still edit the information that has been imported with the result of an updated design.

Once a road design is ready, Infinity can export it to a format suitable for field use.

See also:

Export to Leica ConX

The tutorial **"How to manually create a road and use it for volume calculations"** https://leica-geosystems.com/-/media/2b014fed-0b854491b295bac4ee1df766.ashx

The tutorial **"How to prepare a Road job from data defined by cross sections"** https://leica-geosystems.com/-/media/7508f29835254de1be874acac79736ef.ashx

The tutorial **"How to create a Road design from polylines in a CAD file"** https://leica-geosystems.com/-/media/45f1d1d913764be29b-c2b8d9cc13b325.ashx

☞ The tutorial can be downloaded in the Localisation Tool.

Alignments:

An alignment, is a line object used to denote the path of a road, railway, tunnel, and so on. A full design requires a 3D alignment in order to serve as centreline and to calculate the rest of the infrastructure elements.

See also:

Alignment Objects

Road Objects:

A road object, consists of a group of stringlines. The stringlines can be defined with a centreline and cross sections or imported from a design file. If imported from a design file it is possible to set one of the stringlines as centreline, which creates an alignment object in the background.

See also:

Road Objects

Cross Sections:

A cross section, is a slice or a cut at a certain chainage of the road. In a vertical view, it indicates the position of a roads material layers.

Stringlines result from interconnecting the nodes of subsequent cross sections.

See also:

Cross Sections

Infrastructure Manager:

The infrastructure manager, is the tool that allows for the input and the editing of the different components that lead to a full 3D alignment object and, based on an existing alignment acting as centreline, for the creation, editing and viewing of the rest of elements that lead to a finished road design object.

If any potential conflicts are found, Infinity notifies you.

See also:

Infrastructure Manager

Material Layers:

Within Infinity, the material layers represent the different layers of a road and group sets of stringlines that belong to the same level, material or phase of construction and are, therefore, thematically interconnected.

See also:

Material Layers

3.8.2 Alignment Objects

Alignment Objects

Each road object needs at least one alignment object which serves as the centreline of the road.

The centreline is the main line in a road model. It is the reference axis that determines the distance along the road (chainage) and across it (offsets).

All alignments available in a project are grouped and listed in the navigator and the inspector.

By default they are imported or written to the alignments layer.

To create a new alignment:

1.	Select **Alignment** from the Infrastructure ribbon bar. Next to the Property Grid the Create New Alignment tab opens up.
2.	Enter the alignment properties and select **Create**.
3.	The Infrastructure Manager opens up next to the Inspector.
4.	In the Infrastructure Manager builds up the alignment by defining its horizontal (mandatory) and vertical (optional) elements. For both the POB/VPOB (Point of Beginning/Vertical Point of Beginning) are already predefined assuming the Start Station as defined in the Alignment Properties.
5.	In the Inspector view right-click on the POB and select **Add below** from the context menu. Alternatively, select **Add below** in the **Infrastructure** > **Road Design** ribbon bar. By default an element of type Straight is added. Optionally choose a different Type and Method and edit the elements as required.
6.	Continue adding new elements below or in between (above).

To edit existing alignments:

1.	Right-click on the Alignment object in the Navigator or the Inspector and select **Edit** from the context menu.
2.	Edit the elements of the alignment in the Infrastructure Manager.

To use an alignment as centreline in a road:

1.	In the Navigator or in the Inspector select the Road object.
2.	In its Property Grid go to Details and select **Edit** next to Centreline.
3.	Choose a different alignment from the list and confirm your selection with **OK**.
4.	Select **Apply** in the Property Grid to update the Road object.

3.8.3 Road Objects

Road Objects

Road objects can be cross section based or stringline based.

Cross section based road objects:

Cross section based road objects consist of:

- A 3D alignment that has been created or chosen to serve as its centreline.

  ☞ Any of the alignments available within the project can be chosen as centreline.

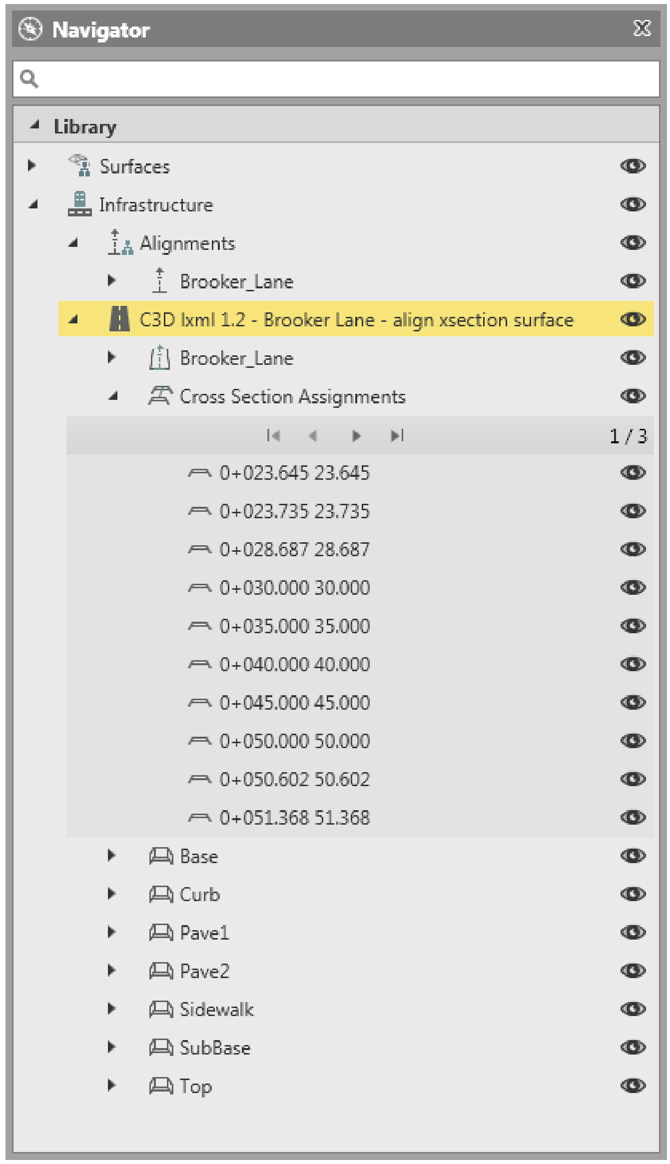
- Cross section templates assigned to chainage values along the centreline.

  ☞ Cross section templates remain in the project even if assignments are deleted.

- Several material layers containing stringlines of the same level, material or phase of construction.

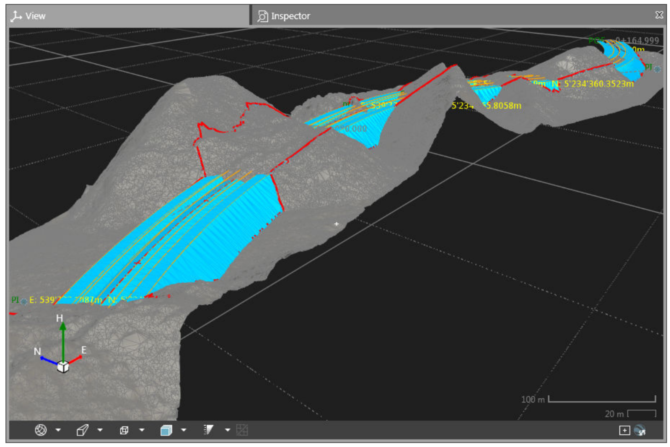
☞ Daylight stringlines require a target surface to exist in the road object.

Example:



Optionally a target surface can be assigned to the road object to visualise cut or fill slopes. It must be available within the project as a surface, before it can be chosen as target surface.

Example:



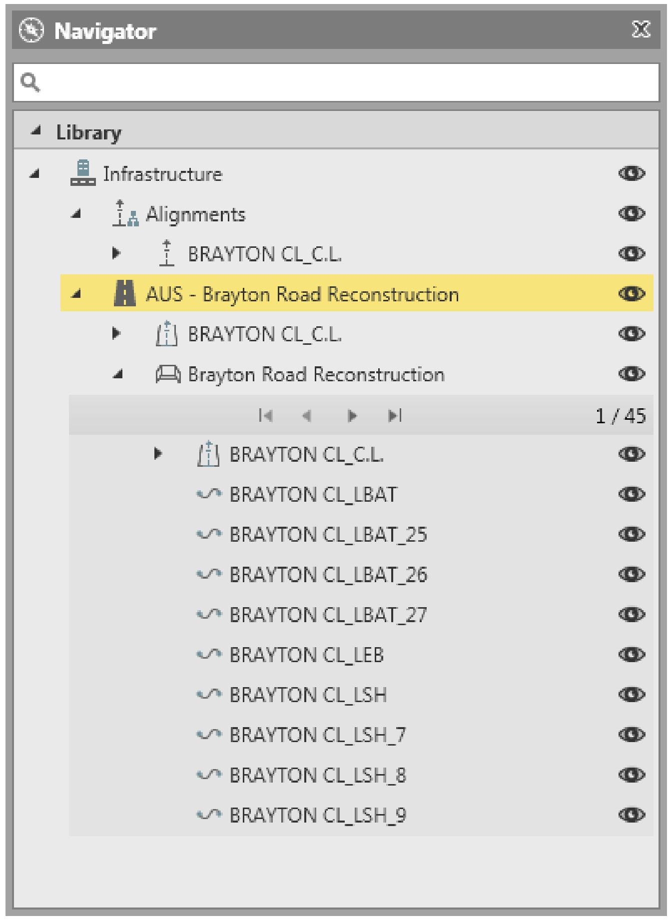
☞ If the target surface is removed from the road object, the so-called daylight stringlines which indicate the intersection of the road design with the terrain model also disappear.

Stringline based road objects:

Road objects imported from external files often come in in the form of a list of stringlines.

In this case the stringlines define the road object, which has typically been calculated in a 3rd party software and sent to Infinity for its preparation and conversion to a field-compatible format.

Example:



Any of the stringlines available within the road object can be set as centreline:

Right-click on the stringline and select **Set as Centreline** from the context menu.

To create a new road object:

1.	Select **Road** from the Infrastructure ribbon bar. Next to the Property Grid the New Road tab opens up.
2.	Enter the road properties. Go to Details and select **Edit** next to Centreline to optionally select a centreline. All alignments available in the project are available for selection. Select **Edit** next to Target Surface to optionally select a target surface. All surfaces available in the project are available for selection. Having a target helps when you intend to assign cross sections to the road.
3.	Select **Create**.

The new road object is added to the list in the navigator and the inspector.

To edit a road object:

1.	To edit a road right-click on the Road object and select **Edit** from the context menu. The Infrastructure Manager opens up next to the Inspector.
2.	To assign cross section templates go to the Cross Sections tab. Each assigned cross section is added under Cross Section Assignments in the Inspector and the Navigator.
3.	Optionally add Material Layers. To add a new layer select **Road Design** > **Material Layer** in the Infrastructure ribbon bar.

☞ When you assign cross sections to a stringline based road the nodes have to be assigned to material layers of their own.
Simple stringlines cannot share a material layer with stringlines resulting from cross section assignments.

3.8.4 Cross Sections

Cross Sections

Within Infinity cross sections can be imported or manually created. Manually created cross sections are always stored as a template and are as such available for roads.

Imported cross sections are not stored as a template automatically, but you can copy them to a new cross section template.

See also:

Cross Section Assignments

If available as a template, a cross section can be selected to be assigned to a road from the list in the Cross Sections tab of the Infrastructure Manager.

Assigned cross sections can be edited and if available as a template your changes can be updated to the template for further use.

☞ Assignments using a modified template are not updated automatically. Reassign the updated template if necessary.

To manually create a cross section choose to create a cross section template:

1.	Select **Cross Section** from the Infrastructure ribbon bar.
2.	Enter the template properties. Go to Details and select the **Pencil** icon next to material layers to select the layers that shall be available to define elements upon. ☞ If not, all elements are written to a default material layer. ☞ Available for selection are all the layers of the road object. New layers can be added any time.
3.	Select **Create**.

See also:

Material Layers

The new template becomes available for selection in **Infrastructure**, under the **New Cross Section Template** tab.

On how to extract a new cross section by interpolation between existing cross section assignments, see Cross Section Assignments.

3.8.5 Material Layers

Material Layers

Within Infinity the material layers represent the different layers of a road and are used to group sets of stringlines that belong to the same level, material or phase of construction.

☞ It is not possible to combine imported stringlines and cross section nodes in the same material layer.

Layers defined for infrastructure or road objects are imported and stored to the road object as material layers. Material surfaces that are created from material layers can have their own code or style attributes.

Stringlines imported or created from cross sections are stored to the layer stringlines and can have their own code and style attributes.

See also:

Layer Manager

To create a new material layer:

1.	Select the Road to which you want to add a new Material Layer.
2.	Select **Road Design** > **Material Layer** in the Infrastructure ribbon bar. Next to the Property Grid the New Material Layer tab opens up.
3.	Give the new Material Layer a name.
4.	Optionally, select **Edit** next to Centreline to select a centreline.
5.	Select **Create**.

All material layers are visible from the navigator. When used with a cross section based-road, the material layers are also visible in the **Infrastructure Manager** > **Cross Sections** tab.

A material layer applies to all cross sections.

With stringline-based roads, the new material layer can be used to add stringlines to it. Which can be either existing stringlines on other layers or library lines (for example, imported from DBX or created in Infinity).

1.	Select the stringlines or library lines that shall be assigned to another or a new Material Layer.
2.	Right-click and select **Add to Material Layer** from the context menu or select **Add to** in the **Infrastructure** > **Road Design** ribbon bar. Choose the target Material Layer from the list.

☞ Reassigning a stringline to a different material layer does only work for stringline-based roads.

3.8.6 Material Surface

Material Surface

A material surface can be created for an existing material layer in a project.

☞ The 2.5D meshing method is used to create the material surface.

To create a material surface:

1. Select a material layer from the Navigator or the Inspector.
2. Select **Material Surface** from ribbon bar.

Alternatively:

1. Select a material layer.
2. Right-click to open the context menu and select **Material Surface**.

Material surface method:

It is possible to select one out of three pre-defined methods for material surface generation.

The method can be selected in the **Info & Settings** > **Infrastructure** tab.

Each method adjusts the deflection angle parameters used to reproduce the material surface with respect to stringline shapes and angles between neighbouring stringlines. Use this setting to improve the quality of the material surface output.

Settings:

- Method 1
- Method 2
- Method 3

3.8.7 Inspect Check Road and Stake Road Application Results

Inspect Check Road and Stake Road Application Results

To check and document stakeout results import the stake road job and/or the check road job into Infinity.

The objective is to check the stakeout results.

The differences between design coordinates and stakeout results can be:

- Seen in the graphical view.
- Checked in the **Inspector** > **Features** tab or in the **Infrastructure** tab under **Checks and Stakes**.
- Saved in a report.

See also:

Staked Points in Infinity

Reports

3.8.8 Infrastructure Manager

3.8.8.1 Overview

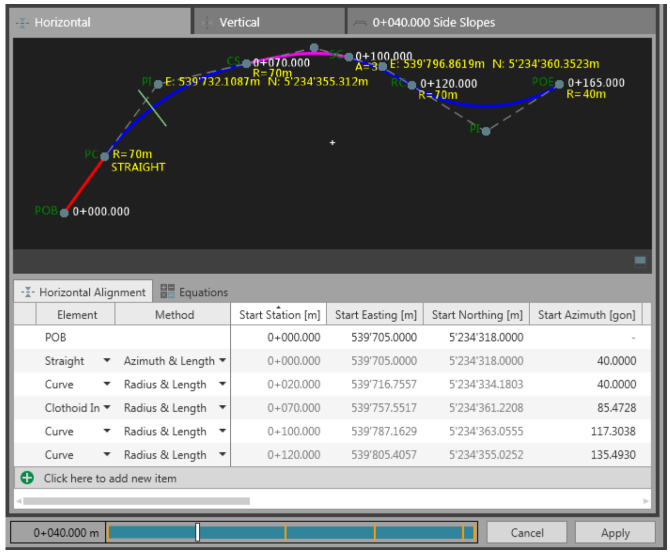
Infrastructure Manager

The infrastructure manager is the dedicated tool to view, create and edit the individual components of an alignment or a road object.

It opens up automatically when you create a new alignment or when you choose to add a new cross section template.

To edit existing elements it can also be invoked from the navigator or the inspector by selecting **Edit** from the context menu.

By default the infrastructure manager opens up as a new tab next to the inspector.



The chainage bar represents the whole horizontal alignment from its start to the end. The bars show you where cross sections are assigned.

- When the numbering of nodes is different between cross sections, then these areas of the alignment are transition areas and marked as such in the chainage bar.
- When cross section assignments are missing or there are gaps in the road, then these areas are without any marking.

You can shift the slider to any position in the road or enter a chainage value to make the slider jump. Both Alignment views (horizontal and vertical) as well as the cross section view follow your movements.

On the bars the cross section view show you the details of assigned cross sections.

3.8.8.2 Alignments

Alignments

Alignments are defined in the horizontal and/or the vertical profile tabs of the infrastructure manager.

In the horizontal tab, you can also define station equations.

How to define a horizontal alignment?

Each horizontal alignment starts at the Point Of Beginning (POB) and is defined with a sequence of geometry elements.

Existing elements can be edited. New elements can be added above or below.

1. Select an element and select **Add Above/Below** in the Road Design ribbon bar or select **Add Above/Below** from the context menu.
   By default straight elements are added.
2. Edit the element properties. Infinity guides you through which definitions are mandatory response or optional.
   The graphical view follows your changes.
3. Select **Apply** to save your changes to the alignment.

To enter station equations switch to the equations tab.

1. Select **Add Below** in the Road Design ribbon bar and enter the Back and Ahead values in the Inspector view.
2. Select **Apply**.

In the graphical view purple lines (perpendicular to the alignment) indicate in which places along the alignment station equations are defined.

For each cross section assignment, its modified station value is listed in the inspector and in the property grid.

How to define a vertical alignment?

Each vertical alignment starts at the Vertical Point Of Beginning (VPOB) and is defined with a sequence of geometry elements.

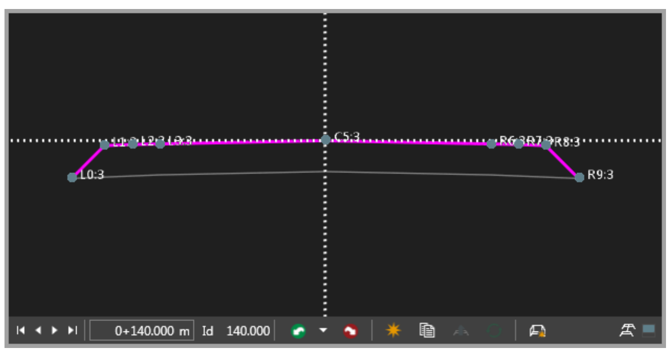
The procedure of adding elements is the same as for horizontal alignments (see previous information).

3.8.8.3 Cross Section Assignments

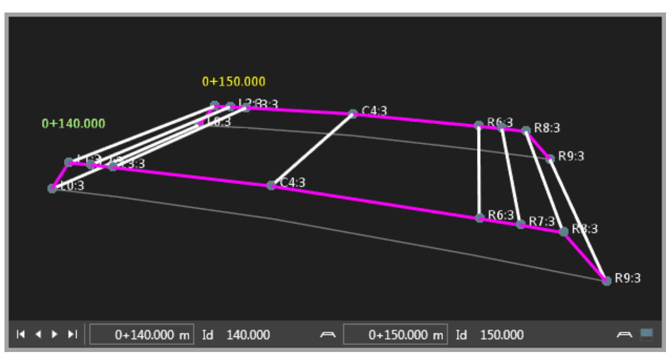
Cross Section Assignments

In the cross sections tab of the infrastructure manager, you can switch between editing cross sections and editing connections.

Edit cross section view:



Edit connections view:



Each cross section is shown as a vertical cut through the road layers at a specific station/chainage.

The inspector view has as many tabs as material layers are defined. The layer of the active tab is colour indicated in the graphical representation of the cross section.

Each of the nodes that constitute a cross section is defined with offsets to the left and the right of the centreline. The centreline is available on all layers.

To create a new cross section assignment:

1.	Shift the slider to the position in the chainage bar where a cross section shall be assigned to or enter the chainage value.
2.	Select **New Template** to create a new cross section template or select **Add to** and select a template from the list.
3.	Select **Apply** in the Infrastructure Manager.

The template is assigned as a cross section to the selected chainage and assume as its ID the current chainage value. Existing cross sections in the position are overwritten.

In the navigator and the road inspector the cross section assignment is added to the list.

☞ Assignments can be deleted. The template remains.
Changes you have made to an assigned cross section are lost when the template has not been updated before.

To edit an assigned cross section:

1.	Shift the slider upon a cross section in the chainage bar or enter its chainage value.
2.	Edit the offsets and/or side slopes of single nodes or add new nodes in the Inspector view. The graphical view follows your changes. To add new nodes right-click and select **Add Above** or **Add Below** from the context menu. ☞ If you design a cross section from scratch and you know that left and right side shall be identical, then you can create the right or the left side and press the Mirror button to automatically create the nodes on the other side. It only works when one side is empty.
3.	If a template has been used, optionally update your changes to the template for further use. ☞ Assignments that have come in with your road data on import do not have underlying templates. Copy them to a new template when you want to save your changes. ☞ Assignments using a modified template are not automatically updated. Reassign the updated template when necessary.
4.	Select **Apply**.

To create a copy of an assigned cross section:

1.	Shift the slider upon the cross section that shall be copied or enter its chainage value.

2. Select **Copy to New Cross Section Template**.
Next to the Property Grid the New Cross Section Template tab opens up.

3. Shift the slider to the position in the chainage bar where the cross section shall be copied to or enter the chainage value.

4. Select **Create** in the New Cross Section Template tab.
The template ID can be modified.

5. Select **Apply** in the Infrastructure Manager.

The copy of the previously selected assignment is created and assigned to the selected chainage. Existing cross sections in the position are overwritten.

The template is available for future use.

To extract a cross section to a new cross section template:

Cross sections can be extracted from a transition area by interpolating all its nodes from the stringlines between the two existing cross sections.

1. Shift the slider to the position in the chainage bar where the new cross section shall be interpolated or enter the chainage value.

2. Select **Extract to New Cross Section Template**.
Next to the Property Grid the New Cross Section Template tab opens up.

3. Select **Create** in the New Cross Section Template tab.
The new cross section is interpolated at the selected chainage.

4. Select **Apply** in the Infrastructure Manager to assign the new cross section.
The template is available for future use.

How to edit connections?

To fix problems found within road design data using cross sections:

1. Switch to the Edit Connections view.

2. Shift the slider in the chainage bar to the cross section for which you want to edit connections.
The graphical view shows the selected cross section and the following one.
The Inspector splits to show the details of both cross sections.

3. In the Inspector select the elements to be connected in both cross section profiles and then **Link** from the context menu or from the **Infrastructure** > **Road Design** ribbon bar.
To disconnect elements select them both and then **Unlink**.

The stringline is recalculated and redrawn in the graphical view.

3.9 Adjustments

3.9.1 Overview

Adjustments

From the observations you have made in the field, the next step is to compute an end result, the coordinates. When redundant observations are available, as should be the case, choose a strategy to get a unique and optimal solution. In geodesy, this strategy is usually a least squares adjustment, which is based on the following criterion: The sum of the squares of the observational residuals must be minimised. After carrying out a least squares adjustment, you know that based on the available observations, you have achieved the best possible solution.

But after you have found the best possible solution, it is further important to be able to assess the quality of this solution to verify whether your network meets the requirements. The quality of a network can be assessed in terms of precision and reliability, which can be quantified by setting the adjustment parameters.

Finally, quality control has to include statistical testing to clear the result of possible outliers. The effectiveness of testing depends on the reliability of the network. The more reliable a network is, the higher the probability that outliers are detected by the testing.

To summarise the relationship between least squares adjustment, precision and reliability and statistical testing we can say that:

- Based on the available data, the least squares adjustment produces the best possible result.
- Statistical tests check the result to make it error-free.
  See also:
  Statistical Testing
- The precision and reliability settings quantify the quality of the result.
  See also:
  General Adjustment Settings

In Infinity, the adjustment computation is performed using the MOVE3 adjustment kernel, licensed to Leica Geosystems AG by Grontmij Geo Informatie, bv, Rosendaal, The Netherlands. For detailed information, see https://move3software.com/.

See also:

The tutorial **"How to adjust a GNSS+Level network"** https://leica-geosystems.com/-/media/94056ba161ce4993b46bd2848ad58902.ashx

The tutorial **"Advanced Network Adjustment"** https://leica-geosystems.com/-/media/02614143f73640338df088b7f7123ef0.ashx

The tutorial **"Advanced Adjustment Concepts"** https://leica-geosystems.com/-/media/80026c2d4e22481fb05a0df1fd2224f8.ashx

☞ The tutorial data can be downloaded in the Localisation Tool.

Prepare the data for adjustment:

In Infinity there are four basic steps to performing an adjustment:

1.	Define the control points - Fixed in 3D, 2D, 1D or not fixed.
2.	Select the observations to be considered in the adjustment - TPS, GNSS and level observations.
3.	Run the adjustment and evaluate the results, even comparing several adjustment runs.
4.	Store the results that satisfy the adjustment criteria.

Run full adjustment:

To run a network adjustment:

Select the option in the Adjustments ribbon bar to run an adjustment.

Select from the drop-down menu whether you want to adjust **3D** (position and height), **2D** (position only) or **1D** (height only).

☞ The last selected method is remembered.

Depending on the chosen dimension, the following types of data is adjusted:

	TPS	GNSS	Level
3D	Direction, Zenith Angle and Slope Dist.	DX, DY, DZ	---
2D	Direction and Horizontal Dist.	DX, DY, DZ	---
1D	Trig. Height Diff.	---	Height Diff.

Adjustment Method	Description
3D	Data is adjusted in all available dimensions. If only 2D or 1D data is available the 2D and 1D observations are adjusted instead.
2D	If your network has GNSS baselines these are always adjusted in 3D even if the dimension is set to 2D. TPS slope distances and zenith angles are reduced to horizontal distances. Slope distances without zenith angles are ignored. This adjustment is stored with the adjusted least squares 2D point role.
1D	If you choose 1D the GNSS baselines are excluded from the adjustment. TPS slope distances and zenith angles are reduced to trigonometric height differences. TPS directions are ignored. This adjustment is stored with the adjusted least squares 1D point role.

The adjustment result is available in the Adjustment Results view of the Inspector and can be stored if it meets your needs for accuracy.

The results of all adjustment runs are kept in the list until you select them and delete them. To delete a result from the list select it and select the **Delete** option in the top left corner of the main Infinity window.

Network Reports

For each result in the Inspector, you can create a report to compare individual adjustment runs.

Select the **Reports** option in the ribbon bar and select **Network Adjustment Report** from the drop-down menu.

The report opens in a separate window and can be saved as a PDF or HTML file. Both types of file are added to the Archive section of the Navigator under Reports and can be opened from there again.

Store Result

In the General Adjustment Settings, you can choose to automatically store a result after the network computation is finished. If you have deselected this setting, you can store the result manually. A reason for doing that, can be to first inspect a result before storing it.

See also:

General Adjustment Settings

Inspecting Adjustments

1. Go to the **Adjustment Results** view of the Inspector and select the result which you want to store from the list.
2. Select the **Store Result** option in the ribbon bar to store the adjustment result manually.

After saving a result, the point role adjusted least squares are assigned to all adjusted points. You can select this point role for each point in the Navigator and inspect the properties in the Property Grid.

☞ Only one result can be selected to be stored. You cannot store multiple Results.

When you want to store another result, select it from the list and select the **Store Result** option again.

When you want to remove the adjusted least squares point roles from the project, select the **Remove Adjustment** option in the ribbon bar.

What is stored to the project:

A useful feature of Infinity is allowing you to run several adjustments to compare results and in general decide which adjustment satisfies the adjustment criteria.

☞ It is important to recognise the adjustment results are stored to the project and merged with the existing data.
The adjusted coordinate values are merged to the library points and the point role reflects the value that was stored.
It is also possible to merge the coordinate values from two adjustment runs and to combine the 2D and 1D runs.

Adjusted least squares point roles:

After storing an adjustment result, the following table explains the point role that is stored to the points project data:

Point Role	Storing Adjustment Dimension
Adjusted Least Squares 3D	Storing 3D creates a point role that indicates all three coordinate values are from the adjustment. If an existing adjusted point role exists, then it is replaced with the adjustment values of the most recent store.
Adjusted Least Squares 2D	Storing 2D creates a point role that indicates the position coordinate values are from the adjustment. If an existing adjusted point role exists, then this store replaces the position coordinate values. The height value is not replaced.
Adjusted Least Squares 1D	Storing 1D creates a point role that indicates the height values are from the adjustment. If an existing adjusted point role exists, then this store replaces the height coordinate value only. The position coordinate values are not replaced.
Adjusted Least Squares 2D + 1D	When storing a 2D and then a 1D adjustment the height coordinate value is merged with the position coordinate values to provide a 2D + 1D adjusted coordinate. Run and store a 2D adjustment, the adjusted least squares 2D point role is available. Run and store a 1D adjustment, the adjustment respects the 2D value and updates the height value.
Adjusted Least Squares 1D + 2D	When first storing a 1D and then a 2D adjustment the position coordinate values are merged with the height coordinate value to provide a 1D + 2D adjusted coordinate. Run and store a 1D adjustment, the adjusted least squares 1D point role is available. Run and store a 2D adjustment, the adjustment respects the 1D value and updates only the position value.

Remove Adjustment:

To remove a stored adjustment and with it the adjusted point roles:

Select the **Remove Adjustment** option from the ribbon bar to remove the stored result.

The status of the stored adjustment run is set back to not stored and the point roles adjusted least squares are removed from all adjusted points.

But the adjustments run itself is kept in the Adjustment Results section of the Inspector for future reference and could be saved again if necessary.

To delete an adjustment result from the list, select the result and select the **Delete** option in the top left corner of the main Infinity window.

3.9.2 Free and Constrained Adjustments

Free and Constrained Adjustments

The adjustment of a network is usually subdivided into two separate steps or phases:

1. Computation of a free network adjustment.
2. Computation of a constrained adjustment.

This approach is intended to separate the statistical testing of observations and known stations.

See also:

Statistical Testing

Free adjustments:

A free network is a network in which the geometrical layout is determined by the observations only. The position, scale and orientation of the network are fixed by a minimum number of constraints. The control coordinates of known stations do not impose any extra constraints on the adjustment solution, yet. In a free network adjustment the emphasis is on the quality control of the observations rather than on the computation of coordinates. Selecting other stations to fix the position, scale and orientation changes the coordinates but not the results of statistical testing.

Constrained adjustments:

Having eliminated possible outliers in the observations by running a free adjustment, the network can be connected to the known stations. This does impose extra constraints on the solution. Now the emphasis is on the analysis of the known stations and on the computation of the final coordinates.

There are two types of constrained adjustments:

- Absolutely constrained.
- Weighted constrained.

The difference between these two types lies in the way coordinates are computed:

- In an absolutely constrained adjustment the coordinates of the known stations are kept at their original value, that is, they do not get a least squares correction. An absolutely constrained adjustment is sometimes called a pseudo least squares adjustment.
- In a weighted constrained adjustment however, the known stations do get corrections.

Your decision whether to compute the adjustment absolutely or weighted constrained leaves the testing results unchanged.

3.9.3 Adjustment Settings

3.9.3.1 General Adjustment Settings

General Adjustment Settings

☞ The parameters for adjustment computations can either be set in the File tab under Info & Settings or from inside the Adjustments ribbon bar.
Changes are stored with the project and applied to all future adjustment runs.

To set the adjustment parameters back to the default values:

Go to the **File** tab and select the **Defaults** option in the **Adjustment Parameters** section under **Info & Settings**.

Computation

Store results immediately after computing:

Decide whether you want to store the results of an adjustment immediately and automatically after computing, or whether you want to store the results manually later on.

By default the option ☑ Store results immediately after computing is active. When you change this setting, it has an immediate effect and also applies to projects which are already open.

When you de-activate this setting, you can store the results manually by selecting **Store Results** from the **Adjustments** ribbon bar.

Controls

Select how control coordinates shall be introduced into the adjustment computation.

- **Constrained**
  Two or more control points are kept fixed.
- **Weighted**
  Control points are kept relatively fixed and are allowed to move according to their standard deviations.

Iterations

Max. Iterations

Set the maximum number of iterations or computation runs to perform to try and reach the iteration criteria (see the following). The maximum number of iterations defines an upper limit and prevents the computation from running into an endless loop if there are problematic observations taking part in the adjustment which prevents the iteration criteria from ever being met. With GNSS observations, one iteration is normally sufficient to meet the iteration criteria. In this case, the adjustment automatically stops the computation even if maximum three iterations have been set.

Iteration Criteria

The iteration criteria is the maximum allowed size of corrections to the adjusted coordinates which must be reached before the iteration stops. The iteration criterium is subject to the maximum number of iterations.

Visualisation Exaggeration Factors

Absolute and Relative Error Ellipses/Reliability

Set factors in order to enlarge the visualisation of error ellipses and reliability boxes such that the error ellipses and reliability boxes can properly be seen in the graphical view.

Threshold

Set a threshold value above which error ellipses and reliability boxes shall be shown in the graphical view. Errors or reliability values below are not displayed independent of the given exaggeration factor.

Confidence Levels

Here you can set the confidence levels (that is the standard deviations) for Heights - 1D and absolute and relative Error Ellipses - 2D. The given values are stored in the adjustment results and shown in the adjustment report. In the graphical view absolute and relative error ellipses are shown.

The axes of the absolute and relative error ellipses are multiplied by a scale factor that is derived from the confidence levels as set by you for the stored adjustment result. If you use other than the standard confidence levels then confidence ellipses/ellipsoids are shown instead of error ellipses/ellipsoids.

The standard deviation for heights is treated in the same way.

The system default values are the standard one-sigma values, that are:

- For Heights - 1D: 68.3%.
- For Error Ellipses - 2D: 39.4%.

☞ Changing the confidence levels does not have any effect on the statistical tests.

TPS Accuracy Information

Here you can define the standard deviations and the centering and height errors to be taken into account for the adjustment of TPS data. Altogether the values defined here are used to define an overall accuracy information to be fed into the adjustment.

Standard Deviations

Here you can define the default standard deviations that shall be applied to all TPS observations when computing an adjustment.

Source for Standard Deviations

- Select **Individual** to make the adjustment apply the accuracies which are stored with the observations. These are normally the standard deviations pre-defined for all measurements on the instrument.
- Select **Use Defaults** to make the adjustment apply the default settings to be defined below for all observations. A ppm value can be entered to account for the relative length (the distance) of an observation.

Centering/Height Errors

Here you can define the default accuracy that shall be applied to the two end points (setup and target) of a measurement.

The centring error defines the predicted error that could have been made when centring the instrument/target on a point. The height error defines the predicted error when measuring the instrument/target height above a point.

Source for Centering/Height Errors

- Select **Individual** to make the adjustment apply the accuracies which are assigned to the two end points (setup and target) individually.
- Select **Use Defaults** to make the adjustment apply the default settings to be defined below for setup and target.

GNSS Accuracy Information

Here you can define the standard deviations, the centering and height errors and the sigma a priori to be taken into account for the adjustment of GNSS data. Altogether the values defined here are used to define an overall accuracy information to be fed into the adjustment.

Standard Deviations

Here you can define the default standard deviations that shall be applied to all GNSS observations when computing an adjustment.

Source for Standard Deviations

- Select **Individual** to make the adjustment apply the accuracies which are stored with the observations.
- Select **Use Defaults** to make the adjustment apply the default settings to be defined below for all observations. A ppm value can be entered to account for the relative length of a baseline.

Centering/Height Errors

Here you can define the default accuracy that shall be applied to the two end points (reference and rover) of a measurement.

The Centring error defines the predicted error that could have been made when centring the reference/rover on a point. The height error defines the predicted error when measuring the reference/rover height above a point.

Source for Centering/Height Errors

- Select **Individual** to make the adjustment apply the accuracies which are assigned to the two end points (reference and rover) individually.
- Select **Use Defaults** to make the adjustment apply the default settings to be defined below for reference and rover.

Sigma a Priori

Here you can enter a value to compensate for too optimistic GNSS observations. Often, observations coming from GNSS post-processing programs are too optimistic in their accuracy information. This in itself does not matter when adjusting pure GNSS observations but it becomes important when combining GNSS and TPS observations.

Level Accuracy Information

Here you can define the standard deviations to be taken into account for the adjustment of level data. Altogether the values defined here are used to define an overall accuracy information to be fed into the adjustment.

Source for Standard Deviations

- Select **Individual** to make the adjustment apply the accuracies which are stored with the observations. These are normally the standard deviations pre-defined for all measurements on the instrument.
- Select **Use Defaults** to make the adjustment apply the default settings to be defined below for all observations. A ppm value can be entered to account for the relative length (the distance) of an observation.

3.9.3.2 Test Criteria

Test Criteria

☞ The parameters for adjustment computations can either be set in the File tab under Info & Settings or from inside the Adjustments ribbon bar.
Changes are stored with the project and applied to all future adjustment runs.

To set the adjustment parameters back to the default values:

Go to the **File** tab and select the **Defaults** option in the **Adjustment Parameters** section under **Info & Settings**.

Level of Significance (α0)

The Alpha0 criterium determines the probability of rejecting a good observation. By default the level of significance is set to 5% as this is regarded as a good compromise. Setting the Alpha0 value too low may result in a bad observation being accepted.

Power of Test (1 - β)

The 1-Beta criterium determines the power of the test, that is, the probability of accepting a bad observation. By default the power of testing is set to 80% as this is regarded as a good compromise. Setting the Beta value too high may result in good observations being rejected.

☞ The Alpha0 and Beta settings are subjective and should be determined by an experienced surveyor who carried out the field work. If you feel unsure about what Alpha0 and Beta should be set to, accept the suggested default values. For a theoretical background and further information on both these values read the topic on the W-Test.

Sigma a posteriori

This is a global setting that compensates for the uncertainty of the a priori value. It affects the estimated precision of the adjusted coordinates. You can, if required, only apply the sigma a posteriori if the adjusted coordinates fail the so-called F-Test. The F-Test is a test that is applied to the sigma a priori and the sigma a posteriori. If both are statistically different the result of the F-Test indicates that the stochastic values awarded to the observations a priori are incorrect (assuming outliers have been removed!). Applying the sigma a posteriori then compensates for this problem.

See also:

W-Test

F-Test

3.9.3.3 Advanced Terrestrial Parameters

Advanced Terrestrial Parameters

☞ The parameters for adjustment computations can either be set in the File tab under Info & Settings or from inside the Adjustments ribbon bar.
Changes are stored with the project and applied to all future adjustment runs.

To set the adjustment parameters back to the default values:

Go to the **File** tab and select the **Defaults** option in the **Adjustment Parameters** section under **Info & Settings**.

Use reduced observations:

When calculating a network adjustment by default all active observations are used. Select ☑ this option, if you want to only use the reduced observations for all sets of angles applications stored in a project. The single observations contained in the sets of angles are then not taken into account.

The standard deviations for the reduced observations are taken from the sets of angles calculation, that is, the mean error of the averaged observations.

Use vertical refraction coefficient:

- Select **Ignore**, if you want to apply a vertical refraction coefficient to zenith angles individually before feeding the data into the adjustment.
- Select **Compute**, if you want to estimate the vertical refraction coefficient. This is used in projects which have many zenith angle measurements and thus a good estimate can be obtained.
- Select **Use**, if you want to apply the above set default value to all zenith angles.

☞ As the vertical refraction coefficient is applied to all vertical angle measurements in the network, it should only be used in networks that cover relatively small areas.

Vertical Refraction Coefficient

The vertical refraction coefficient accounts for the influence of refraction on zenith angles. The default is set to a typical value of 0.13.

Use scale factor correction:

- Select **Ignore**, if you want to apply a scale factor to distances individually before feeding the data into the adjustment.
- Select **Compute**, if you want to estimate the scale factor. This is used in projects which have many distance measurements and thus a good estimate can be obtained.
- Select **Use**, if you want to apply the above set default value to all distances.

Scale Factor Correction

Here you can define the scale factor, that shall be applied to distance measurements as an extra factor used to compensate for atmospheric conditions.

3.9.3.4 Coordinate System Settings

Coordinate System Settings

☞ The parameters for adjustment computations can either be set in the **File** tab under Info and Settings or from inside the **Adjustment** ribbon bar.
Changes are stored with the project and applied to all future adjustment runs.
To set the adjustment parameters back to the default values:

Go to the **File** tab and select the **Defaults** option in the **Adjustment Parameters** section under **Info & Settings**.

Coordinate System

- Select **WGS84** if you want to adjust your observations in the WGS84 system.
- Select **Local Geodetic** if you have a local coordinate system attached to your project and you want to adjust the observations in the local system of your **Control** points.

☞ The transformation used with your local coordinate system has to be either a **Classic 3D** or **None** for this option to become active.
Read more in the topic "Status Bar" on how to attach a local coordinate system to a project.

- Select **Local Grid** if you want to adjust pure TPS observations in a local grid system. In this case, no information on the local ellipsoid is necessary.

☞ If your project has GNSS and TPS observations and you select this option then only the TPS observations are adjusted. GNSS observations are ignored. Select **Local Geodetic** if you want to compute a **combined** adjustment.

Height Mode

Depending on the selected height mode, the adjusted coordinates have ellipsoidal or orthometric heights:

- Select **Ellipsoidal** if you want to adjust your observations with ellipsoidal heights.
- Select **Orthometric** if you want to adjust your observations with orthometric heights.

If for your **Control** points orthometric heights are available and a geoid model is included in the coordinate system to be used with the project then:

Run the adjustment with setting the height mode to **ellipsoidal**.

After the adjustment, the geoid separations are applied again to get the orthometric heights of the adjusted coordinates.

Parameters

In order transform observations **four** parameters are required: three rotations about each axis and a scale factor. These parameters can either be predefined and used as such in the adjustment or you can make the adjustment compute the parameters.

- Select **Compute** if you want the adjustment to compute the parameters independent of the parameters given by the attached coordinate system.
- Select **Use** if you want your observations to be adjusted using the parameters as defined here. This does not have any influence on the parameters given by the attached coordinate system.

☞ In order to **compute** transformation parameters you should keep at least three of your points fixed. Otherwise the network computation creates error messages. It is also important to note that the computed transformation parameters are only valid for transforming observations - they cannot be used for transforming coordinates.

3.9.4 Inspecting Adjustments

3.9.4.1 Overview

Inspecting Adjustments

With Infinity you are flexible to run adjustments under various pre-conditions and keep the results next to each other. You can inspect and select control points to be fixed, observations not to be used, adapt the settings and compare and judge afterwards which result fits your needs best before you finally save one solution to your project.

To inspect adjustments:

Go to the Inspector and open the **Adjustment** tab.

Define coordinate constraints:

The **Define Coordinate Constraints** view, lists all control points in your project. Control points are needed when you want to adjust a constrained or weighted constrained network.

If you want a control point to take part in the adjustment:

Select the ☑ **Fix 2D** checkbox, if you want its position coordinates to take part in the adjustment.

Select the ☑ **Fix 1D** checkbox, if you want only the height coordinate to take part.

Select ☑ **both** checkboxes, if you want all coordinate components to take part.

The role of all control points selected here, changes to Control Fixed 3D (or 2D or 1D) to indicate that these points are considered for taking part in the adjustment.

See also:

Objects, Point Roles and Symbols

If you want to compute a weighted constrained network standard deviations have to be available for each control point taking part in the adjustment.

☞ Before you start the adjustment, the control coordinates can be edited in the property grid if necessary.

☞ You can import control coordinates, for example, from an ASCII file, or you can enter them manually by selecting a point and selecting **Create Control point** next to the existing point role in the Feature section of the Property Grid.

Select Observations

The select observations view lists all observations in your project. By default all observations are selected to be used in the adjustment computation.

If you want to exclude observations:

- De-select the **Use** checkbox for a single observation.
  Or
- Multi-select a set of observations, right-click into the selection and select **Adjustment** > **Don't Use** from the context menu.

A reason for excluding an observation could be that it has been identified as an outlier in a previous adjustment run.

See also:

W-Test

Possible reasons for outliers could be mistakes in elements like the target height or type, offsets or the atmospheric or geometric corrections.

All these elements as well as the coding information can be edited in the property grid before you start an adjustment run.

See also:

Observation Properties

Computed Loops Results

The computed loops results view lists all loop computation results. To inspect the results open the computed loops report.

For detailed information, see:

Loops Computation

Pre-Analysis Results

The pre-analysis results view lists all pre-analysis results. To inspect the results open the pre-analysis report.

See also:

Pre-Analysis

Adjustment Results

The adjustment results view lists all adjustments that have been computed within the current project.

On the top level for each adjustment run you can see:

- The Adjustment Type, that has been computed, that is, inner constrained, constrained or weighted constrained.
- The Dimension, that is, the position only or position and height or height only.
- The Confidence Levels, that have been defined for heights and error ellipses.
- The Degrees of Freedom.
- The Critical Value for the F-Test, as computed on the basis of the currently given level of significance plus the F-Test result.

To inspect the resulting adjusted coordinates and observations:

Drill into a result by selecting the arrow next to it.

Under coordinates you can inspect the adjusted coordinates together with:

- The values by which they have been corrected in their components.
- The resulting standard deviations for all points that have not been kept fixed or only weighted fixed.
- The values defining the error ellipses.
- The external reliability in each coordinate direction.

Under observations you can inspect the adjusted observations separately for TPS and GNSS together with:

- The residuals that have been amended in each observation component.
- The resulting standard deviations for each observation component.
- The W-Test results for each observation component.

To manually save the result that meets the demanded accuracy best:

Select it and select **Store result** from the ribbon bar to save it.

☞ On how to remove a stored result from the database and on how to remove an adjustment run from the list of results, refer to the overview topic in adjustments.

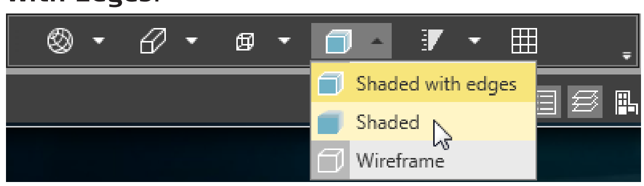
3.9.4.2 Error Ellipses/Ellipsoids

Error Ellipses/Ellipsoids

A surveying network can be checked in terms of both, accuracy and precision. The most common approach to check a network precision is the calculation of the absolute and relative error ellipses.

Infinity can display both kinds of ellipses/ellipsoids in the graphical view as well as show the respective information in the inspector and the network adjustment report.

To show the error ellipses/ellipsoids in the graphical view:

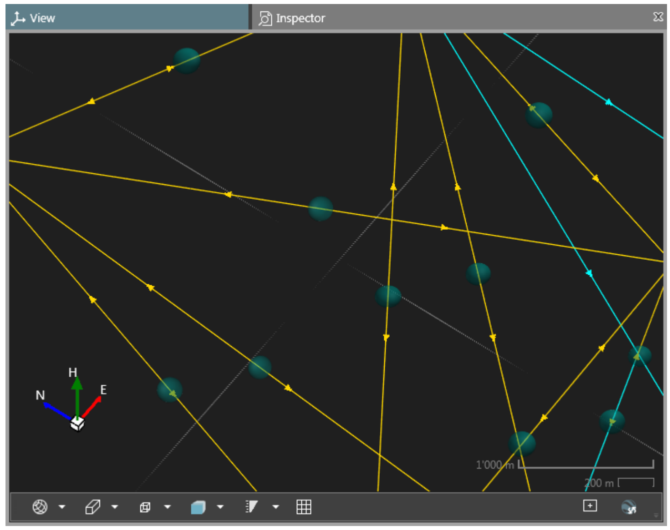
1.	Run a network adjustment in 2D or 3D.
2.	Store the result.
3.	Activate (switch on:) the **Absolute Error Ellipses** and **Relative Error Ellipses** layers in the Layer Manager.
4.	Switch the graphical view to display objects as **Shaded** or **Shaded with Edges**. 

For further information on how to run adjustments and store results, see Adjustments.

☞ If the graphical view is in 3D mode, then error ellipsoids are shown (instead of 2D error ellipses).
On how to switch between standard views, see Graphical View.

☞ Error ellipses/ellipsoids are only shown if you run the adjustment using standard confidence levels, that is 68.3% for heights - 1D and 39.4% for absolute and relative error ellipses - 2D.
If you choose other confidence levels, then the graphical view shows confidence ellipses/ellipsoids instead.

Illustration:



☞ If errors are too small, go to **Settings** > **General** and enlarge the **Visualisation Exaggeration Factors**. Else you might not be able to see the reliability boxes.
If errors are below the given threshold value they are not shown. Set the threshold to a value smaller than the errors.

See also:

General Adjustment Settings

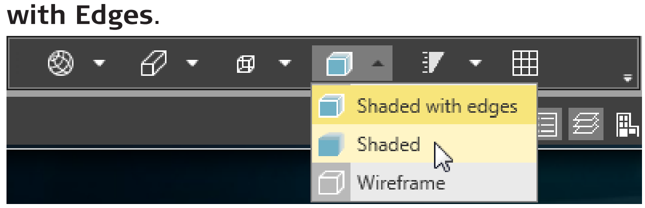
3.9.4.3 Reliability Boxes

Reliability Boxes

External reliability can be expressed in terms of reliability boxes. For each station in a project a reliability box is computed as a means to express the influence of a possible error in the observations or of known coordinates on the adjusted coordinates. In constrained adjustments the selection of control points that shall remain fixed during adjustment has a direct influence on the reliability boxes. The box represents the area that the adjusted station is in (with a probability of 80%) if all observations and known coordinates of the network are accepted by the statistical tests. Depending on the possible error or known coordinate with the largest influence on the adjusted coordinate, the size of the box is computed according to the size of the minimum detectable bias.

To show the reliability boxes in the graphical view:

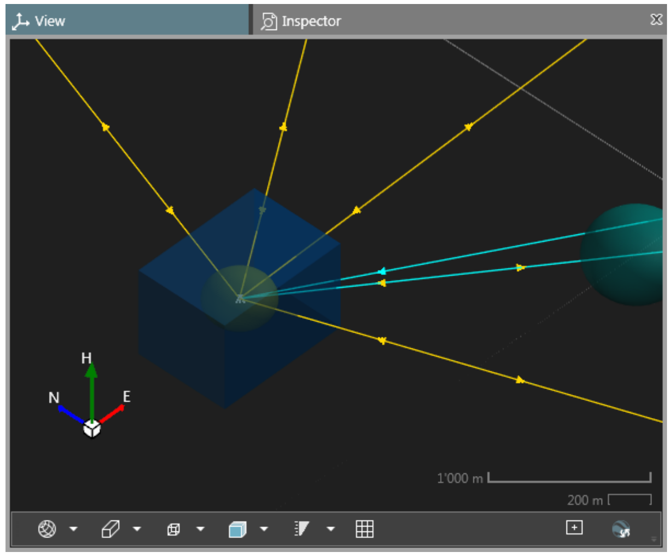
1. Run a network adjustment in 2D or 3D.
2. Store the result.
3. Activate (switch on:) the **Reliability** layers in the Layer Manager.
4. Switch the graphical view to display objects as **Shaded** or **Shaded with Edges**.



For further information on how to run adjustments and store results, see Adjustments.

☞ If the graphical view is in 2D mode, then squares are shown (instead of 3D boxes).
On how to switch between standard views, see Graphical View.

Illustration:



☞ If errors are too small, go to **Settings** > **General** and enlarge the **Visualisation Exaggeration Factors**. Else you might not be able to see the reliability boxes.
If errors are below the given threshold value they are not be shown. Set the threshold to a value smaller than the errors.

See also:

General Adjustment Settings

3.9.5 Tools

3.9.5.1 Loops Computation

Loops Computation

The compute loops functionality is one of the pre-analysis tools included in the adjustment module. It is used for automatic computation of network loops and loop misclosures.

A loops computation is always applied to the unadjusted network as a means of checking observations before running the adjustment. It detects all loops with a minimum number of sides (triangles).

☞ Loops that could be formed by combining different types are not considered.

As concerns loop type definitions, see the following:

Loop Types

Loop Type	Description
GNSS Loops	The three sums of all coordinate differences DX, DY and DZ yield closing errors in X, Y and Z.

Loop Type	Description
Direction and Distance Loops	The sum of the angles in the loop should be a multiple of 200 gon or 180 deg. The remainder is the angular closing error. The closing errors in X (local Easting) and Y (local Northing) are computed in a local XY system, with the positive Y axis being parallel to the first side of the loop, and the X axis being perpendicular to it. ☞ If the two directions of one angle in a loop are not from the same setup, then the angular closing error cannot be computed. The closing errors in X and Y however can still be computed by starting at the point with the missing angle. If two or more angles are missing no closing errors can be computed.
Zenith Angle and Distance Loops	The sum of the derived trigonometric height differences also yields a closing error in height.
Height Difference Loops	The sum of all height differences equals the closing error in height.

The calculated closing errors are tested using the W-Test.

☞ The routine does not use approximate and/or known coordinates, and it does not necessarily take into account all observations.

☞ Instrument heights, scale factors and vertical refraction coefficients are accounted for.

☞ The results do not point to a specific observation that could possibly be a blunder.

Compute Loops

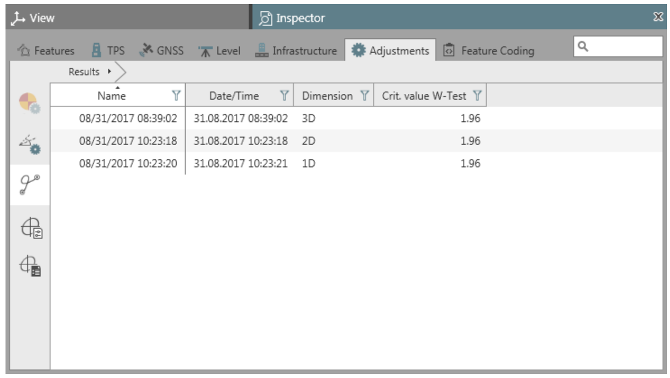
To run a loops computation:

Select **Compute Loops** from the Adjustments tab.

☞ Select from the drop-down menu whether you want to compute **3D** (Position and height), **2D** (position only) or **1D** (height only) loops. By default, loops are computed according to the main adjustment settings.

☞ Only when set to 3D are all loop types considered in the computation.
2D only considers GNSS Baseline loops and Direction and Distance loops.
1D only considers Zenith Angle and Distance loops and Height Difference loops.

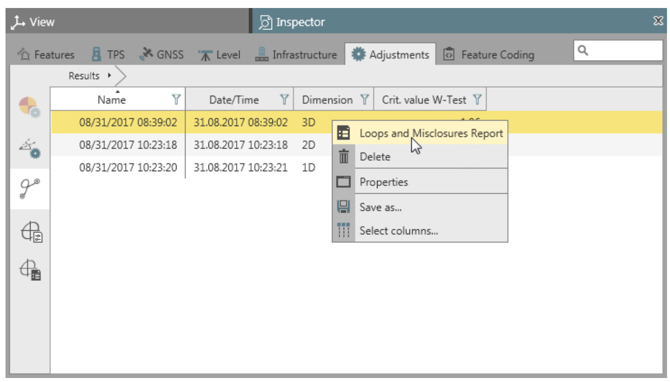
Computation results are available under **Computed Loops Results** in the Adjustments tab of the Inspector.



☞ The results of all loop computations are kept in the list until you select them and delete them. To delete a result from the list right-click upon it and select **Delete** from the context menu.

Loops and Misclosures Reports

In order to inspect results in detail select a computation run and select **Loops and Misclosures Report** from the context menu.



Loops and Misclosures Report

You can generate a report for each result listed in the Inspector in order to inspect misclosure details. Loops and closing errors are listed by loop type as described in Loops Computation.

☞ The inspector itself does not allow for any drilling-in to results of loop computations.

To generate a loops and misclosures report:

1. Select a computation result in the Inspector.
2. Select **Reports** from the Adjustment ribbon bar and select **Loops and Misclosures Report** from the drop-down menu.
   Or:
   Right-click and select **Loops and Misclosures Report** from the context menu.

☞ The report opens up in a separate window and can be saved as a PDF or HTML file. Both types of file are added to the navigator under **Archive** > **Reports** and can be opened from there again.

3.9.5.2 Pre-Analysis

Pre-Analysis

Pre-analysis is a tool that can be used:

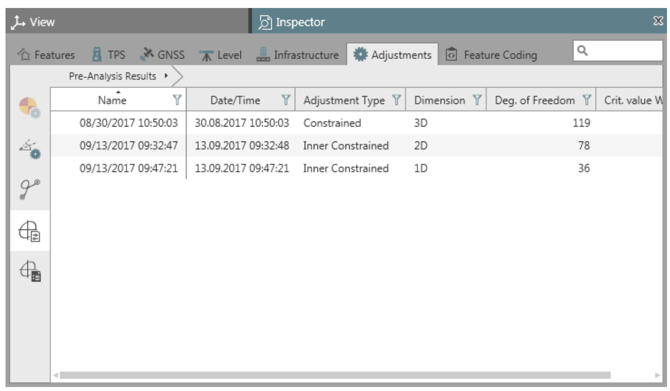
- To detect possible weaknesses in a surveying network before adjustment.
- To detect unknowns which cannot be solved.
- To check input data.

To run a pre-analysis:

Select **Run Pre-Analysis** from the Adjustments ribbon bar.

Select from the drop-down menu whether you want to compute **3D** (position and height), **2D** (position only) or **1D** (height only) loops. By default, loops are computed according to the main adjustment settings.

Computation results are available under **Pre-Analysis Results** subtab under the Adjustments tab in the Inspector.

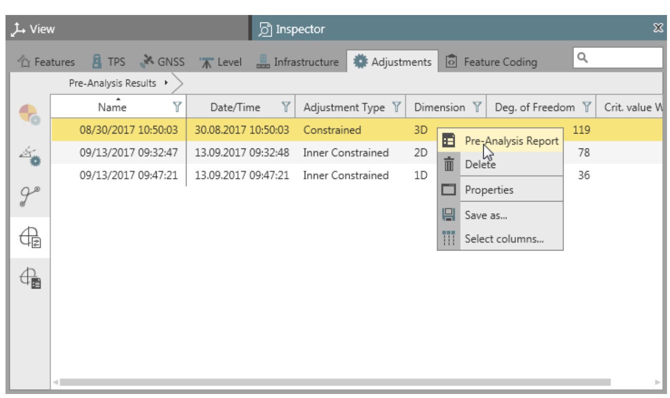


☞ The results of all pre-analysis computations are kept in the list until you select them and delete them.

To delete a result from the list right-click upon it and select **Delete** from the context menu.

Pre-Analysis Reports

In order to inspect results in detail select a computation run and open the **Pre-Analysis Report** from the context menu.



Pre-Analysis Report

You can generate a report for each result listed in the inspector in order to inspect details of a pre-analysis.

☞ The inspector itself does not allow for any drilling-in to results of a pre-analysis.

To generate a pre-analysis report:

1. Select a computation result in the Inspector
2. Select **Reports** from the Adjustments ribbon bar and select **Pre-Analysis Report** from the drop-down menu.
   Or:
   Right-click and open the **Pre-Analysis Report** from the context menu.

☞ The report opens up in a separate window and can be saved as a PDF or HTML file. Both types of file are added to the navigator under **Archive** > **Reports** and can be opened from there again.

The pre-analysis report includes the following information:

- **Configuration Defects:**
  This sections reports upon singularities that may come up when a network is adjusted. It lists unknowns that cannot be solved, either because of the network geometry or because of the type of observations or because of the coordinate system that shall be used for the adjustment.
- **Comparison of Identical Observations:**
  This section reports upon multiple observations to the same point that differ by a certain amount and may, therefore, be suspect. If there is a high probability that an error exists, the W-Test value is marked with bold red text. Other possibly suspect observations are left to your discretion.
- **Comparison of Observations and Approximate Coordinates:**
  This section reports upon observations checked against the pseudo-observations, that are derived from approximate coordinates. Observations, for which a large difference is identified, are listed.
- **Possibly Identical Observations:**
  This section reports upon observations which are suspected to be identical. You may want to investigate if observations are really separate.
- **Possibly Coinciding Stations:**
  This section reports upon stations which coincide with a distance of less than 2 metres. Such stations are suspected to be identical, though they have not been given the same point ID.

Depending on your data additional warning messages may appear after the last section of a pre-analysis report.

3.9.6 Least Squares Adjustment

3.9.6.1 Mathematical Model

Mathematical Model

Usually in a survey, the observations themselves are not the quantities, which we are aiming for. Instead, we use the observations to determine unknown parameters, for example the coordinates of stations in a network. The observations are expressed as a function of the parameters in a so-called functional or mathematical model.

In some cases, the model representing the relations between the observations and the unknown parameters is simple. The relation, for example, in a one-dimensional leveling problem between the observed height differences and the unknown heights is linear:

$$\Delta X_{ij} = h_j - h_i$$

More complicated is the case for a GPS network where the unknowns are coordinates (X,Y,Z) to be determined in a reference system different from that of the observed baselines ΔX:

$$\Delta X_{ij} = \text{function } (\alpha, \beta, \gamma, \mu, X_i, Y_i, Z_i, X_j, Y_j, Z_j)$$

With $\alpha, \beta, \gamma, \mu$ = transformation parameters

As the least squares approach requires linear equations, the complex model above must be linearised. Usually this means that a number of iterations are needed to reach a solution. Moreover, approximate values for the coordinate unknowns in the adjustment are required. Bad approximate values can lead to an increasing number of iterations or, in the worst case, to no convergence at all.

Nuisance parameters

Since coordinates are our main concern, other unknowns in the mathematical model are not always useful for us. Unknowns such as the transformation parameters mentioned above, are called additional or nuisance parameters. Typical nuisance parameters are: transformation parameters, scale factors, azimuth offsets, orientation unknowns and refraction coefficients. Some of these parameters can be kept fixed at a certain value, which makes them not be corrected in the adjustment. Whether or not to keep parameters fixed is a question which cannot easily be answered. We must always be careful to avoid over constraining as well as not to over parameterise. The introduction of refraction coefficients, for example, could result in the absorption of systematic effects which are not caused by refraction. However, ignoring the refraction, when in fact it does have an influence on the measurements, causes an equally unfavourable effect. The success of what could be referred to as tuning of the model depends largely on the expertise.

Another typical nuisance parameter is the scale factor which may be estimated for distance measurements. The aim of introducing a free scale factor in the adjustment is to overcome a possible bias in the internal scale of the measurement equipment and, in more general terms, to prevent the over constraining of the network in a free adjustment. A free scale factor shrinks or blows up the network in order to make it fit to the known stations in constrained adjustments. As a result, in some situations a free scale factor may obstruct the statistical testing of known stations: An outlier in the coordinates of a known station could remain undetected when, due to the shrink or blow up effect, the network is forced to fit to the known stations without any rejections. The outlier is absorbed by the scale factor, which then has a value distinctly different from 1.0. It is, therefore, recommended to examine the value of the scale factor after the adjustment and, in case of doubt, to rerun the adjustment with a fixed scale.

Singularity

An adjustment fails when the mathematical model, as represented by the design matrix and normal matrix, is singular.

Singularity is caused by:

- An ill-posed problem.
- An improperly formulated model.

A problem may be ill-posed because we expect too much from the observations, or because not enough observations are included. An example of an ill-posed problem is the determination of the 2D coordinates of an unknown station by a single horizontal direction from another station.

A model is improperly formulated, when too many parameters are included. In general, an improperly formulated model does not correctly represent the existing physical reality.

Additionally, the ill-conditioning of the normal matrix N could result in singularity, a problem which occurs especially with computerised solutions to least squares problems. An ill-conditioned matrix can become singular as a result of the internal accuracy limits of the computer hardware. An example of an ill-conditioned problem is the intersection of a station by two or more nearly parallel directions.

Minimum number of constraints

Apart from the problems mentioned above, an adjustment of terrestrial observations cannot be solved unless the location, orientation and scale of the network are established, that is, a datum must be defined. This is done by imposing constraints on the solution. The minimum number of constraints depends on the dimension of the network:

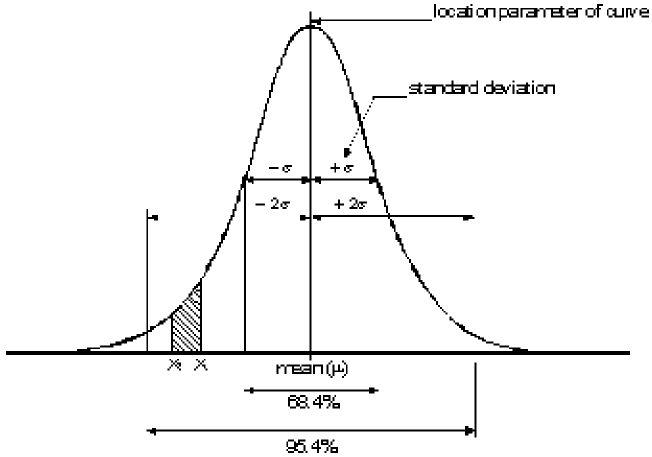
In a 3D network, there are three translations, three rotations and one scale factor. The singularity is then eliminated by fixing at least seven coordinates of three stations (Lat1, Lon1, h1, Lat2, Lon2, h2, h3).

3.9.6.2 Stochastic Model

Stochastic Model

A geodetic observation, such as a direction, distance or height difference, is a random or stochastic variable. A stochastic variable cannot be described by a single and exact value because there is an amount of uncertainty involved in the measurement process. For example, repeatedly measuring the distance between two stations results in a range of different values. This variation is accounted for by a probability distribution. This means that in addition to the mathematical model, it is necessary to formulate a second model which describes the stochastic deviations of the observations; the stochastic model.

For geodetic observations a normal probability distribution is assumed (see the following). The distribution is based on the mean μ and the standard deviation σ.



Normal Distribution

The mean μ represents the value of the mathematical expectation of the observable. The standard deviation is a measure of the dispersion or spread of the probability. The standard deviation characterises the precision of the observation. The square of σ is called the variance. By definition there is a 0.684 probability that normally distributed stochastic variables fall within a window limited by -σ and +σ. For a window limited by -2σ and +2σ this probability is 0.954. In general, the probability that a stochastic variable takes a value between x1 and x2 is equal to the area enclosed by the curve, and the x1 and x2 ordinates. This is the shaded area in the diagram above.

Correlation of observations

It is possible for two or more observations to be interdependent or correlated. This means that a deviation in one observation influences the other. The correlation between two observations x and y is mathematically expressed by the covariance σxy. The covariance is also used in the correlation coefficient, defined as:

$$\rho = \frac{\sigma_{XY}}{\sigma_X \sigma_Y}$$

The coefficient takes values between minus and plus one:

$-1 \leq \rho \leq 1$

If the observations are not interdependent it follows that ρ= 0. The vector elements (DX,DY,DZ) of a GPS baseline are an example for correlated observations. To express this correlation a 3x3 matrix is used. This symmetric matrix is a combination of standard deviations and correlation coefficients:

$$\text{Des and Adjust } \rho \text{ - matrix} = \begin{bmatrix} \sigma_{DX} & & \\ \rho_{DXDY} & \sigma_{DY} & \\ \rho_{DXDZ} & \rho_{DYDZ} & \sigma_{DZ} \end{bmatrix}$$

Sigma a priori

In essence the stochastic model consists of a choice for the probability distribution of the observations. Practically this means that for each observation a standard deviation σ is chosen. The value for s is based on knowledge about the measurement process (conditions in the field, type of instrument) and experience. The standard deviation of most observation is supposed to consist of an absolute part, and a relative part. In the relative part the dependence on the distance between station and target, which characterises the precision of most observations, is accounted for. The standard deviations are entered in the variance-covariance matrix Σb. The precision of the unknowns in the adjustment depends on the precision of the observations given in Σb, and on the propagation of this precision by the mathematical model.

3.9.7 Statistical Testing

3.9.7.1 Overview

Statistical Testing

Both the mathematical and the stochastic models are based on a set of assumptions. This set is called a statistical hypothesis. Different assumptions result in different hypotheses. Statistical testing is used to verify the hypotheses. A special set of assumptions is referred to as the null-hypothesis H_0.

This hypothesis implies that:

- Your observation does not have any gross errors (blunders).
- Your mathematical model delivers a correct description of the relations between your observations and unknown parameters.
- The chosen stochastic model for your observations appropriately describes the stochastic properties of the observations.

It is clear that there are two possible results for the testing of a hypothesis: Acceptance or rejection. A specific cut-off point or critical value decides on acceptance or rejection. The critical values establish a window of acceptance. The further beyond this window a result is, the less certain the set of assumptions is satisfied. Critical values are determined with you choosing a level of significance α. The probability that the critical value is exceeded, although the set of assumptions is valid, is equal to α. In other words, α is the probability of an incorrect rejection. Alternatively, the complementary level of confidence 1-α, is a measure of the confidence you can have in the decision.

While testing the null-hypothesis H_0 there are two unfavourable situations that might occur:

- Rejection of H_0 while in fact it is true. The probability of this situation occurring is equal to the significance level α. This situation is called a Type I error (see the following table).
- Acceptance of H_0 while in fact it is false. The probability of this situation occurring is 1-β, with b being the power of the test. This situation is called a Type II error (see the following table).

Situation	Decision: accept H_0	Decision: reject H_0
H_0 true	correct decision: probability = 1-α	Type I error: probability = α
H_0 false	Type II error: probability = 1-β	correct decision: probability = β

See the following topics for further information on testing the null-hypothesis and alternative hypotheses:

F-Test

W-Test

T-Test

3.9.7.2 F-Test

F-Test

The F-Test is a commonly used multi-dimensional test for checking the null-hypothesis H_0. The F-Test is often called the overall model test, because it tests the given mathematical and stochastic models in general.

The F-value is defined by the expression:

$F = s^2/\alpha^2$

with

s^2 = a-posteriori variance factor derived from the computed residuals and the redundancy

α^2 = a-priori variance factor.

The F-value is tested against a critical value of the F-distribution, which is a function of the redundancy and the significance level α. But the information provided by the F-Test, that is, the acceptance or rejection of the null-hypothesis, is not specific. Therefore, if H_0 is rejected it makes sense to ask for the reasons why by tracing errors in observations or pre-assumptions.

There are three reasons for rejection:

- **Gross errors**
  If you suspect that H_0 is rejected because of a gross error in one of the observations, then the so-called data snooping provides you with revealing information by applying the W-Test in order to seek for and find errors in individual observations. The F-Test and the W-Test are interlinked by a common value for the power of test β. The method is the called the B-Method of testing.
- **Incorrect mathematical model**
  If you suspect that H_0 is rejected because of a mathematical model that is incorrect or not refined enough, then check the adjustment settings. For example, redefine and make the vertical refraction coefficient or the scale factor correction be used to improve the mathematical model.
  See also:
  Advanced Terrestrial Parameters
- **Incorrect stochastic model**
  If you suspect that H_0 is rejected because of a stochastic model that is incorrect or not refined enough, then check the adjustment settings . If the a-priori variance-covariance matrix is too optimistic increase the input standard deviations and/or centering/height errors of the observations to improve the stochastic model.
  See also:
  TPS Accuracy Information

☞ When you optimise the mathematical or stochastic model always bear in mind that the purpose of statistical testing is not to have all observations accepted, but to detect outliers and model errors.

3.9.7.3 Chi-Square Test

Chi-Square Test

Unlike the F-Test used in the B-Method of testing, the Chi-Square is a two-tailed test, that is, it tests the Chi-Square value against both the upper and the lower bound.

In practice, if the Chi-Square value is lower than the lower bound, this means that the standard deviations are too pessimistic. On the other hand, if the Chi-Square value is higher than the upper bound, this can be interpreted in the same way as if the F-Test fails.

See also:

Interpreting Test Results

3.9.7.4 W-Test

W-Test

A rejection of the F-Test does not directly indicate the reason for the rejection itself. Thus, in case the null-hypothesis is rejected, other hypotheses must be formulated if you want to describe a possible error or a combination of errors.

There is an infinite number of hypotheses which you could formulate as an alternative for the null-hypothesis. But the more complex these hypotheses become the more difficult they are to interpret. A simple but effective hypothesis, though, is the so-called conventional alternative hypothesis, based on the assumption that you have to suspect an outlier in only one observation while all others are correct. The one-dimensional test associated with this hypothesis is called W-Test.

The assumption that just a single outlier corrupts your network is, indeed realistic. A strong rejection of the F-Test can often be traced back to a gross error in just one observation. The conventional alternative hypothesis for each observation implies that each individual observation is tested. The process of testing each observation in a network by a W-Test is called data snooping.

Often the size of the least squares correction alone is not precise enough to indicate an outlier when you check the observations in your network. A better test quantity, though only suited for uncorrelated observations, is the least squares correction divided by its standard deviation. For correlated observations, like for example the three coordinate elements of a baseline, the complete weight matrix of the observations must be taken into account. This condition is fulfilled with the test quantity W of the W-Test, which has a standard normal distribution and is most sensitive to errors in one of the observations.

The critical value W_{crit} depends on your choice of the significance level α_0. If the W-Test is rejected because of $W > W_{crit}$, there is a probability of $1-\alpha_0$ that the corresponding observation indeed holds an outlier. On the other hand, there is a probability of α_0 that the observation does not hold an outlier, which implies that the rejection was unjustified. In geodesy, values for α_0 between 0.001 and 0.05 are most commonly used. The following table gives you an overview on the α_0-values and their corresponding critical values. Which α_0 you effectively choose depending on how strict and rigid you intend to test your observations. A strict testing (with a small critical value), leads to a larger α_0 and therefore, to an increased probability to reject valid observations. An α_0 value of 0.001 implies one false rejection in every 1000 observations. This has proven to be a realistic choice.

significance level α_0	0.001	0.010	0.050
critical value W-Test	3.29	2.58	1.96

Essential for a successful testing by the B-Method is that an outlier is detected with the same probability by both the F-Test and the W-Test. To achieve this, the test power β is by default fixed on a level of usually 0.80 for both tests. You can adapt β by 10% up or down in the settings for the test criteria. The W-Test level of significance α_0 is selected, to which leaves the F-Test level of significance α to be determined. With α_0 and β being fixed, α strongly depends on the redundancy in the network and increases for large networks with many observations and a considerable amount of redundancy. In such networks, it becomes difficult for the F-Test to react on a single outlier. The F-Test, being an overall model test, is not sensitive enough to achieve this task. Thus, it is common practice to do data snooping, no matter what the result of the F-Test might be.

See also:

Test Criteria

3.9.7.5 T-Test

T-Test

As discussed in the topic on the W-Test, the W-Test is a 1-dimensional test which checks the conventional alternative hypothesis. This hypothesis assumes that there is just one erroneous observation at a time. This so-called data snooping works very well for single observations, for example for directions, distances, zenith angles, azimuths or height differences. However, for some observations such as GPS baselines, it is not enough to test the DX-, DY-, DZ-components of each vector separately. It is imperative to test the baseline as a whole as well.

For this purpose we introduce the T-Test. Depending on the dimension of the quantity to be tested, the T-Test is a 3- or 2-dimensional test. As with the W-Test, the T-Test is also interlinked with the F-Test by the B-Method of testing. The T-Test has the same power as both other tests, but has its own levels of significance and its own critical values (see tables below).

Significance level/critical value for 2-dimensional T-Test, based on α_0 of the W-Test:

significance level α_0	0.001	0.010	0.050
significance level α (2-dim)	0.003	0.022	0.089
critical value T-Test	5.91	3.81	2.42

Significance level/critical value for 3-dimensional T-Test, based on a_0 of the W-Test:

significance level α_0	0.001	0.010	0.050
significance level α (3-dim)	0.005	0.037	0.129
critical value T-Test	4.24	2.83	1.89

The T-Test is equally useful when you want to test for slight deformations on known stations. The data snooping successfully finds an outlier that might have occurred, for example, due to a typing error in either the Easting or the Northing or the height component causing a gross error in one of the coordinate components. But the deformation of a whole station might stay undetected by the data snooping when the shifts caused by the deformation in each coordinate component are relatively small. For testing a possible deformation influencing the whole coordinate triplet, like with the Easting and the Northing and the height, a different alternative hypothesis is needed. The 3-dimensional T-Test on the complete coordinate triplet is better equipped to trace the deformation, although its not able to trace the exact direction into which the station has moved.

☞ A situation in which the W-Test is accepted but the associated T-Test of the observation is rejected, which is not unlikely in practice, does not necessarily imply a contradiction. It is simply a matter of having tested different hypotheses.

3.9.7.6 Interpreting Test Results

Interpreting Test Results

When dealing with test results we always have to keep in mind that a certain amount of probability is involved in the process, and that there is no absolute truth. Statistics in general should be used with discretion, together with common sense, practical experience and external independent evidence.

As discussed in the topic on the F-Test a rejection of the F-Test, meaning a rejection of H_0, may happen because of:

- Gross errors or blunders.
- An incorrect mathematical model.
- An incorrect stochastic model.

Needless to say, a combination of these reasons for a rejection is also possible, which makes it difficult to define strict rules for drawing conclusions from the F-Test results. In general, a rejected F-Test in combination with a pattern of rejected W-Tests points to a model error. An incidental W-Test rejection points to one or more gross errors or blunders.

Since F-Test, W-Test and T-Test are interlinked, it is best to interpret all tests in combination:

- A rejected F-Test in combination with a limited number of rejected W-Tests (T-Tests) usually points to one or more gross errors.
- If the F-Test is rejected and all observations of a specific type (for example all zenith angles) are rejected as well, the problem probably lies within the mathematical model which needs correction or refinement then. For instance, if all W-Tests for the zenith angles are rejected, it could be helpful to include refraction coefficients.
- If the F-Test is rejected together with most of the W-Test results (without extremes), the problem probably lies within the stochastic model and you can suspect that the input standard deviations are too optimistic. On the other hand, if the F-Test result is well below the critical value and the W-Test (T-Test) results are all close to zero, then the input standard deviations are possibly too pessimistic.

Example:

Suppose that the data snooping on the observations in your network results in a (limited) number of rejections. Further assume that the rejections are not caused by mathematical model errors and that obvious errors such as typing mistakes have been fixed.

Then this leaves you with a number of options:

- **Remove the corresponding observations**
  This is a valid but rather abrupt way of handling rejections. Remember that the removal of observations decreases the redundancy and, therefore, influences precision and reliability.
- **Remeasure the corresponding observations**
  Remeasuring observations is an obvious but often expensive way to eliminate rejections, especially when the fieldwork is already completed. Thus, it is recommended to process and cross-check as much of the data as possible on site.
- **Increase the standard deviation of the corresponding observations**
  Increasing the standard deviation of an observation always works, because it always results in decreasing the F-, W-, T-Test values. Keep in mind, though, that the goal is not to get all tests accepted, but rather to detect blunders or model errors!
- **Ignore the rejections**
  This option is obviously very risky and should only be applied if the W-Test results just exceed the critical value. Have a look at the estimated error coming with the rejection and see whether it is acceptable or not, because we also have to keep in mind that, depending on the level of significance, it may always happen that a valid observation is rejected.

☞ Unless there is clear evidence of the reason for an error, for example a typing error, never edit observations just to make it better fit into your network!