Lumo Recorder - User Guide 1.1

specim

SPECTRAL IMAGING

Contents

Lumo Recorder User Manual

Contacting Support

Further information and technical support are available from **Specim, Spectral Imaging Oy Ltd.** in Finland. Contact information:

- WWW: *www.specim.fi*
- Email: *support@specim.fi*
- Tel: +358 (0)10 4244 400

Disclaimer

All information provided in this guide and provided manuals is believed to be complete, accurate and reliable at the time of delivery. No responsibility is assumed by Specim, Spectral Imaging Oy Ltd. for its use. Specim, Spectral Imaging Oy Ltd reserves the right to make changes to this information without notice. Reproduction of this manual in whole or in part, by any means, is prohibited without prior permission having been obtained from Specim, Spectral Imaging Oy Ltd.

Company names and product names are trademarks or registered trademarks of their respective owners.

Installation

This section will guide you through the installation of the software.

Installing Qt

This section describes how to install Qt.

All Specim applications are built on top of the Qt application framework (see: *http://qt-project.org*). To minimize the installer sizes and speedup the update, we provide a shared installer.

The installer is named `Specim_QtRuntime_redist_setup_5.4.1.exe`. **5.4.1** is the Qt revision number. This information is subject to change if Qt updates are required.

Proceed as follows:

1. Double-click the `Specim_QtRuntime_redist_setup_5.4.1.exe` file.

   The installation wizard is opened.

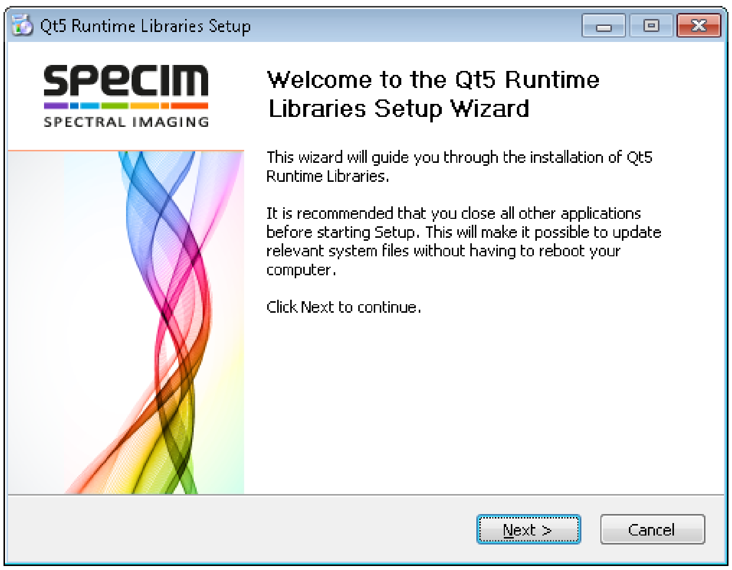


Figure 1: Qt Installation Wizard

2. Select **Next**.

   The license screen is opened.

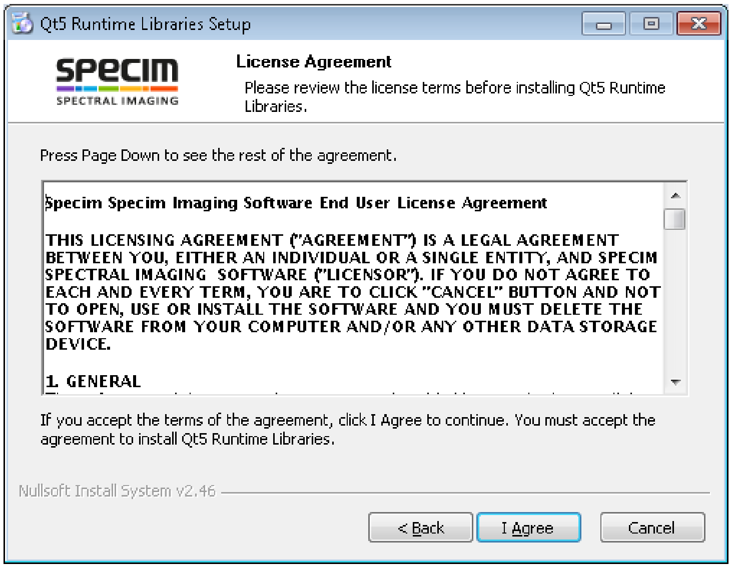


Figure 2: Qt License Screen

3. Accept the license by selecting **I Agree**.

   The component selection screen is opened.

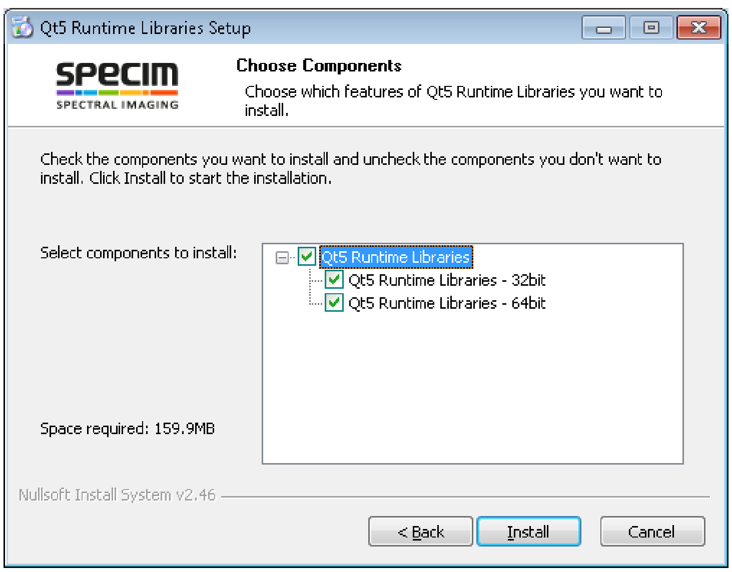


Figure 3: Qt Component Selection Screen

   Choose the components to be installed. We recommend using the default settings.

4. Select **Install**.

   The installation progress screen is opened.

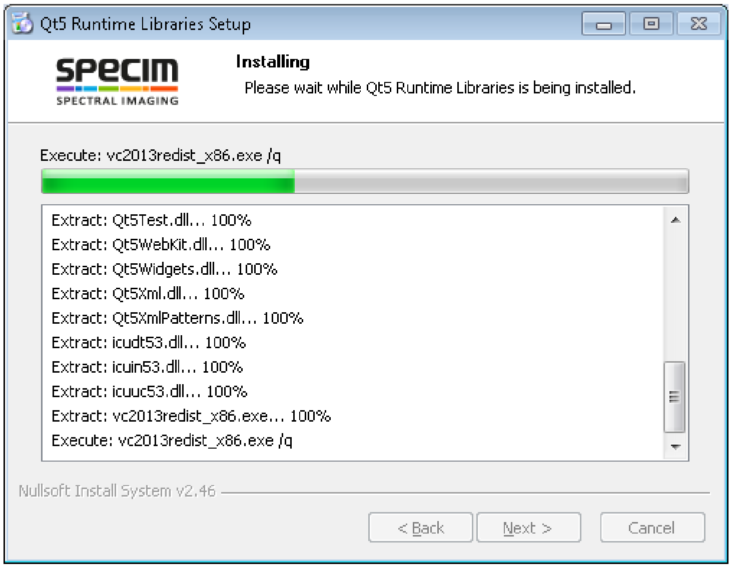


Figure 4: Qt Installation Progress Screen

5. After a successful installation, the wizard prompts you to reboot the computer.

   The installer declares some environment variables at the system scope, thus we recommend rebooting the computer to insure the new settings are properly propagated.

   Reboot the computer by selecting the **Reboot now** radio button and selecting **Finish**.

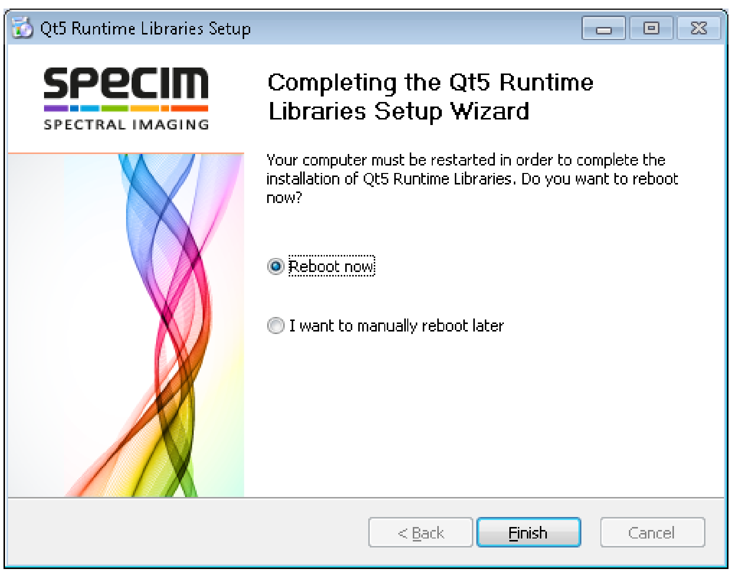


Figure 5: Completing the Qt Installation

Installing Intel Libraries

This section describes how to install Intel libraries.

The application engine makes use of some Intel optimized libraries. The installer is large in size and the libraries are shared, so they come as a separate installer.

The installer name is `Specim_Intel_redist_setup_1.1.0.exe`. The 1.1.0 revision number is subject to changes.

Proceed as follows:

1. Double-click the `Specim_Intel_redist_setup_1.1.0.exe` file.

The installation wizard is opened.

Figure 6: Intel Libraries Installation Wizard

2. Select **Next**.

   The license screen is opened.

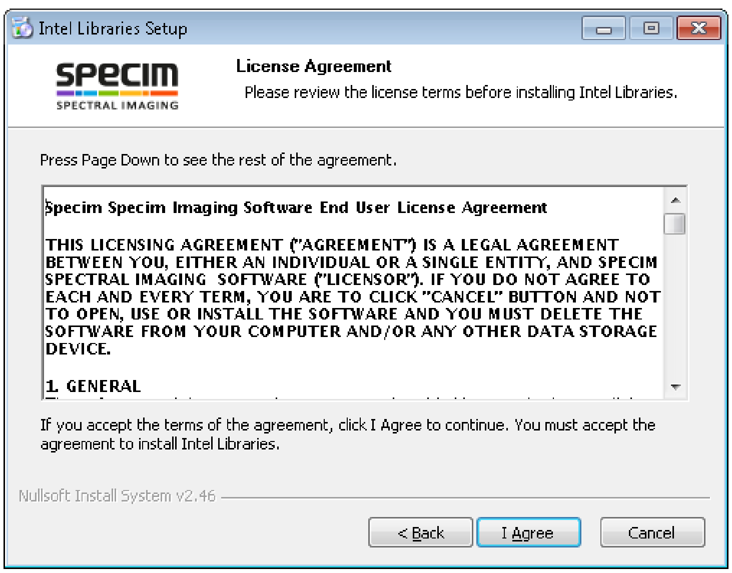


Figure 7: Intel Libraries License Screen

3. Accept the license by selecting **I Agree**.

   The component selection screen is opened.

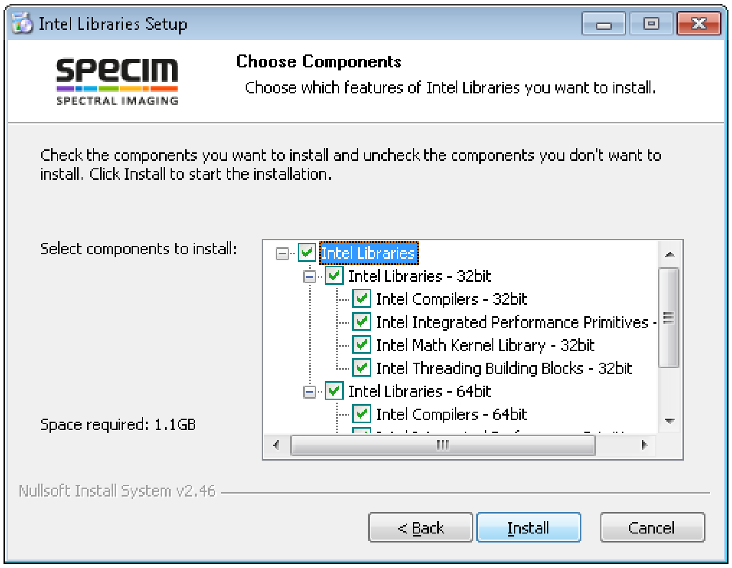


Figure 8: Intel Libraries Component Selection Screen

Choose the components to be installed. We recommend using the default settings.

4. Select **Install**.

   The installation progress screen is opened.

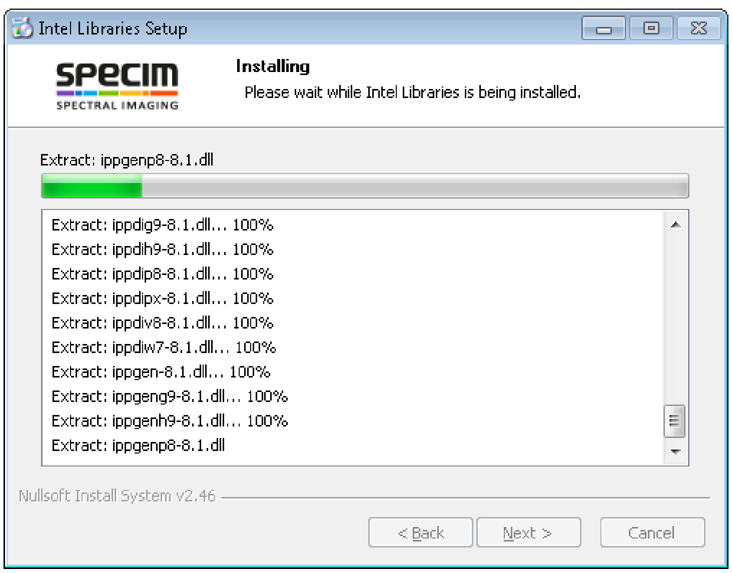


Figure 9: Intel Libraries Installation Progress Screen

5. After a successful installation, the wizard prompts you to reboot the computer.

   The installer declares some environment variables at the system scope, thus we recommend rebooting the computer to insure the new settings are properly propagated.

   Reboot the computer by selecting the **Reboot now** radio button and selecting **Finish**.

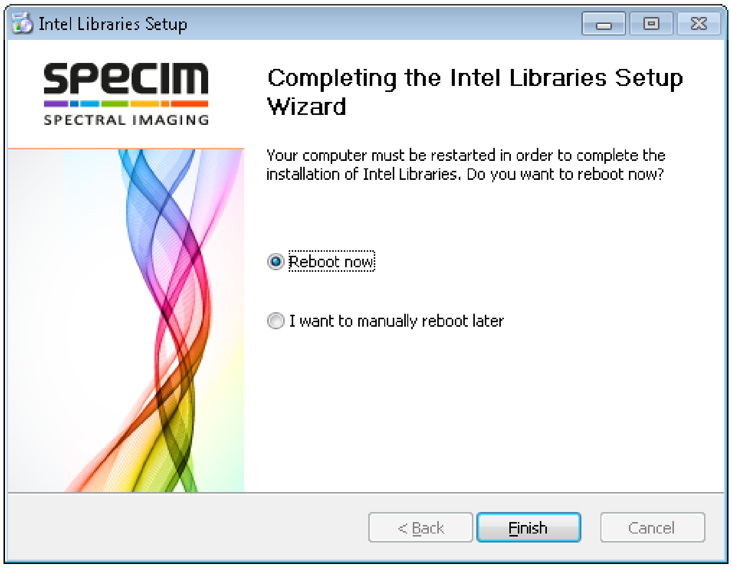


Figure 10: Completing the Intel Libraries Installation

Installing Lumo Recorder

This section describes how to install Lumo Recorder.

The sensor hardware must installed before installing Lumo Recorder, otherwise the paths detection for coping camera files will fail and you will not be able to use your sensor.

Proceed as follows:

1. Run the Lumo Recorder installer.

   The installation wizard is opened.

Figure 11: Lumo RecorderInstallation Wizard

2. Select **Next**.

   The license screen is opened.

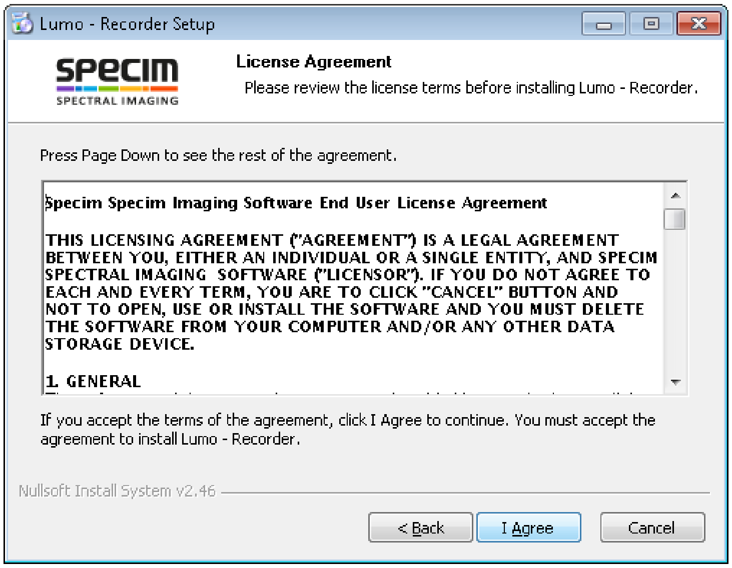


Figure 12: Lumo Recorder License Screen

3. Accept the license by selecting **I Agree**.

   The component selection screen is opened.

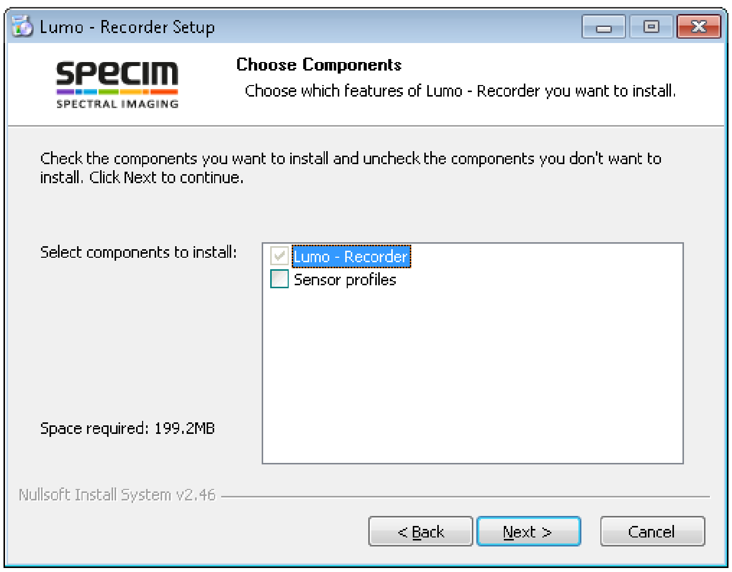


Figure 13: Lumo Recorder Component Selection Screen

4. Select the components to install and select **Next**. If you select the **Sensor profiles** component, the installer will copy the *SSP*s to **C:\Users\Public\Documents\Specim**.

   The start menu selection screen is opened.

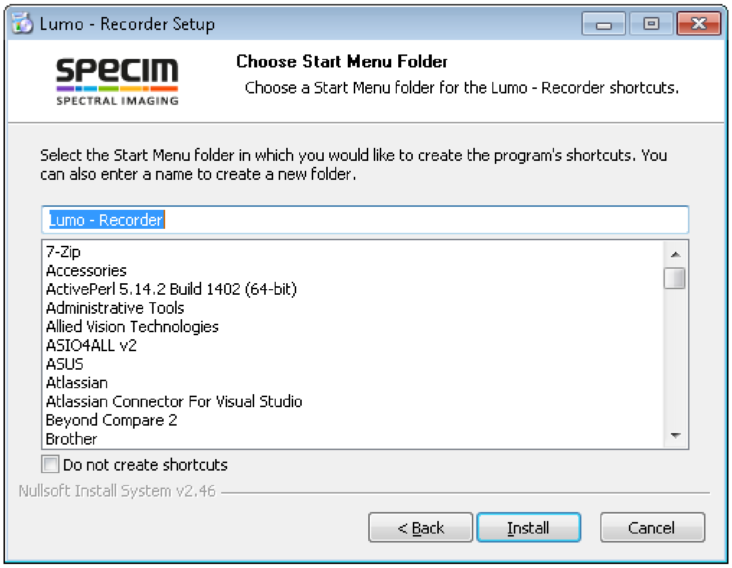


Figure 14: Lumo Recorder Start Menu Selection Screen

5. Select **Install**.

   The installation progress screen is opened. At the end of the installation, an import setting dialog will be opened.

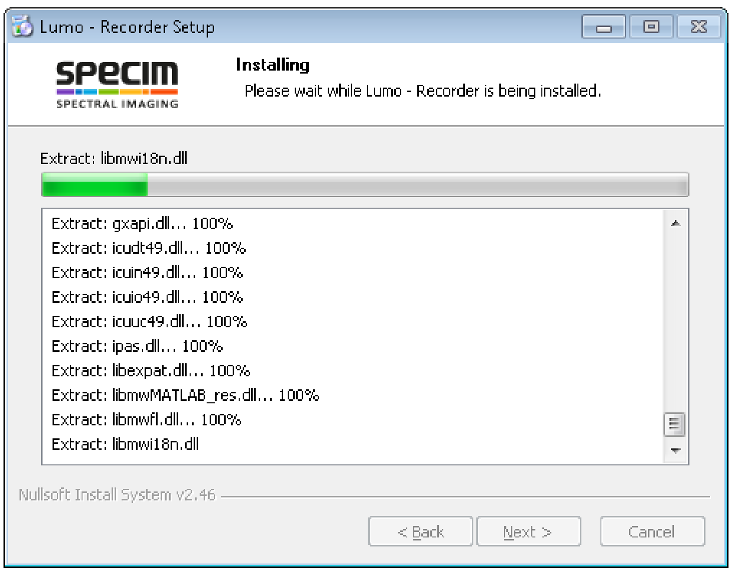


Figure 15: Lumo Recorder Installation Progress Screen

6. Select **Yes** to import the settings from the previous installation. It the import fails, please contact Specim's support (support@specim.fi) and forward error report is created.

   The wizard final page is opened.

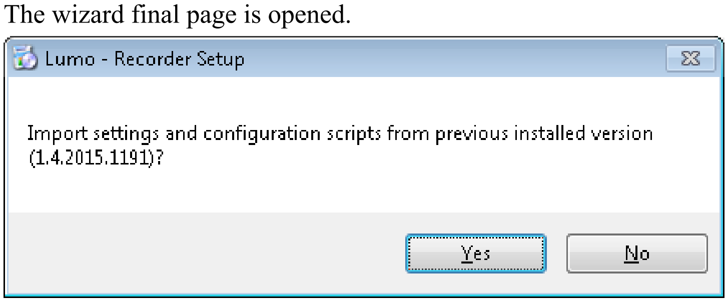


Figure 16: Lumo Recorder Import Settings Screen

7. Close the OpenGL Warning Screen

If your computer does not meet the minimum OpenGL requirements, a message box is opened. Click the **OK** button to continue with the installation. You cannot use Lumo Recorder until you install the correct drivers or install a graphics card with OpenGL 2.0 support, at least.

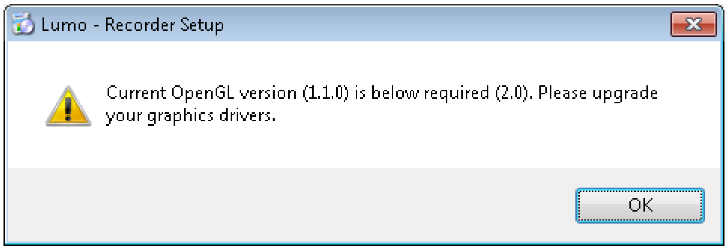


Figure 17: OpenGL Warning Screen

8. After a successful installation, the wizard prompts you to reboot the computer.

   The installer declares some environment variables at the system scope, thus we recommend rebooting the computer to insure the new settings are properly propagated.

   Reboot the computer by selecting the **Reboot now** radio button and selecting **Finish**.

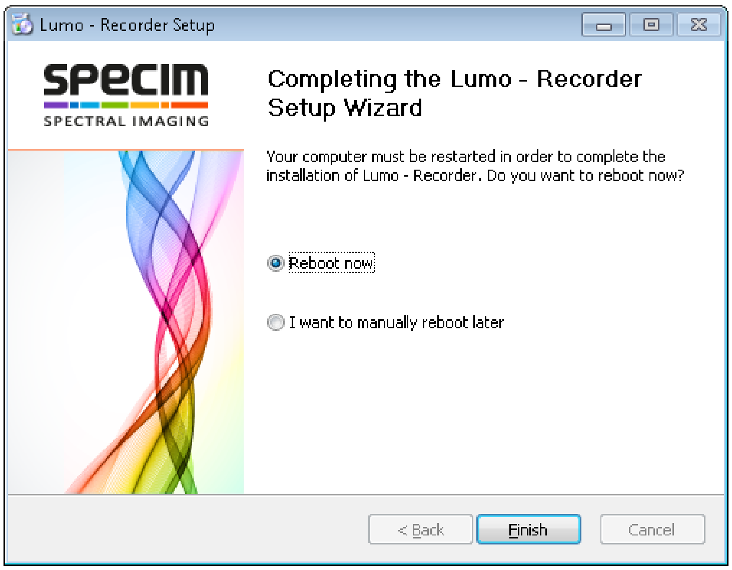


Figure 18: Completing the Lumo Recorder Installation

About Lumo Recorder

Lumo Recorder is a data recording application for Specim's **spectral cameras** and **airborne** systems.

Lumo Recorder can control *HSI* and *GPS*/*INS* sensors, and store their data onto the computer hard drive. It can also control other external devices, required for certain sensors, such as thermal calibrator, triggering electronics, motors, and so on.

Lumo Recorder has a built-in scripting and **sequential workflow** engine **controlling** all the **steps** and **automation** to be performed for each measurement.

Thanks to its **remote control server**, Lumo Recorder can be controlled through **Lumo Remote SDK.** It allows integrating Lumo Recorder into other systems, and allows building more sophisticated networks of sensors as in **SisuROCK** systems.

If the acquisition computer hardware and performance allow, Lumo Recorder can handle multiple sensors on the same computer.

If the imaging setup requires multiple sensors, which cannot be installed on the same computer, Lumo Recorder can be configured to use **remote** sensors. These remote sensors are controlled using **Lumo Remote SDK**.

Lumo Recorder offers several options to control the quality of the data (***QA/QC***) as part of the standard recording **workflow**.

If the recorded **dataset** must be **transferred** to another computer, Lumo Recorder has an option to automatically queue the dataset to **Lumo FileTransfer Server**.

Minimum requirements

This application is supported on Windows 7 and onward. The computer must meet the following hardware requirements:

- Intel Core 2 Duo.
- 4GB of RAM.
- 128GB SSD hard drive or more for the operating system.
- 256GB SSD hard drive or more for each sensor to record the data.
- Graphics card with OpenGL 2.0 support or better.
- 64-bit operating system to allow large file handling.
- Screen resolution of 1024 x 800.

Lumo Graphical User Interface

All Lumo applications use the same tabbed interface layout. Each tab categorizes controls and features.

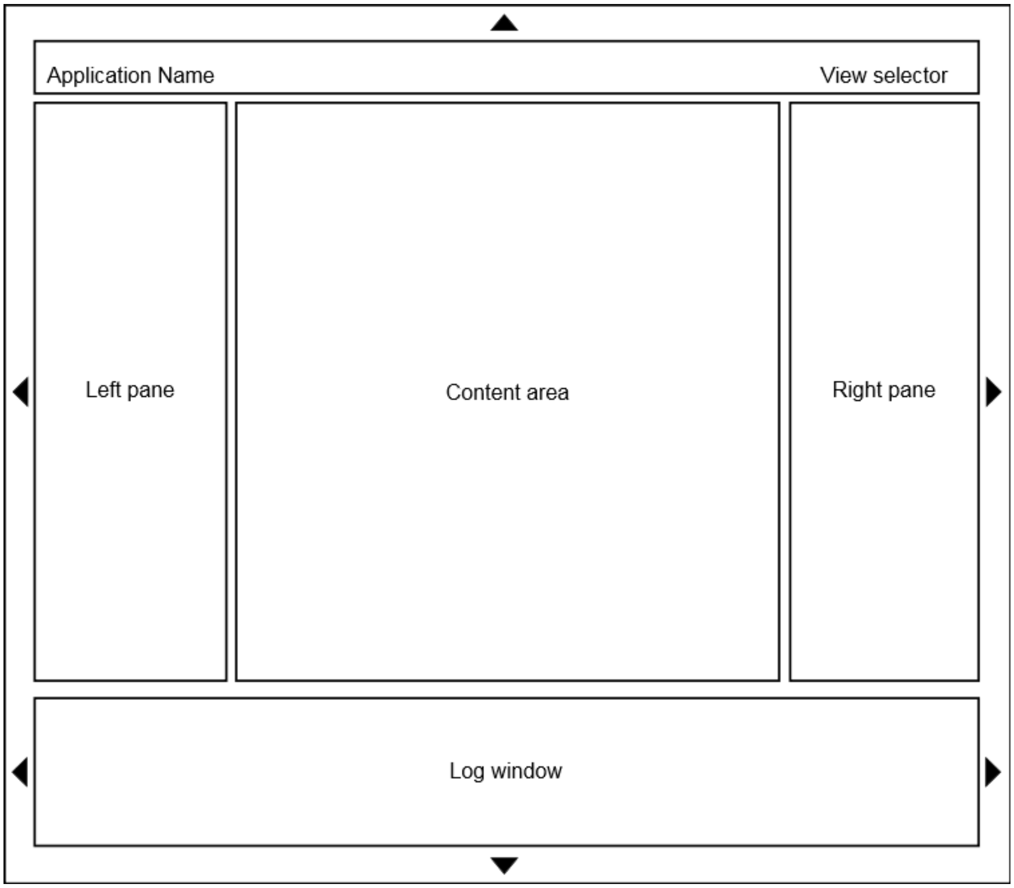


Figure 19: Lumo Graphical User Interface Layout

The general layout consists of a **title bar**, **left pane**, **right pane**, **bottom pane** and the **content area**.

GUI areas

The *GUI* areas are:

- **Title bar** — The **Title bar** contains the application name and the view selector. The name of the active view is highlighted.
- **Left pane** and **Right pane** — The **Left pane** and **Right pane** widths can be changed, and are independent for each view. In other words, each view can have a different width for the panes.

  You can change the width by clicking the sliders between the **content area** and the panes, and sliding them.
- **Bottom pane** — You can change the **Bottom pane** height by clicking the slider between the view area and the **bottom pane**, and sliding it.

  The **bottom pane** height is the same for all the views.
- **System tray icon** — When enabled in the software, the application can add an icon to the Windows system tray.

  Usually, the icon is used to set the application window back to visible, when the **Minimize the application to system tray**option is enabled (see *Defining the General Parameters*).

Visibility

The visibility of the **title bar** and the three panes can be controlled by clicking the areas with the triangle icons.

- The triangle above the **title bar**. The visibility of the **title bar** is the same for all the views.
- The triangle on the left side of the **left pane**. The visibility of the **left pane** is independent for each view.
- The triangle the right side of the **right pane**. The visibility of the **right pane** is independent for each view.

- The triangle below the **bottom pane**. The visibility of the **bottom pane** is the same for all the views.
- The triangles on the left and right side of the **Log window** are not visible, as they are reserved for future use.

Lumo Recorder Views

You can switch between the views by using the View selector, located in the upper right corner of the Title bar.

Lumo Recorder has three views:

- The Setup View.
- The Adjust View.
- The Capture View.

Carry out the initial setup-related tasks in the Setup View, see *Initial Setup*.

After the intial setup is completed, carry out the sensor adjustments in the Adjust View, see *Adjusting the Sensor*.

Once the system is ready, you can operate Lumo Recorder in the Capture View, see *Operating Lumo Recorder*.

Setup View

The Setup View has a wizard-like interface, where the initialisation steps are arranged in the execution order. Clicking a **navigation button** brings up the desired dialogue. Use the mouse wheel to scroll the dialogues view.

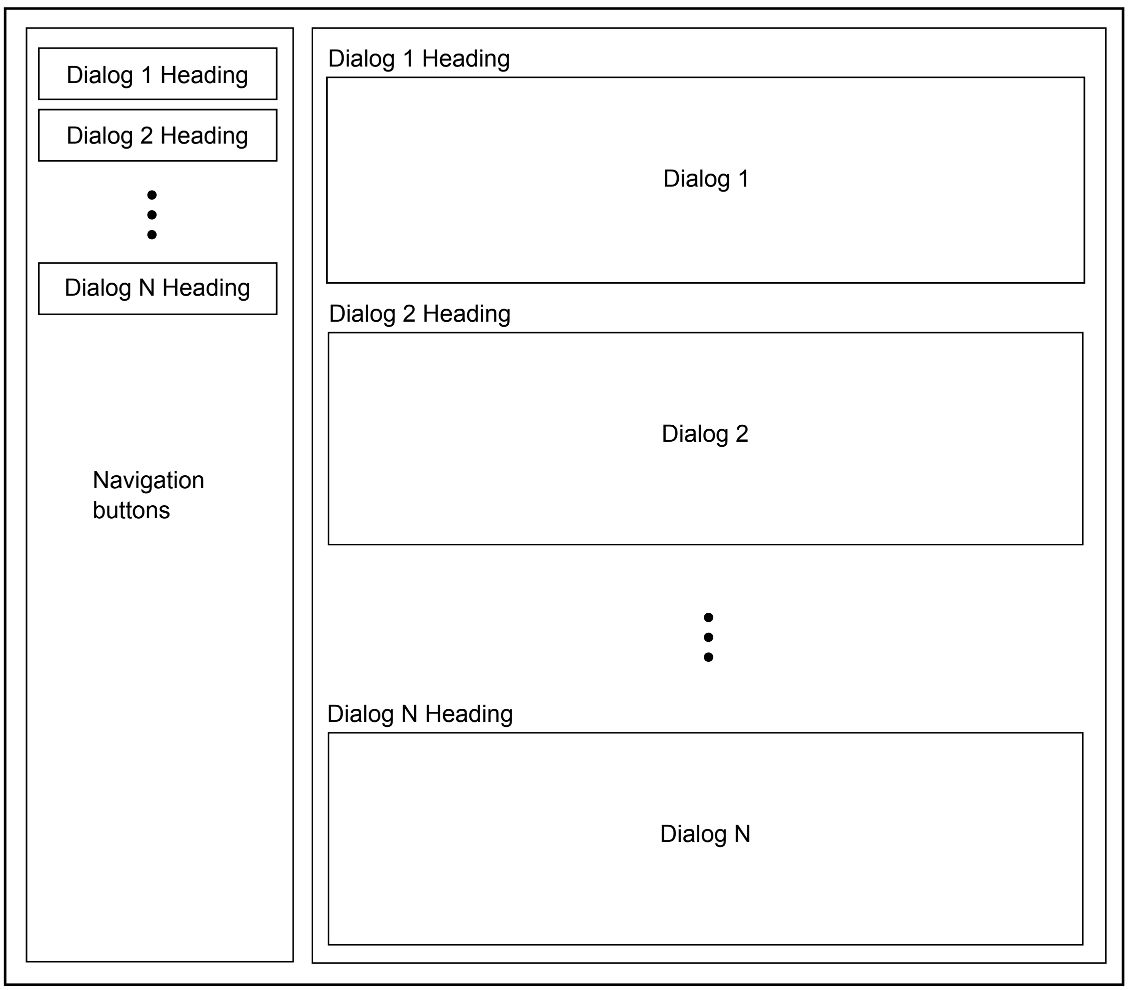


Figure 20: The Setup View

The **Dialog N Headings** in *Figure 20: The Setup View*, also referred to as settings group, are replaced with the settings group name, such as:

- Dataset
- Remote control server
- Logging

Adjust View

The Adjust View View area contains four sections:

- The Detector *Widget*.
- The Waterfall *Widget*.
- The Spectral Plot *Widget*.
- The Spatial Plot *Widget*.

Note: The four *Widget*s display data from the activated sensor only. With multiple sensor setups, a **View** button is visible on the upper right corner of the sensor parameters group, located in the **left pane**. Click the **View** button of the corresponding sensor to show its live data in the widgets.

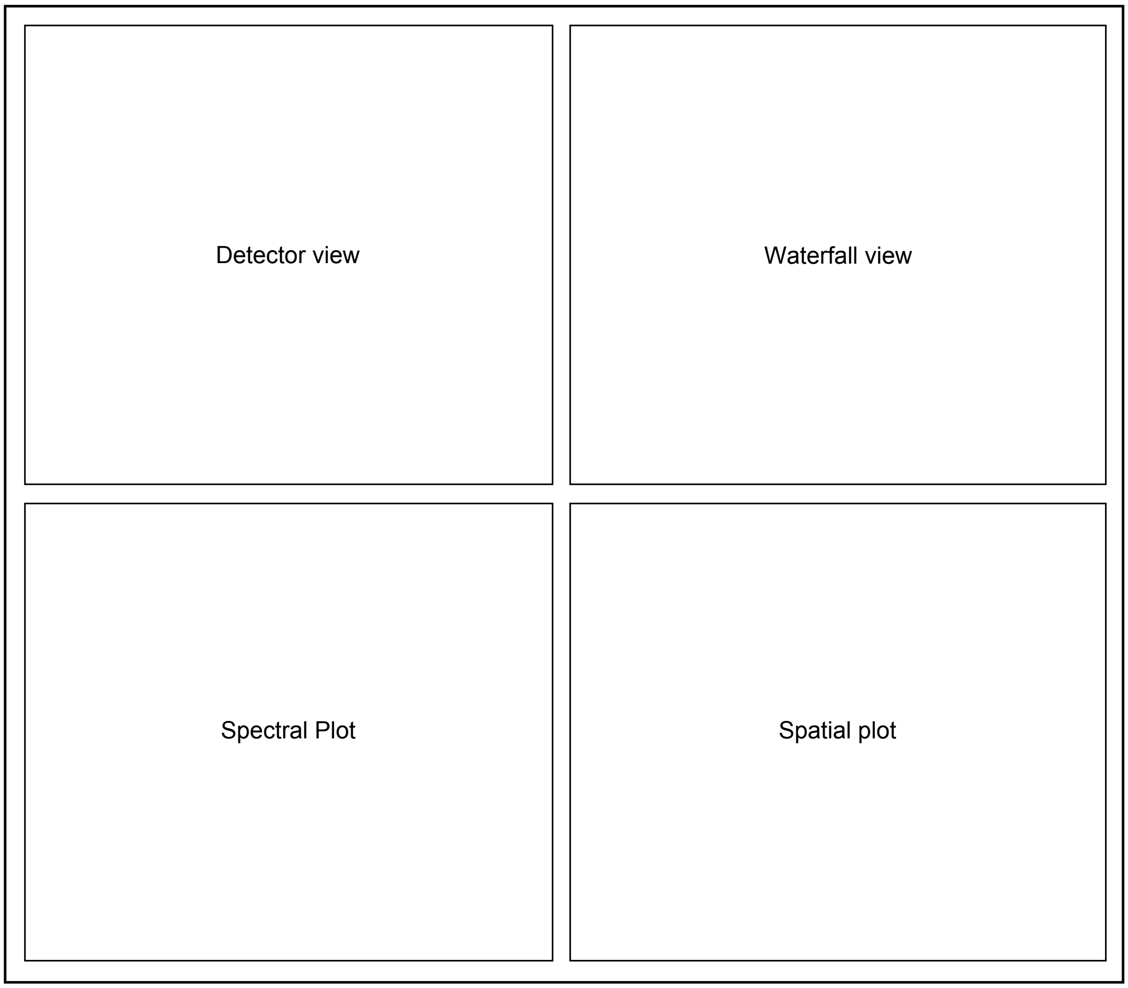


Figure 21: The Adjust View View area

Left Pane

When the Adjust View is selected, the left pane contains the following controls:

- Sensor(s) parameters

Right Pane

When the Adjust View is selected, the right pane contains the following controls:

- Image correction parameters
- Focusing tool
- *QA/QC* parameters

Detector widget

The Detector *Widget* provides a greyscale image of the incoming frames.

- A dark grey pixel in the image corresponds to a low raw pixel value.
- A white grey pixel in the image corresponds to a large raw pixel value.

The image in the Detector *Widget* shows the incoming frame as a matrix, where the first pixel coordinate is on the upper left-hand corner.

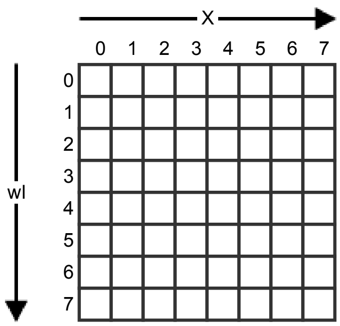


Figure 22: Detector view

The pixels along the `X` direction refer to the spatial dimension, and the pixels along the `wl` direction refer to the spectral dimension. A detector column represents the entire spectra at a spatial location.

Saturated pixels are shown as bright red. A pixel is considered saturated if its value is equal to the maximum pixel value.

The Detector *Widget* is useful to perform tasks, such as:

- Saturation monitoring.
- Image control.
- Setting the *RGB* bands.
- Spectral references definition.
- Spatial references definition.
- Focusing.
- Setting the GUI ROI.

Detector Widget Parameters

The Detector *Widget* context menu contains the following parameters:

- **Zoom in** — This parameter allows you to zoom in the image. You can also zoom in by holding the `Ctrl` key and using the mouse scroll button.
- **Zoom out** — This parameter allows you to zoom out the image. You can also zoom out by holding the `Ctrl` key and using the mouse scroll button.
- **Fit to window** — This parameter will automatically calculate the zoom level required to fit the entire Detector *Widget* image in the parent window, while keeping the aspect ratio.

  Tip: While **Fit to window** is activated, you cannot **Zoom in** or **Zoom out**.

- **Show cursor value** — When this parameter is enabled, a blue text is displayed in the lower left-hand corner of the Detector *Widget*. The text is in the format:

```
dn:..., x:..., y:...[...nm]
```

where

- The `dn` section displays the raw pixel value under the mouse cursor.
- The `x` section displays the pixel's x coordinate.
- The `y` section displays the pixel's y coordinate.
- The `nm` section displays the corresponding wavelength value.

- **Show *RGB* bands** — This parameter controls the visibility of the *RGB* lines on the Detector *Widget*. When enabled, a red, a green and a blue horizontal line is plotted on top of the detector image. The vertical position of the lines correspond to the wavelength they are set to.

  The RGB bands can be changed in the Detector *Widget* as follows:

  1. Place the mouse cursor on top of a band.
  2. Select it by holding the left button of the mouse.
  3. Move the band by sliding the mouse up or down.
  4. Release the left button of the mouse.
- **Show spatial references** — This parameter controls the visibility of the spatial reference lines on the Detector *Widget*. Spatial references are added from the Detector *Widget* **Options** dialog (see *Managing References*), and plotted in the Spatial Plot *Widget*.

  The spatial reference indices can be changed in the Detector *Widget* as follows:

  1. Place the mouse cursor on top of a reference line.
  2. Select it by holding the left button of the mouse.
  3. Move the band by sliding the mouse up or down.
  4. Release the left button of the mouse.
- **Show spectral references**: This parameter controls the visibility of the spectral reference lines on the Detector *Widget*. Spectral references are added from the Detector *Widget* **Options** dialog (see *Managing References*), and plotted in the Spectral Plot *Widget*.

  The spectral references indices can be changed in the Detector *Widget* as follows:

  1. Place the mouse cursor on top of a reference line.
  2. Select it by holding the left button of the mouse.
  3. Move the band by sliding the mouse left or right.
  4. Release the left button of the mouse.
- **Select focus region**: This parameter allows you to draw a *ROI* rectangle on the Detector *Widget*.

  1. Hold the left button of the mouse.
  2. Move the mouse to draw a rectangle on the *ROI*
  3. Release the left button of the mouse.

  Tip: The *ROI* rectangle is only visible when **Select focus region** is enabled. When enabled, you cannot change the *RGB*, spatial reference indices, or spectral references indices.
- **ROI settings** — The *GUI ROI* allows you to only show a *ROI* of the acquired frame, in the Detector *Widget*. This is particularly useful for sensors like the OWL, where only a subset of the frame contains image data. All the band indices will be recalculated to fit the *GUI ROI* selection.

  This menu item opens a dialog, where you can **Enable** the *GUI ROI*, set the **Start band**, and set the **Stop band**. The **Start band** and **Stop band** parameters contain a list of all the bands and wavelengths available for the sensor. The options are in format:

  ```
  Band ...: ...nm
  ```

  where

  - The `Band` section is the band index.
  - The `nm` section is the corresponding wavelength.

Managing References

You can add, edit or remove spatial and spectral references in the Detector *Widget*'s options dialog.

The options dialog has two tabs; one for the spectral references and one for the spatial references.

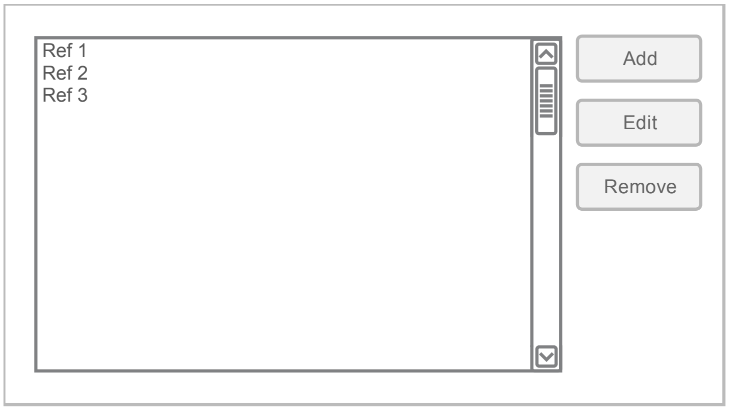


Figure 23: Reference Tab in Detector *Widget*'s Options Dialog

The each tab shows a list of the declared references. Each reference is shown with a rectangle of the selected color, followed by the label.

On the right hand side you can find the **Add**, **Edit** and **Remove** buttons.

Adding a reference

1. Click the **Add** button.
2. Enter a **Label** name (optional).
3. Select a **Color** to differentiate the new reference from the others.
4. Click **OK**.

The new reference is added to the list.

In the case of a spectral reference, a new vertical line is added in the middle of the Detector *Widget*, and the new reference is plotted in the Spectral Plot *Widget*.

In the case of a spatial reference, a new horizontal line is added in the middle of the Detector *Widget*, and the new reference is plotted in the Spatial Plot *Widget*.

Editing a reference

1. Double click the reference in the list, or select the reference, and click the **Edit** button.
2. Modify the **Label**.
3. Modify the **Color**.
4. Click **OK**.

The label and color changes are reflected in the list, Detector *Widget*, Spectral Plot *Widget* and Spatial Plot *Widget*.

Removing a reference

1. Select the reference in the list
2. Click the **Remove** button.

The reference is removed from the list, Detector *Widget*, Spectral Plot *Widget* and Spatial Plot *Widget*.

Waterfall widget

The Waterfall *Widget* provides a false color representation of the hyperspectral imager data - as it is acquired - in real-time.

As a new frame is acquired, the Waterfall *Widget* is updated in the following order:

1. The Waterfall *Widget* image is shifted by one pixel in the Y direction, freeing the first row for the new line.
2. The three selected *RGB* bands are extracted from the new frame.
3. These three bands are interlaced, and scaled to 8-bit for each color palette into one line.
4. The resulting line is added to the first row.

The *RGB* bands can be configured in:

- The sensor setup settings group, see *Defining the Sensor Parameters*.
- The Detector *Widget*, see *Detector widget*.
- The Spectral Plot *Widget*, see *Spectral Plot Widget*.

The Waterfall *Widget* **hightlights saturated data**. Saturated pixels are shown in bright red. A pixel is considered saturated, if more then 2% of its wavelength pixels are equal to the maximum pixel value.

Data is visualized after image correction operations. For example, enabling the image correction (see *Defining the Image Correction Parameters*) causes the Waterfall *Widget* to show normalized data.

Waterfall Widget Parameters

The Waterfall *Widget* context menu contains the following parameters:

- **Zoom in** — This parameter allows you to zoom in the image. You can also zoom in by holding the `Ctrl` key and using the mouse scroll button.
- **Zoom out** — This parameter allows you to zoom out the image. You can also zoom out by holding the `Ctrl` key and using the mouse scroll button.
- **Fit to window** — This parameter will automatically calculate the zoom level required to fit the entire Waterfall *Widget* image in the parent window, while keeping the aspect ratio.

  Tip: While **Fit to window** is activated, you cannot **Zoom in** or **Zoom out**.

- **Horizontal flip** — This parameter flips the image horizontally. The recorded data remains unaffected.
- **Vertical flip** — This parameter flips the image vertically. The recorded data remains unaffected.
- **Rotate *CW*** — This parameter rotates the image clockwise. The recorded data remains unaffected.
- **Rotate *CCW*** — This parameter rotates the image counterclockwise. The recorded data remains unaffected.
- **Contrast Brightness** — This menu item opens a **Contrast** & **Brightness** dialog, where you can fine tune both parameters. The recorded data remains unaffected.

Spectral Plot Widget

The Spectral Plot *Widget* shows the pixels values at the locations of **spectral references** as two-dimensional plots.

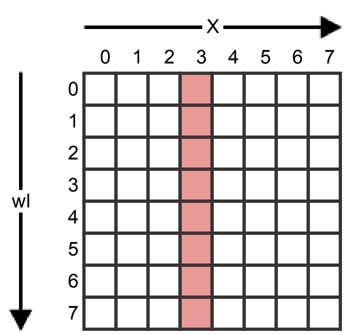


Figure 24: Spectral Reference on the Detector

A spectral reference refers to a column on the detector.

The corresponding wavelength of the row number is shown in the X axis. The raw pixel value is shown on the Y axis, and the maximum pixel value depends on the sensor used.

You can add spectral references through the Detector *Widget*'s context menu. (See *Managing References*).

RGB bands in Spectral Plot *Widget*

The RGB bands are drawn in the Spectral Plot *Widget* at the positions specified by their corresponding wavelengths.

The RGB bands can be changed in the Spectral Plot *Widget*:

1. Place the mouse cursor on top of a band.
2. Select it by holding the left button of the mouse.
3. Move the band by sliding the mouse left or right.
4. Release the left button of the mouse.

Plot Widget Parameters

The plot parameters allow the customization of the X and Y axis. The parameters can be found from the **Options** menu item in the plot's context menu.

Axis parameters

- **Show label** — This parameter controls the visibility of the label on top of the plot (X axis), or on the right-hand side of the plot (Y axis). It is enabled, that is, visible, by default.
- **Label** — This parameter resides on the right-hand side of **Show label** parameter, and controls the text in the label on top of the plot (X axis), or on the right-hand side of the plot (Y axis). The default labels are:
  - Spatial plot X axis —`Pixels`
  - Spectral plot X axis —`Wavelength`
  - Spatial and Spectral Y axis — *DN*
- **Show grid** — This parameter controls the visibility of the vertical grid lines (X axis), or the horizontal grid lines (Y axis). It is enabled, that is, visible, by default.
- **Grid** —This parameter resides on the right-hand side of **Show grid** parameter, and controls the number of vertical grid lines (X axis), or horizontal (Y axis) grid lines.
- **Autoscale** —This parameter automatically calculates the **Min** and **Max** range parameters for the X and Y axis.
  - Spatial plot X axis — All the spatial pixels are plotted.
  - Spectral plot X axis — All the spectral pixels are plotted.
  - Spatial and Spectral Y axis — The range is continuously updated. For the **Min** parameter, the closest exponent to two, that is below the parameter value itself, is used. For example, if the value is 130, the closest exponent of two is 2*7=128. For the **Max** parameter, the closest exponent to two, that is above the parameter value itself, is used. For example, if the value is 230, the closest exponent of two is 2*7=256. With the values above, the range would be [128, 256].

Plot parameter

- **Update graph** — When this check box is checked, redrawing is enabled. Otherwise, it is paused.

Context menu shortcuts

The plot's context menu provides the following shortcuts:

- **Update** — A shortcut for **Update graph**
- **Autoscale**:
  - **X-axis** — A shortcut for the X axis **Autoscale**
  - **Y-axis** — A shortcut for the Y axis **Autoscale**
- **Options** — Opens the plot's options menu.

Keyboard shortcuts

The keyboard shortcuts are:

- **Update** — `space` key
- **Set the manual scale** — Hold the `Ctrl` key, press the left mouse button, select an area on the plot, and release the left mouse button.
- **Reset the manual scale** — Hold the `Ctrl` key, and click the right mouse button.

Spatial Plot Widget

The Spatial Plot *Widget* shows the pixels values at the locations of **spatial references** as two-dimensional plots.

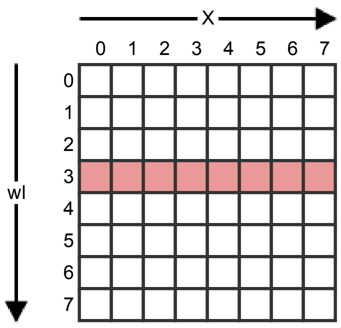


Figure 25: Spatial Reference on the Detector

A spatial reference refers to a row on the detector.

The column number is shown as the X axis. The raw pixel value is shown on the Y axis, and the maximum pixel value depends on sensor used.

You can add spatial references through the Detector *Widget*'s context menu. (See *Managing References*).

Plot Widget Parameters

The plot parameters allow the customization of the X and Y axis. The parameters can be found from the **Options** menu item in the plot's context menu.

Axis parameters

- **Show label** — This parameter controls the visibility of the label on top of the plot (X axis), or on the right-hand side of the plot (Y axis). It is enabled, that is, visible, by default.
- **Label** — This parameter resides on the right-hand side of **Show label** parameter, and controls the text in the label on top of the plot (X axis), or on the right-hand side of the plot (Y axis). The default labels are:
  - Spatial plot X axis —`Pixels`
  - Spectral plot X axis —`Wavelength`
  - Spatial and Spectral Y axis — *DN*
- **Show grid** — This parameter controls the visibility of the vertical grid lines (X axis), or the horizontal grid lines (Y axis). It is enabled, that is, visible, by default.
- **Grid** —This parameter resides on the right-hand side of **Show grid** parameter, and controls the number of vertical grid lines (X axis), or horizontal (Y axis) grid lines.
- **Autoscale** —This parameter automatically calculates the **Min** and **Max** range parameters for the X and Y axis.
  - Spatial plot X axis — All the spatial pixels are plotted.
  - Spectral plot X axis — All the spectral pixels are plotted.
  - Spatial and Spectral Y axis — The range is continuously updated. For the **Min** parameter, the closest exponent to two, that is below the parameter value itself, is used. For example, if the value is 130, the closest exponent of two is 2*7=128. For the **Max** parameter, the closest exponent to two, that is above the parameter value

itself, is used. For example, if the value is 230, the closest exponent of two is 2*7=256. With the values above, the range would be [128, 256].

Plot parameter

- **Update graph** — When this check box is checked, redrawing is enabled. Otherwise, it is paused.

Context menu shortcuts

The plot's context menu provides the following shortcuts:

- **Update** — A shortcut for **Update graph**
- **Autoscale**:
  - **X-axis** — A shortcut for the X axis **Autoscale**
  - **Y-axis** — A shortcut for the Y axis **Autoscale**
- **Options** — Opens the plot's options menu.

Keyboard shortcuts

The keyboard shortcuts are:

- **Update** — `space` key
- **Set the manual scale** — Hold the `Ctrl` key, press the left mouse button, select an area on the plot, and release the left mouse button.
- **Reset the manual scale** — Hold the `Ctrl` key, and click the right mouse button.

Capture View

The Capture View content area contains one to multiple Waterfall *Widget*, depending on the number of sensors present in the system.

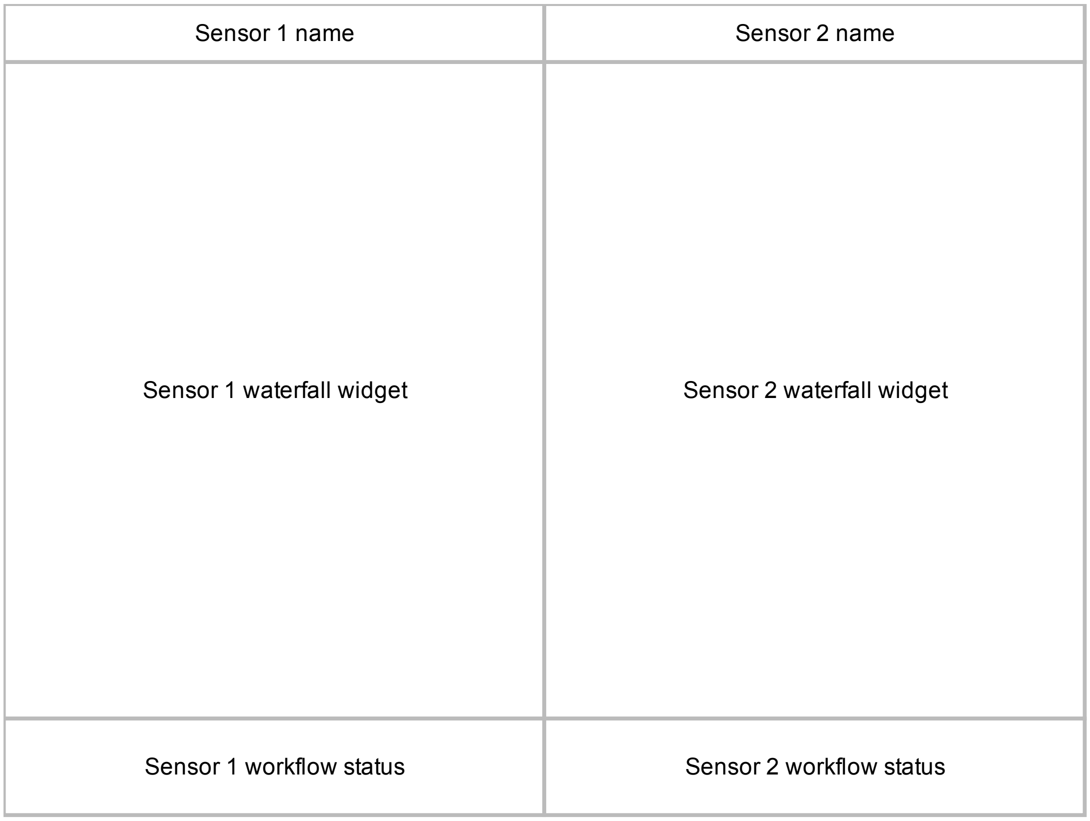


Figure 26: Capture View with Two Sensors

The Capture View content area shows, side by side, the sensor name, the waterfall status, and the workflow status.

- The height of the **Sensor X name** is fixed. The width is synchronized with the Waterfall *Widget* and the workflow status widget below. **Sensor X name** is replaced with the actual sensor name.
- The height of the **Sensor X waterfall widget** changes dynamically to fill the entire heigh of the content area. It displays the same image data, saturations, and so on, as the Adjust View for sensor X.
- The height of the **Sensor X workflow status** changes automatically as new status information is added. The area contains:
  - The `Frame rate` and `disk space` status, in one of the following formats:
    - `Fps: ... | Free disk space: ...h ...m ...s`.
    - `Fps: ... | Free disk space: ...GB`.

    Note: You can select to display the free disk space either in time or in size. Select **Free disk space** > **Show disk space in time**, or **Free disk space** > **Show disk space in size**.
  - The recording status for each new file type recorded. The format is: `Type: Recorded: ... | Dropped: ...`, where:
    - `Type` is replaced with `Image`, `White reference`, `Dark reference`, and so on. See *Table 5: File Prefixes* for the list of types.
    - `Recorded` is followed by the number of recorded frames on the disk.
    - `Dropped` is followed by the number of dropped frames.

Each sensor name, waterfall and workflow status column width can be enlarged to have more space for one sensor.

1. Place the mouse cursor on top of the splitter between the columns.
2. Hold the left button of the mouse.
3. Move the mouse left or right to change the column size.

4. Release the left button of the mouse.

Left Pane

When the Capture View is selected, the left pane contains the following controls:

- Sensor(s) parameters

Right Pane

When the Capture View is selected, the right pane contains the following controls:

- Recorder group: This group has the **Stop** and **Record** buttons used to start, stop or abort the workflows.
- Workflow settings group: This group contains the workflow selection and the parameter definition for each of them.
- Metadata group: Usually, this group contains the **Dataset name**, **Operator** and **Description**. The metadata fields are configurable so the content of this group will change depending on the setup.
- Traffic lights group: shows some vital system information, warning and error. See *Monitoring the Traffic Lights*.

Initial Setup

This section will guide you through the initial setup and configuration of the software.

Activating the License

You must activate your software copy to use it. The license is bound to one computer, and an Internet connection is required to complete the process.

When you receive your software copy, Specim will supply you with an activation code (for example, `330DA7-AEEB3F-F4F5C4-B8175C-78846C-5B2184-6440`). This activation code is necessary to activate your copy, and also to create a new user account on our licensing portal.

If you already have an existing account on Specim's licensing portal, you can jump to the *Email Activation* or *Online Activation* section.

Registering to the Licensing Portal

This section describes how to register to the licensing portal.

Proceed as follows:

1. Open the licensing portal:
   - Open *https://licensing.specim.fi*.
   - In Windows, select **Start** > **All programs** > **Specim** > **Your application**.
2. The SpecSensor license wizard is opened.

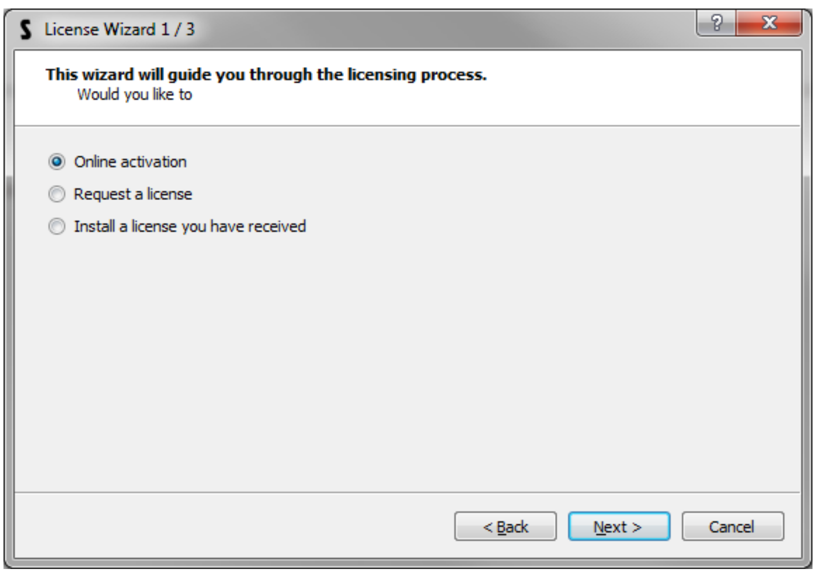


Figure 27: License Wizard

3. Select the **Online activation** radio button.
4. Press the **Next** button.
5. The licensing portal is opened in your browser.

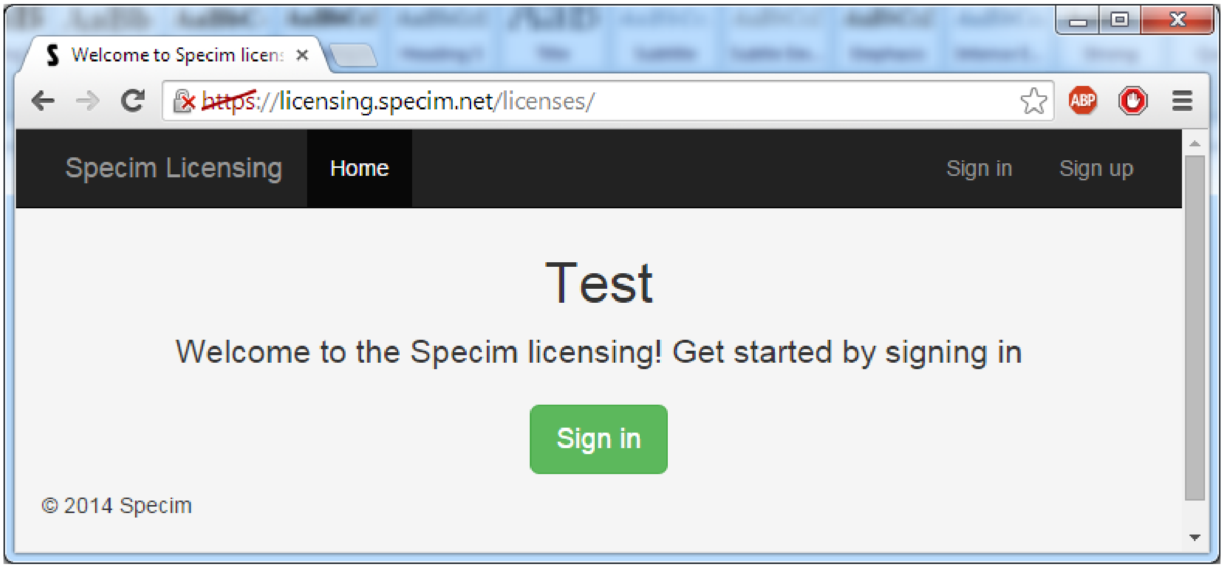


Figure 28: Specim Licensing Portal

6. Select **Sign up**.
7. A registration form is opened.

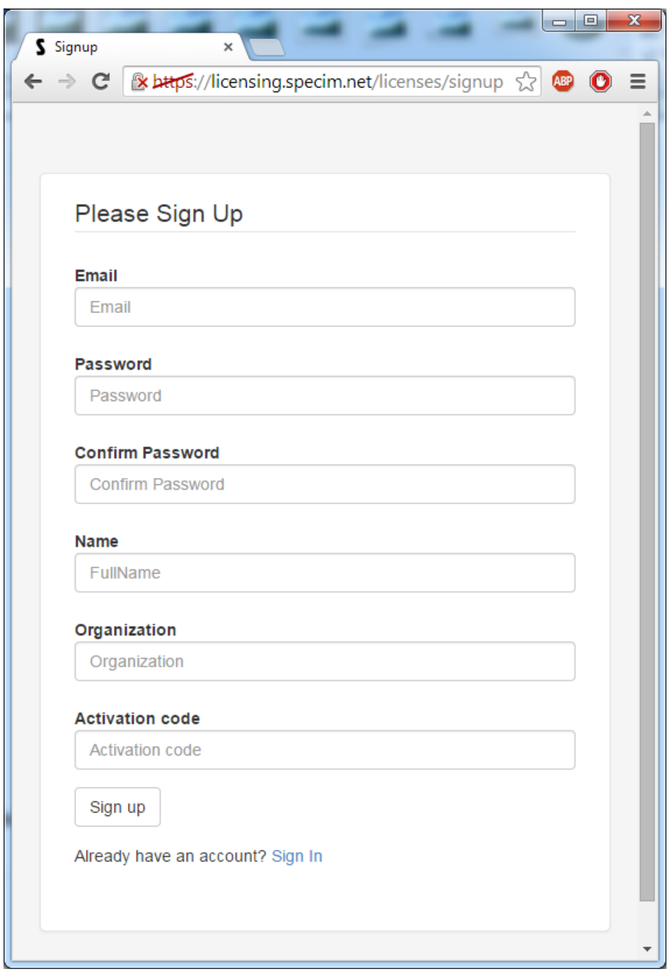


Figure 29: Specim Licensing Portal Registration Form

8. Fill in the fields and press the **Sign up** button.

9. Shortly, you will receive a validation email containing a link to activate your account; click on it.
10. You now have a valid account and can activate your software license.

Online Activation

This section describes how to activate your license online.

Proceed as follows:

1. In Windows, select **Start** > **SpecSensor License Wizard**.
2. The SpecSensor license wizard is opened.

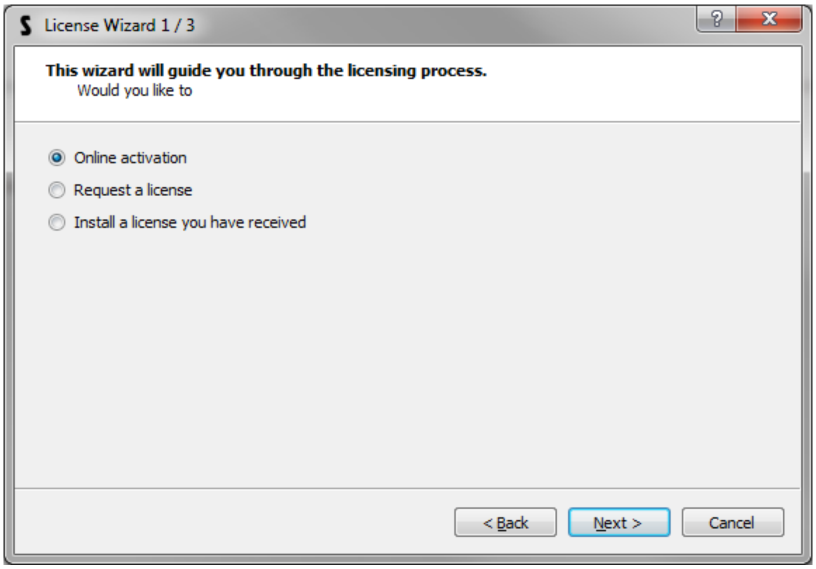


Figure 30: License Wizard

3. Select the **Online activation** radio button.
4. Press the **Next** button.
5. Enter the email address you used to create your account, your password, and the activation code you received.

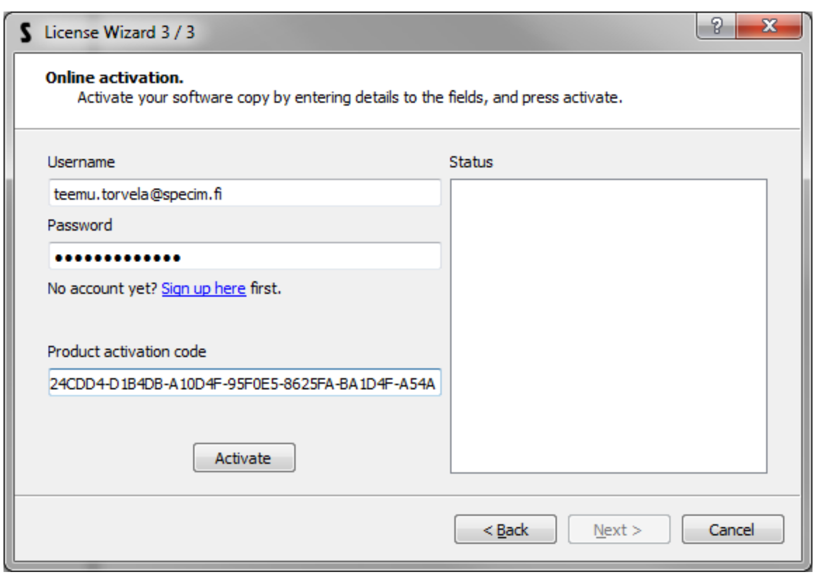


Figure 31: License Wizard - Online Activation

6. Select **Activate**.
7. The wizard will contact the licensing server, check for valid information, download and install the license files.

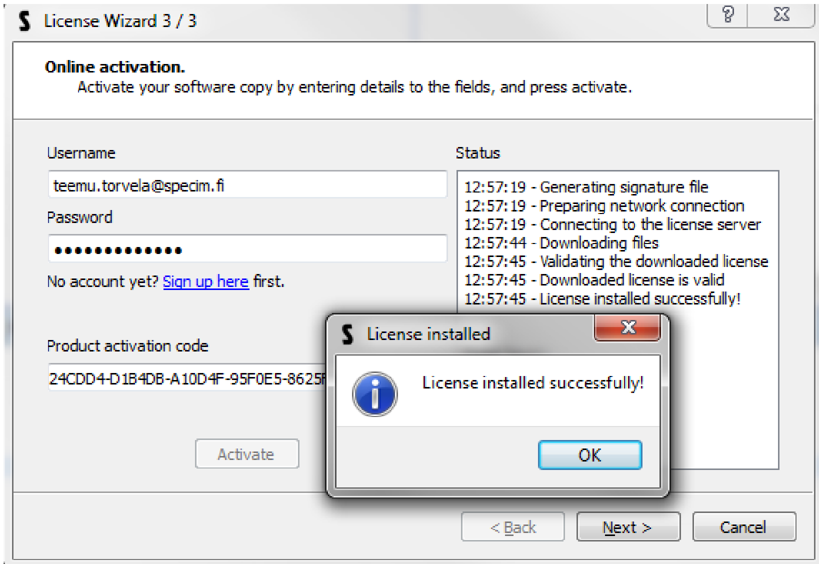


Figure 32: License Wizard - Successful License Activation

If an error occurs, it will be displayed in the **Status** field. Contact Specim support at support@specim.fi with the error information for further help.

8. Upon valid activation, press the **Next** button.
9. Finally, the wizard will thank you for choosing Specim products.

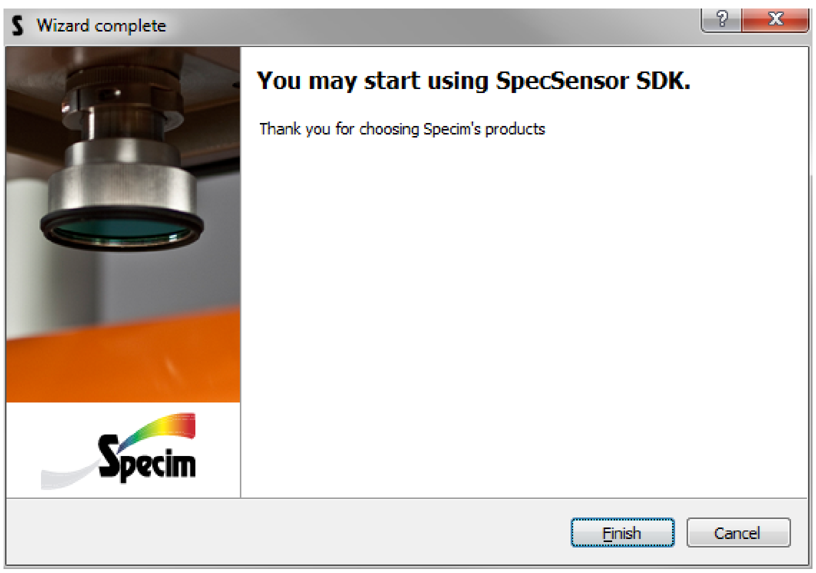


Figure 33: License Wizard - Final Page

10. Press the **Finish** button.

Email Activation

This section describes how to activate your license by email.

If the computer running the software cannot get access to internet, you can first generate a signature file on it, upload it to the licensing portal from another computer, download the license files manually and have the wizard install them.

Proceed as follows:

1. In Windows, select **Start** > **SpecSensor License Wizard**.
2. The SpecSensor license wizard is opened.

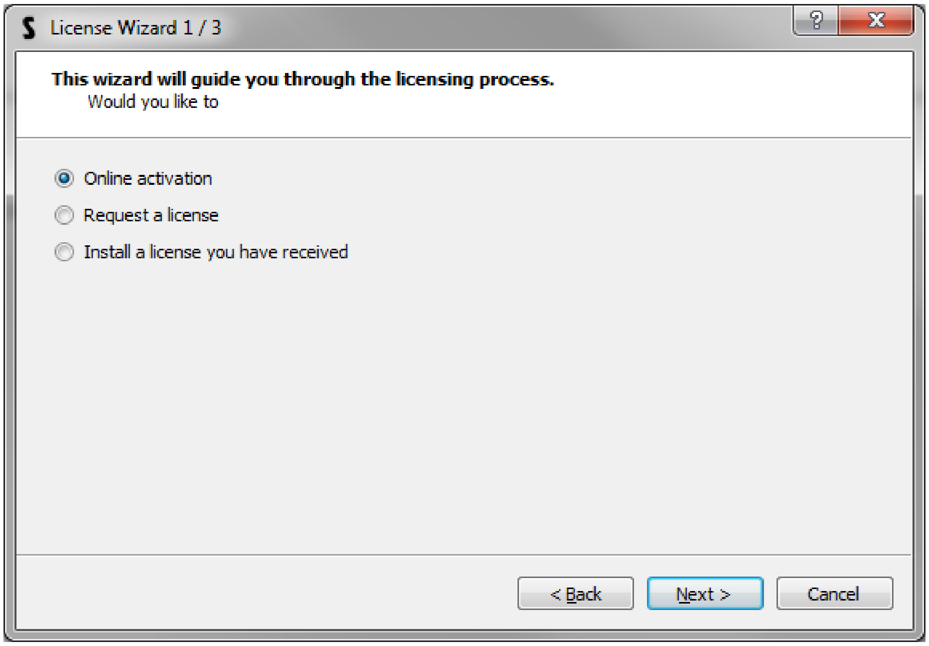


Figure 34: License Wizard

3. Select the **Request a license** radio button.
4. Press the **Next** button.
5. The system generates a license for you and opens the **Request License File** screen.

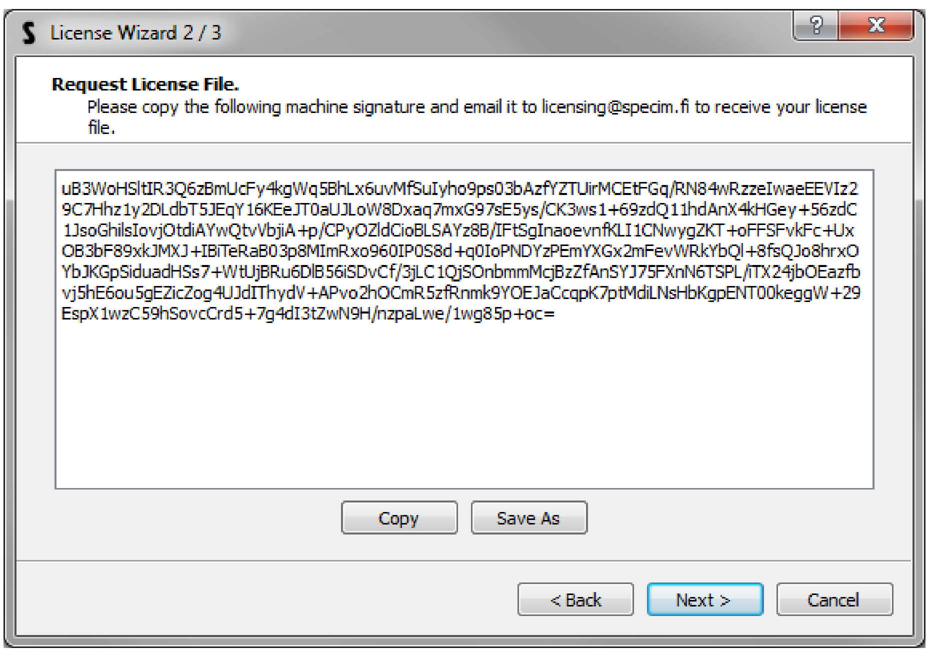


Figure 35: License Wizard - Requesting a License

6. Select **Save as** to to save the signature as text file.
7. Copy this signature file to an external storage device, and find a computer with Internet connectivity.
8. Open the licensing portal at *hpps://licensing.specim.fi*, and login with your credentials.
9. Select **My products** > **Register and request a license**.

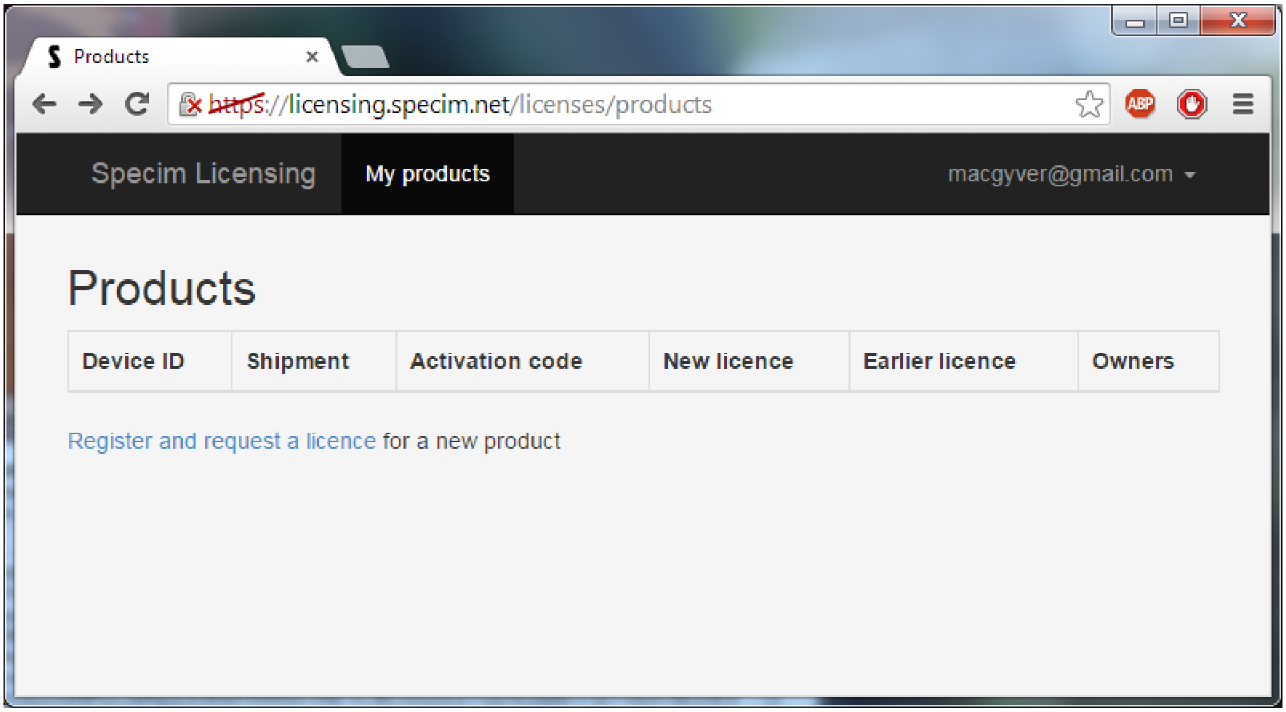


Figure 36: Licensing Portal - My Products

10. The **Request a license** page is opened.
    a) Enter the activation code for your product in the **Activation code** field.
    b) Select **Choose File** and select the signature file that you saved on the external storage device.
    c) Select **Submit**.

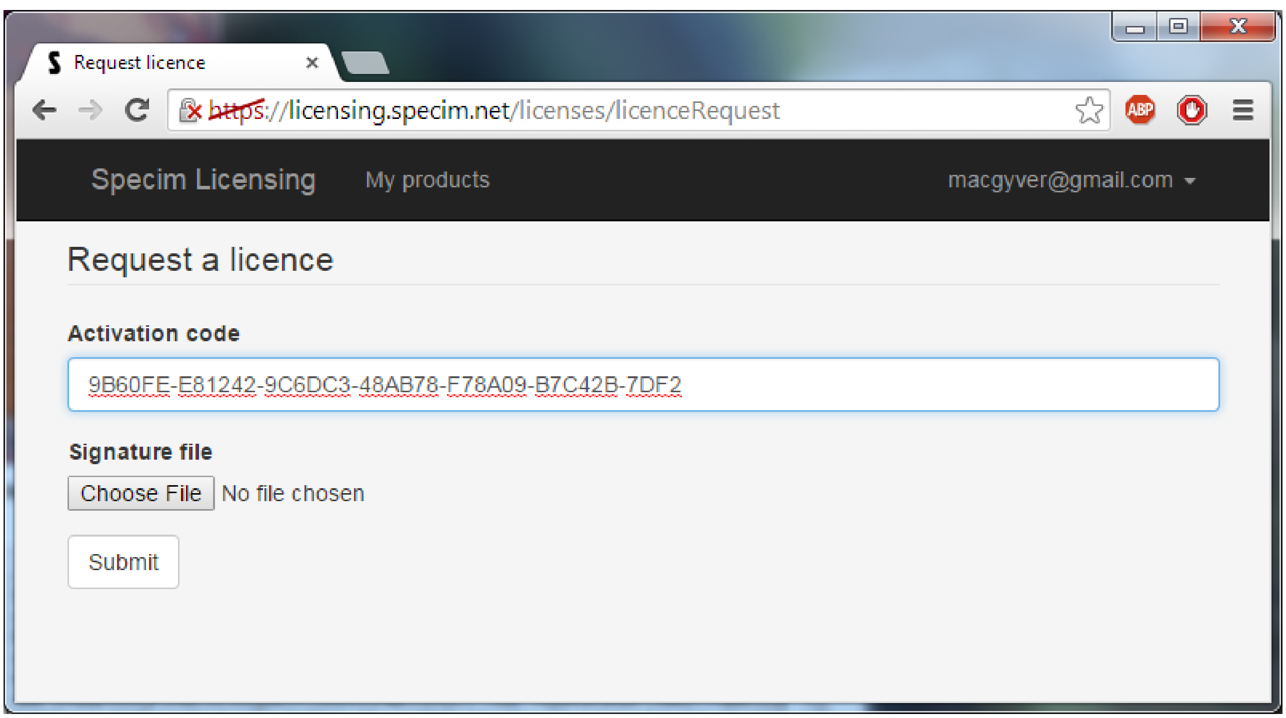


Figure 37: Licensing Portal - Requesting a License

11. Assuming you have supplied a valid activation code and signature file, you will be able to download the license by selecting **Download.**

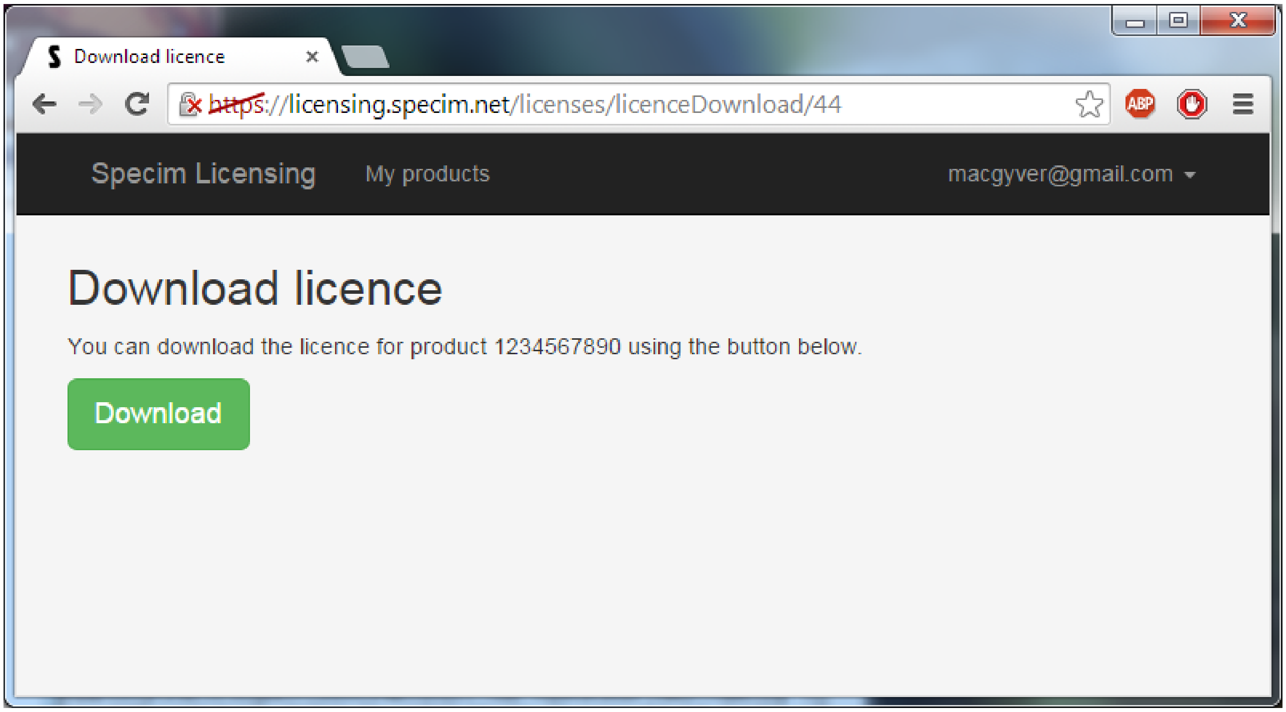


Figure 38: Licensing Portal - Downloading a License

12. The downloaded file is a ZIP file containing two files of the same name, but with different extension; .lic and .tms

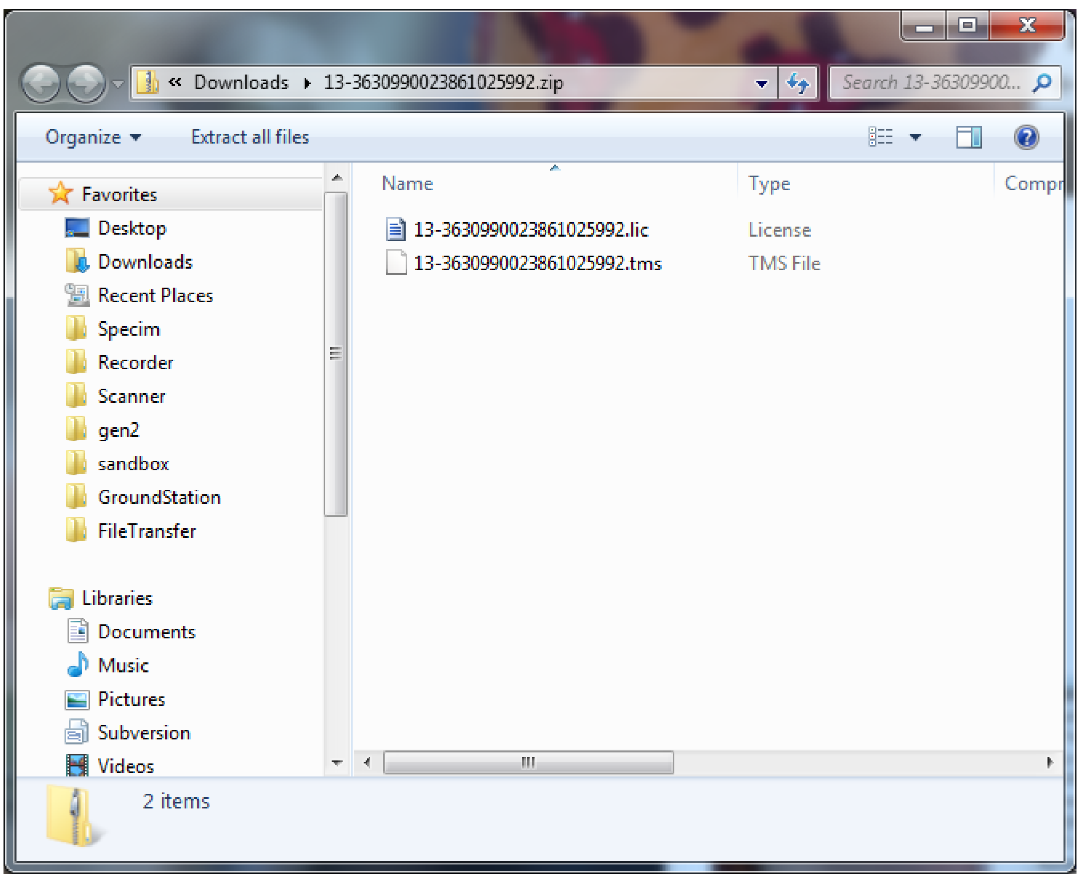


Figure 39: Licensing Portal - License Files

13. Unzip the archive to have the two files in the same folder.

14. Start the licensing wizard and select **Install a license you have received**.

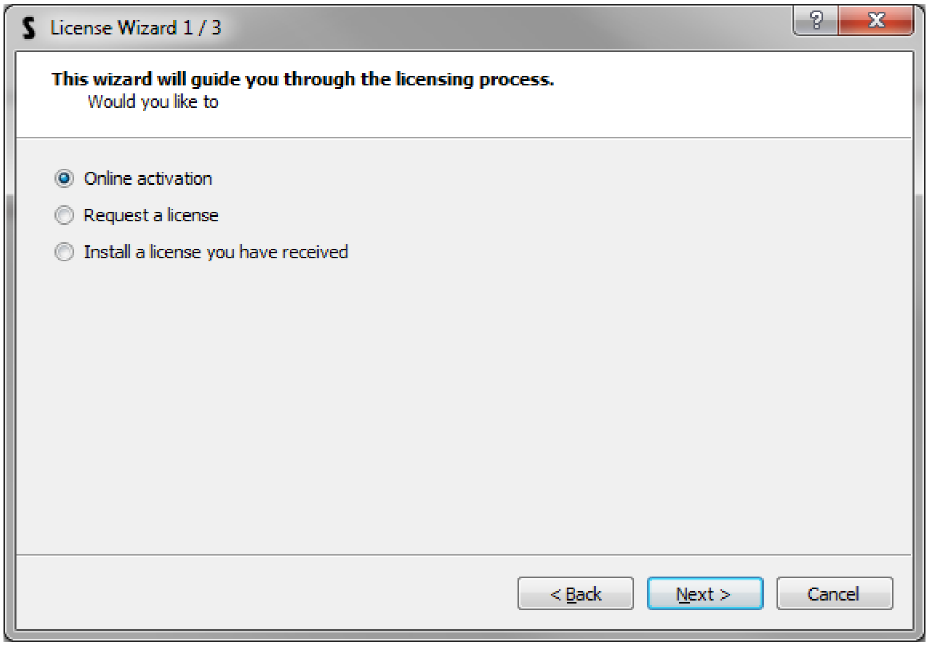


Figure 40: SpecSensor License Wizard

15. Press the **Next** button.

16. Click **Select** and select the license file with the .lic extension

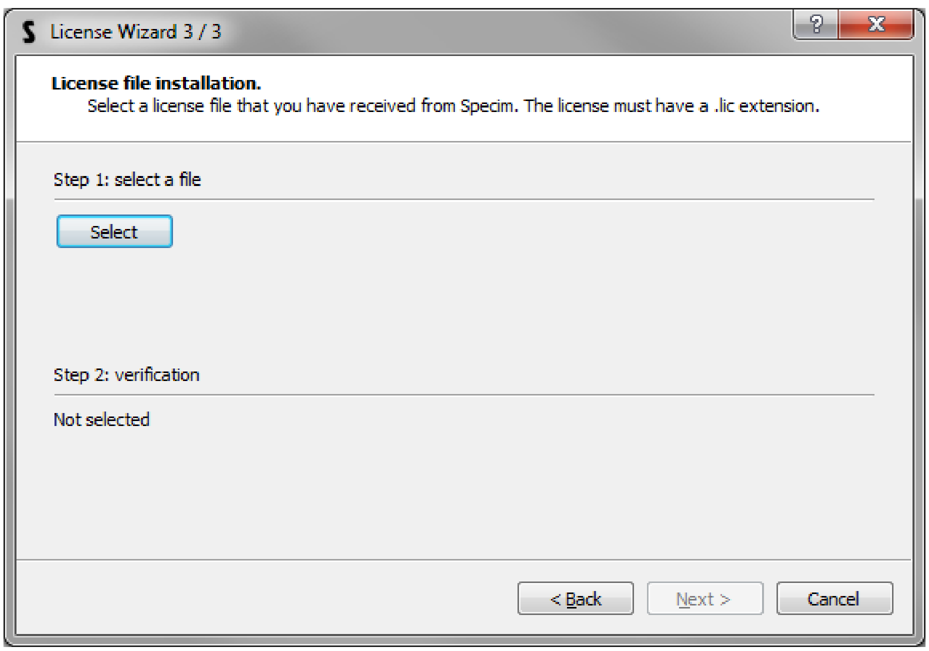


Figure 41: License Wizard - Manual Installation

17. Press the **Next** button.

18. If the license file is valid, the wizard will display a green **Valid license** text.

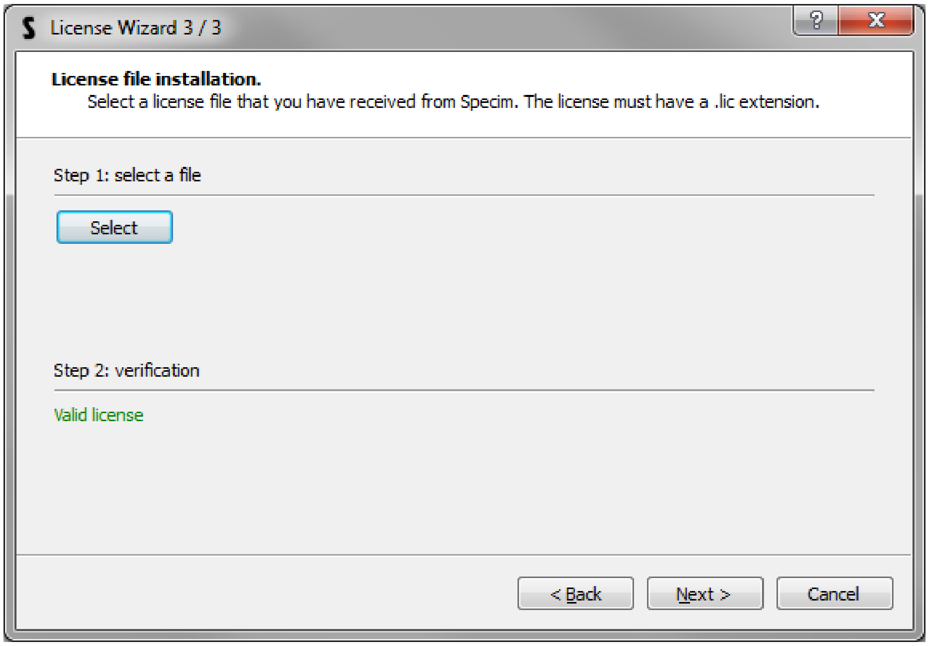


Figure 42: License Wizard - Valid License File

19. Finally, the wizard will thank you for choosing Specim products.

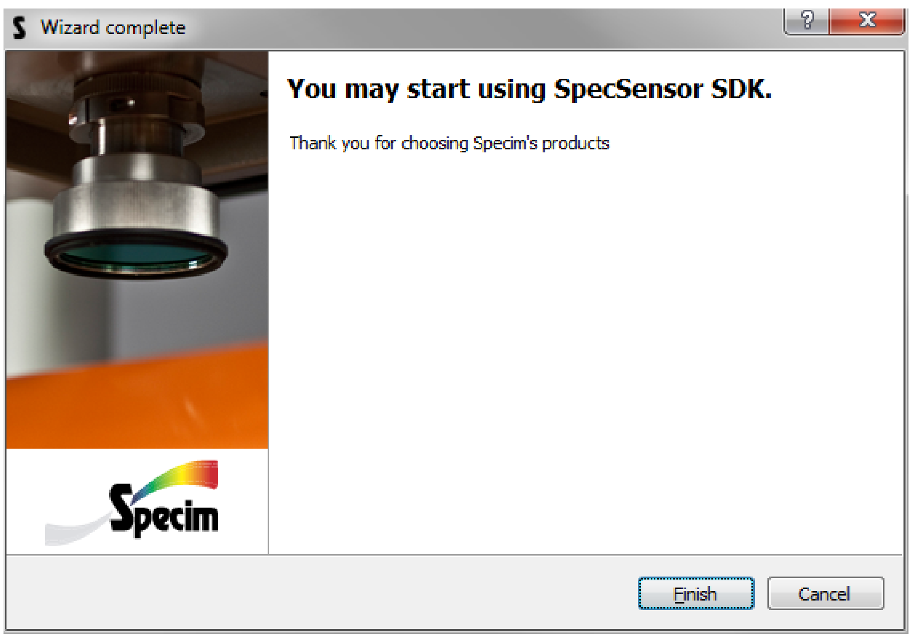


Figure 43: License Wizard - Final Page

20. Press the **Finish** button.

Defining the Logging Parameters

This section describes how to define the application's logging parameters.

Thanks to detailed error messages, the application is able to supply logging information for identifing various issues. The information also includes basic run log information.

To view logging parameters, select **Setup View** > **Logging**.

Proceed as follows:

1. Define the **Log folder**.

You can either enter the log folder path in the line edit *Widget*, or click the button next to it. A folder dialog will be opened, and you can select the folder.

Tip: It is a good idea to have a **dedicated** log **folder** for each application.

2. Select **Enable logging**.

   Defining the Log folder is not enough. To have the logging information **written to the log files**, check the **Enable logging** check box.
3. Select **Show log window**.

   When this check box is checked, a **logging widget** is visible below the content area. In this widget, you can see real-time logging information, and control the visibility of the messages, warnings, and errors. You can also define the order in which new messages are added: ascending or descending.

Defining the Sensor Parameters

This section describes how to define the sensor parameters.

Connect the sensor to the computer, install the drivers, and power up the sensor, before you start the application.

The application supports recording data for one or multiple local sensors. Each sensor in the application and workflows is referred to by an index, that is, Sensor 1, Sensor 2, Sensor 3, and so on. For each sensor:

- Specify the physical hardware sensor.
- Load some calibration data.
- Set some parameters for visualization, dataset naming, and so on.

To view the sensor parameters, select **Setup View** > **Sensor X**, where **X** refers to the sensor index.

Proceed as follows:

1. Select the **Sensor**.

   The first available parameter in the sensor settings group is the sensor selection combo box. This combo box enumerates all the available sensor profiles located in `C:\Users\Public\Documents\Specim`. These profiles are preconfigured by Specim, and provided with the delivery. They have a **.ssp** file extension.

   Select your sensor from the list, make sure it is powered up.
2. **Connect** to the sensor.

   To connect to the sensor, click the **Connect** button. If the connection is successful, the **Disconnect** button is activated, and the **Connect** button is deactivated. If the connection fails, check the Log window for more detailed information, and review the parameters.

   Note: Depending on the sensor, the connection and initialization can take from a few seconds up to dozens of seconds.
3. Select the sensor's **Calibration pack**.

   The calibration pack is an archive containing all the calibration data related to your sensor, including wavelength information, radiometric and geometric calibrations (if available), and so on.

   The calibration pack has a ***.scp** file extension. It is prepared by Specim, and provided with the delivery.

   Warning: It is possible to capture data without a valid calibration pack but all calibration information will default to the generated estimated values, which are incorrect for your sensor. The spectral information being missing, the data will be unusable.

   a) Open the file dialog selector by clicking the **Calibration pack** parameter.
   b) Select the calibration pack you have received.
   c) We recommend that you place the calibration packs in `C:\Users\Public\Documents\Specim\CalibrationPacks`.
4. Click the **Capture folder** parameter button to select the folder.

   The capture folder is the folder were the datasets will be recorded.

Important: The capture folder **cannot** be on the same hard drive as the operating system. This may cause dropped frames, because of insufficient writing speed. Use fast storage disks, such as *SSD*s on *SATA*3 interface.

5. Define the **Dataset prefix**.

The **Dataset prefix** parameter is part of the dataset name, and can help identify the sensor.

For example, a system could have a SWIR camera and a VNIR camera on the same computer. It is **not recommended**, but possible, to set the same capture folder for both sensors. Without a prefix, the dataset names would be identical, and the workflow would fail since each dataset must have a unique name.

Using the following dataset naming parameters:

- **Name generator**: counter with value **20**
- **Naming format**: **prefix**_name_counter
- **Dataset name**: hello
- SWIR sensor **Dataset prefix**: SWIR
- VNIR sensor **Dataset prefix**: VNIR

The full dataset names will be unique, and can be put in the same capture folder:

- SWIR_hello_20
- VNIR_hello_20

6. Select the *RGB* bands.

You can select the *RGB* bands using the **Red band**, **Green band** and **Blue band** parameters. Each choice in the band parameter displays the Band number as an index starting from 1, and the corresponding wavelength number.

Note: When available in the application, it is also possible to set the *RGB* bands from the Detector *Widget*, or the Spectral Plot *Widget*.

Defining the GPS parameters

This section describes how to define the GPS parameters.

Connect and power up the *GPS*, before you start the application.

The application supports recording data for one or multiple local *GPS*es, and to synchronize it with the sensors' data. Each *GPS* in the application and workflows are referred to by an index, that is, GPS 1, GPS 2, GPS 3, and so on. For each *GPS*, specify the physical hardware unit to connect to.

To view *GPS* parameters, select **Setup View** > **GPS X**, where **X** refers to the GPS index.

Proceed as follows:

1. Select the **GPS**.

The first available parameter in the **GPS** settings group is the **GPS** selection drop-down menu. This drop-down menu enumerates all the available *GPS* profiles located in `C:\Users\Public\Documents\Specim`. These profiles are preconfigured by Specim and provided with the delivery.

a) Select your *GPS* from the list, and make sure it is powered up.

2. Connect to the *GPS*.

a) Click the **Connect** button.

- If the connection is successful, the **Disconnect** button is activated, and the **Connect** button is deactivated.
- If the connection fails, check the Log window for more detailed information, and review the parameters.

Defining the Remote Control Server parameters

This section describes how to configure the remote control server parameters.

The application has a built-in remote control server, allowing another application using Lumo Remote SDK to take control over it.

The communication with this application, the server, is done through a network protocol, and the application using Lumo Remote SDK is a client.

To view remote control server parameters, select **Setup View** > **Remote Control Server**.

Proceed as follows:

1. Define the **Address**.

   The address can be the *URL* or the *IP* address of the computer running this application.

   Tip: Using the **0.0.0.0** *IP* address ensures that the server is listening on all the *IP* addresses available on the computer, even if multiple *NIC*s are used.
2. Define the **Port**.

   The port can be a number between 1 and 65535. The port must be available on the computer.
3. Start the remote control server and validate the parameters.

   A **Start** button and a **Stop** button are available below the **Port** parameters. Click the **Start** button to start the server. If the server is started , the **Start** button is deactivated, and the **Stop** button is activated. If the server does not start, see the **Log window** for a detailed description, and review the parameters.

   Note: The first time the remote control server is started, Windows will prompt you to make an exception rule for the port you have selected. If the server is started, but you cannot connect to this application from Lumo Remote SDK, review your firewall settings, and make sure that the port is open on all the required network types (Domain, Home/Work and Public) .

Defining the Remote Sensors Parameters

This section describes how to configure the remote sensors parameters.

The application is able to connect to remote sensors and control the parameters to accommodate setups where it is not possible to install multiple sensors on the same computer. The sensors can be a *HSI* or a *GPS*/*INS*.

This application, the master, connects and controls the parameters and workflows of a slave application. The slave application, Lumo Recorder, is running on a different computer where the sensors are attached.

The communication with this application, the server, and the slave Lumo Recorder is done via a network protocol and usesLumo Remote SDK.

The software is pre-configured to include **X** number of remote computers. Each remote computer can have several sensors.

The remote sensor parameters can be found from the Setup View in the **Remote computer X** settings group where **X** refers to the remote computer index.

Note: Make sure that the slave computer running Lumo Recorder is powered ON, that the sensors are powered ON, that Lumo Recorder is running and that at least its remote control server (see *Defining the Remote Control Server parameters*) is started.

Proceed as follows:

1. Define the **Address**.

   The address can be the *URL* or the *IP* address of the remote computer running Lumo Recorder .
2. Define the **Port**.

   The port can be a number between 1 and 65535. It must be the same than the one the slave Lumo Recorder's remote control server is listening on.
3. Connect to the remote computer and validate the parameters.

   Click the **Connect** button. If the connection is successful, the **Disconnect** button is activated, and the **Connect** button is deactivated. If the connection fails, check the Log window for more detailed information, and review the parameters.

   Note: If the connection is successful, the slave Lumo Recorder's sensors are listed below the remote computer **Connect** and **Disconnect** buttons.

4. Define the Sensor Parameters.

   Each remote *HSI* sensor is listed below the remote computer **Connect** and **Disconnect** buttons, and have the same parameters than a local sensor, i.e.:

   - sensors *SSP* selection
   - Connect button
   - Disconnect button
   - Calibration pack
   - Capture folder
   - Dataset prefix
   - Red band
   - Green band
   - Blue band

   See *Defining the Sensor Parameters*.

5. Define the GPS parameters

   Each remote *GPS* sensor is listed below the remote computer **Connect** and **Disconnect** buttons, and have the same parameters than a local *GPS*, i.e.:

   - *GPS SSP* selection
   - Connect button
   - Disconnect button

   See *Defining the GPS parameters*

Defining the Remote Control Server parameters

This section describes how to configure the remote control server parameters.

The application has a built-in remote control server, allowing another application using Lumo Remote SDK to take control over it.

The communication with this application, the server, is done through a network protocol, and the application using Lumo Remote SDK is a client.

To view remote control server parameters, select **Setup View** > **Remote Control Server**.

Proceed as follows:

1. Define the **Address**.

   The address can be the *URL* or the *IP* address of the computer running this application.

   Tip: Using the **0.0.0.0** *IP* address ensures that the server is listening on all the *IP* addresses available on the computer, even if multiple *NIC*s are used.

2. Define the **Port**.

   The port can be a number between 1 and 65535. The port must be available on the computer.

3. Start the remote control server and validate the parameters.

   A **Start** button and a **Stop** button are available below the **Port** parameters. Click the **Start** button to start the server. If the server is started , the **Start** button is deactivated, and the **Stop** button is activated. If the server does not start, see the **Log window** for a detailed description, and review the parameters.

   Note: The first time the remote control server is started, Windows will prompt you to make an exception rule for the port you have selected. If the server is started, but you cannot connect to this application from Lumo Remote SDK, review your firewall settings, and make sure that the port is open on all the required network types (Domain, Home/Work and Public) .

Defining the Task Manager Parameters

This section describes how to define the task manager parameters.

Lumo Task Manager must be started and the server must listen for incoming traffic.

Lumo Task Manager can be used to start, stop, restart or kill this application from a remote computer. To allow this, the application must register its process information to Lumo Task Manager.

The communication with Lumo Task Manager, the server, takes place through a network protocol, and this application is a client.

Define the network address and the port Lumo Task Manager is listening on.

To view **Task Manager** parameters, select **Setup View** > **Task Manager**.

Proceed as follows:

1. Define the **Address**.

   The **Address** can be a *URL* or an *IP* address. It must be the one Lumo Task Manager is listening on.
2. Define the **Port**.

   The port can be a number between 1 and 65535. It must be the one Lumo Task Manager is listening on.
3. Register the application, and validate the task manager parameters.

   Click the **Register** button. The registration is almost instant. If the **Task Manager** parameters are correct, the **Register** button will become inactive, and the **Unregister** button will become active.

   Additionally, a red or green bullet icon on the right hand side of the **Register** button will display the registration status:

   - Green stands for registered.
   - Red stands for unregistered.

   If the registration fails, review the Task Manager's parameters.

Defining the Thermal Calibrator Parameters

This section describes how to define the thermal calibrator parameters.

The thermal calibrator must connected to the computer and **powered up, before starting** the application.

The thermal calibrator is a device designed for the thermal sensors.

The thermal calibrator has two plates (blackbodies) with configurable temperatures. One blackbody will be measured as a cool reference. The other one, warmer, will be measured as a warm reference. A single controller for the two blackbodies regulates the temperatures.

The thermal calibrator also has a motor allowing the positioning of the blackbodies in front of the fore objective.

Both the controller and the motor must be configured and initialized successfully, to be able to use the thermal calibrator workflows.

To view thermal calibrator controller parameters, select **Thermal calibrator controller** > **Setup View**.

Proceed as follows:

1. Define the **IP address**.

   The blackbody controller's default *IP* address is **192.168.1.201**.

   Note: Do not change the default address, unless networking restrictions force you to.
2. Define the **Port**.

   The blackbody controller's default port is **2101**.

   Note: Do not change the default port, unless networking restrictions force you to.
3. Select the **Motor COM Port**.

   The motor COM ports are **auto detected** when the application starts. Select the thermal calibrators COM port from the combo box.

4. Connect to the thermal calibrator.

   A **Connect** button and a **Disconnect** button are available, below the **Motor COM Port** parameter. Click **Connect** to connect to the thermal calibrator, and initialize it. If the connection succeeds, the **Disconnect** button is activated, and the **Connect** button is deactivated. If the connection fails, check the Log window for more information, and review the thermal calibrator parameters.
5. Monitor the temperature stabilization.

   The blackbody controller can report if the temperatures are stabilized or not. The **T1 stabilized** and **T2 stabilized** reports available below **T2**, report if the blackbodies are stabilized. Also, the **T1 measured (˚C)** and **T2 measured (˚C)** reports display the measured blackbody temperatures.

   Note: When the blackbody controller is powered up, dozens of seconds are required until the temperatures become stabilized. If a blackbody temperature exceeds the tolerance of ± 0.02˚C from the set temperature, the blackbody will be marked as unstable.

   Warning: When the motor moves the blackbodies, their temperatures become unstable. The recording workflow for the cool and warm reference will allow a maximum of 30 seconds for the temperatures to stabilize again after a movement, otherwise the recording workflow will fail.

Defining the Dataset Naming Parameters

This section describes how to define the dataset naming parameters.

A dataset full name is created using three parameters:

- The dataset name often set in the metadata.
- The value of the name generator.
- The value of the sensor prefix (see *Defining the Sensor Parameters*).

To view dataset naming parameters, select **Setup View** > **Dataset**.

Proceed as follows:

1. Select a **Name generator**.

   Possible values are:

   - **Timestamp** — A timestamp is added to the dataset full name when the recroding workflow is started. When selected, the **Timestamp format** parameter is available. This is where you have to define the timestamp format. The following patterns will be replaced by their corresponding values. They can be used in any order.

     - `!` - UTC time
     - `%Y` - Full year
     - `%m` - Month
     - `%d` - Day of month
     - `%H` - Hour, using a 24-hour clock
     - `%I` - Hour, using a 12-hour clock
     - `%M` - Minute
     - `%S` - Second

     For example: `!%Y-%m-%d_%H-%M-%S` would give **2015-12-31_13-59-01**.

     Tip: To display a tooltip of the syntax, hover your mouse over the **Timestamp format** label for one second.
   - **Counter** — When selected, a counter value is added to the dataset full name. The counter **value** is **automatically incremented** at the end of the recording workflow. When selected, the **Counter value** setting is available. It contains the value that will be used at the next recording workflow.
   - **None**: If you select the **None** name generator, no name generator will be used.
2. Select a **Naming format**.

   The **Naming format** drop-down list lets you choose the position of the name generator in the dataset name. The choices are:

- **Timestamp Name generator** selected:
  - `prefix_name_timestamp`
  - `prefix_timestamp_name`
  - `timestamp_name_prefix`
  - `timestamp_prefix_name`
  - `name_prefix_timestamp`
  - `name_timestamp_prefix`
  - `timestamp_name`
  - `name_timestamp`
- **Counter Name generator** selected:
  - `prefix_name_counter`
  - `prefix_counter_name`
  - `counter_name_prefix`
  - `counter_prefix_name`
  - `name_prefix_counter`
  - `name_counter_prefix`
  - `name_counter counter_name`
- **None Name generator** selected:
  - `prefix_name`
  - `name_prefix`
  - `name`

Defining the General Parameters

This section describes how to define the general parameters.

The general parameters control a number of basic aspects in the application, and provide you access to the **About** dialog.

To view general parameters, select **Setup View** > **General**.

Proceed as follows:

1. Select **Minimize the application to system tray**.

   When enabled, the application icon is added to the system tray. If you click the main window's minimize button, the main window will be hidden. To show the main window again, double click the application icon in the system tray, or right click it and select show.

   Note: Each application can have a different set of options in the tray icon context menu.

2. Select **Show application in full screen mode (F11)**.

   When enabled, the application's main window will cover the entire screen. You will not be able to see the Windows start menu button, task bar, system tray or the application's title bar. You can toggle this option with the `F11` key.

3. Select **Start automatically when you log to this computer**.

   When enabled, **Start automatically when you log to this computer** will start the application when the user logs in. If the computer does not have login enabled, the application will start automatically when the computer boots up, since no user name nor password will be asked. Starting the application does not mean that the devices will be initialized.

4. Define **Startup delay (seconds)**

   This option defines the delay from application startup to automatic initialization, when the software is started automatically when logging on to the computer. Some of the computers subsystems may need some time to initialize properly. The necessary delay depends on the systems configuration and can only be determined experimentally. A delay of at least 10s is recommended with the Windows® operating system.

Warning: If the delay is too short, the application startup will fail.

Tip: If initialisation fails, restart the application (not the computer!) and verify that everything works. If everything works as it should, set a longer startup delay and reboot the computer to try again.

5. Select **Auto initialize when the application starts**.

   With **Auto initialize when the application starts** enabled, all the configured devices will automatically be connected and initialized, with the previous settings, when starting the software. If a device fails connecting, the auto-initialization procedure will continue, if possible.

   Tip: Enable this parameter only after you have fully configured your system.

6. Click the **About...** dialog option.

   The **About...** dialog will show the application name, version, and enumerate all the modules loaded from the application directory with their version. This information will help the Specim's support (support@specim.fi) team in case you have issues with your system.

   Tip: The **Contact support** link will open a new email containing all the binaries information. Please use it if you have issues with your system.

Selecting the Recording Workflow

This section describes how to select the workflow.

The application has an internal scripting engine controlling all the steps required to produce a dataset. The recording *Workflow*s consist of sequential steps, each step focusing on a specific task.

A workflow is defined for a specific sensor type and accessory, and can also control the content of the final dataset.

To view workflow selection parameters, select **Capture View** > **Workflow settings**.

Proceed as follows:

1. Select a **Workflow**.

   The workflows are:

   - Image
   - Dark image
   - Image and dark image
   - Image with GPS Data and Dark Image
   - Image, Dark Image and Thermal Images
   - Image with GPS Data, Dark Image and Thermal Images
   - Image with Embedded Dark Image
   - Image with GPS Data and Embedded Dark Image
   - Image with Embedded Dark Image and Thermal Images
   - Image with GPS Data, Embedded Dark Image and Thermal Images
   - Thermal Images
   - Thermal Images with Embedded Dark

   Note: The availability of the workflows depends on the connected devices. For example, workflows involving recording GPS data are only available when a GPS unit is connected, and so on.

2. Define the workflow parameters.

   The workflow parameters are specific for each workflow. See each workflow documentation page for more details:

   - *Image*
   - *Dark image*
   - *Image and dark image*
   - *Image with GPS Data and Dark Image*

- *Image, Dark Image and Thermal Images*
- *Image with GPS Data, Dark Image and Thermal Images*
- *Image with Embedded Dark Image*
- *Image with GPS Data and Embedded Dark Image*
- *Image with Embedded Dark Image and Thermal Images*
- *Image with GPS Data, Embedded Dark Image and Thermal Images*
- *Thermal Images*
- *Thermal Images with Embedded Dark*

Defining the File Transfer Parameters

This section describes how to define the file transfer parameters.

If the recorded datasets must be transferred to another computer, the recording workflows are able to automatically queue the datasets into Lumo FileTransfer Server, which will transfer the files.

To view file transfer parameters, select **Setup View** > **File Transfer**.

Proceed as follows:

1. Select **Enabled**.

   Enabling the file transfer parameter is a global parameter for all the recording workflows. When enabled, every successfully recorded dataset will be queued in Lumo FileTransfer Server.
2. Define the Lumo FileTransfer Server **Executable path**.

   The files will be queued to Lumo FileTransfer Server using the command line interface. To do that, the application must know the full path of the Lumo FileTransfer Server executable.

   a) Click the **Executable path** button.
   b) Select the Lumo FileTransfer Server executable.

      For example, `C:/Program Files/Specim/Lumo - FileTransferServer/1_2_2014_1160/FileTransferServer-x64.exe`.

Importing and Exporting the Settings

This section describes how to import and export the settings.

The application stores settings into an *XML* file. The settings can be imported and exported for backup and restore operation or sharing settings across different computers.

The import / export parameters can be found from the Setup View in the **Import/export settings** settings group.

Proceed as follows:

1. Click **Export settings to**

   When you click the **Export settings to** button, a save as type of dialog asks you to enter the file name for the exported settings file and the folder location. After entering this information, click the **Save** button.

   If the settings have been successfully written a confirmation dialog will prompt `Settings exported successfully.`
2. Click **Importing settings from**

   When you click the **Import settings from** button an open file type of dialog lets you select a settings file previously exported.

   Importing settings require the application to **restart**.

   After selecting the settings file, click on **Restart** button to continue with the import or **Cancel import** button to cancel the import.

   If you continue with the import **all the current settings will be lost**. After the application restart, the settings are now restored.

Warning: Importing settings will **overwrite** the current settings and they will be **lost**. Make sure to backup your settings before an import.

Adjusting the Sensor

Adjusting the Sensor Parameters

This section describes how to define the sensor parameters.

Controlling the Shutter

This section describes how to control the shutter.

The **Shutter** control will open and close the shutter. When the shutter shows **Opened**, the light goes through the instrument.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Shutter** parameter.
4. Open the shutter.

   Click the check box showing **Closed**. The check box now shows **Opened** and the light goes through the instrument.
5. Close the shutter.

   Click the check box showing **Opened**. The check box now shows **Closed** and the light does not go through the instrument.

Defining the Frame Rate

This section describes how to define the frame rate.

The **Frame rate** parameter controls the number of frames per second the sensor will output. The maximum frame rate is sensor-specific. The higher the frame rate, the more data will be recorded by the application.

The frame rate is in close relationship with the exposure time and readout time. These three parameters define the frame time. The sensor's minimum frame time defines the maximum frame rate.

The application is designed to provide a **constant frame rate**. Setting the frame rate may require changing the exposure time, if it exceeds the frame time, see *Figure 44: Frame Times*.

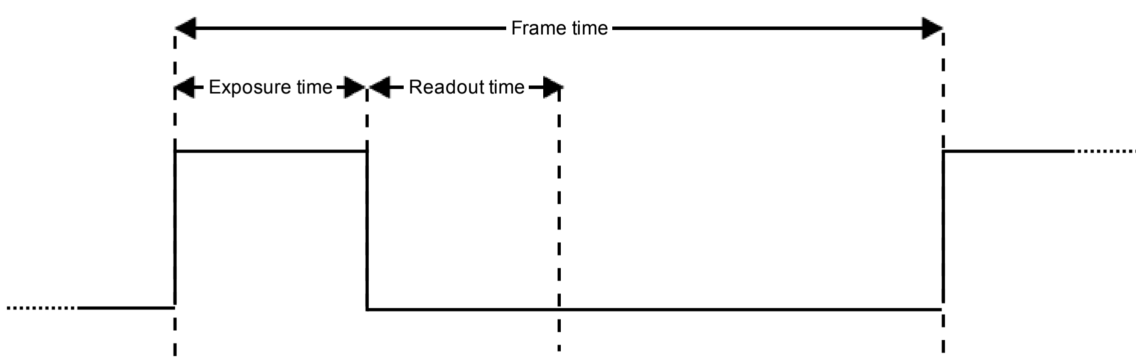


Figure 44: Frame Times

> **Tip:** When dealing with slow hard drives, it may not be possible to record all the data, and dropped frames will be reported. Lowering the frame rate can help to fix this problem, and return to zero dropped frames.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.

2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Frame rate** parameter.
4. Define the frame rate.

   You can change the value by:

   1. Entering the value using the numeric keys.
   2. Using the mouse scroll wheel.
   3. Using the keyboard up and down arrows.
5. Click **Apply**.

   Depending on the sensor, applying the new frame rate may be a very quick action, or take dozens of seconds. If the exposure time value was decreased, the parameter will have a red outline to warn you of the change. Click **Apply** again to get rid of the outline.

Defining the Exposure Time

This section describes how to define the exposure time.

The **Exposure time** parameter controls the length of time the detector will integrate the light. The maximum exposure time is sensor-specific. The higher the exposure time, the brighter the image, and the better the signal to noise ratio.

The exposure time is in close relationship with the frame rate. The frame rate limits the maximum exposure time value. See *Defining the Frame Rate*.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Exposure time** parameter.
4. Define the exposure time.

   You can change the value by:

   1. Entering the value using the numeric keys.
   2. Using the mouse scroll wheel.
   3. Using the keyboards up and down arrows.
5. Click **Apply**.

   Depending on the sensor, applying the new exposure time may be a very quick action, or take dozens of seconds.

Defining the Binning

This section describes how to define the binning.

Spectral binning controls how many rows of the detector will be binned. **Spatial binning** controls how many columns of the detector will be binned. The binning can be summing or averaging, depending on the sensor.

Applying binning reduces the original data size by the spectral and spatial binning factor. If the binning is performed in the camera, less data will have to be transferred and, for certain sensors, it will be possible to achieve a higher frame rate.

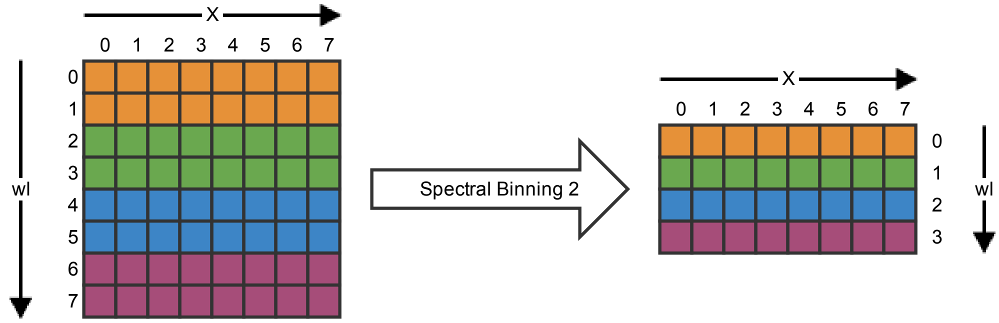


Figure 45: Spectral Binning by 2

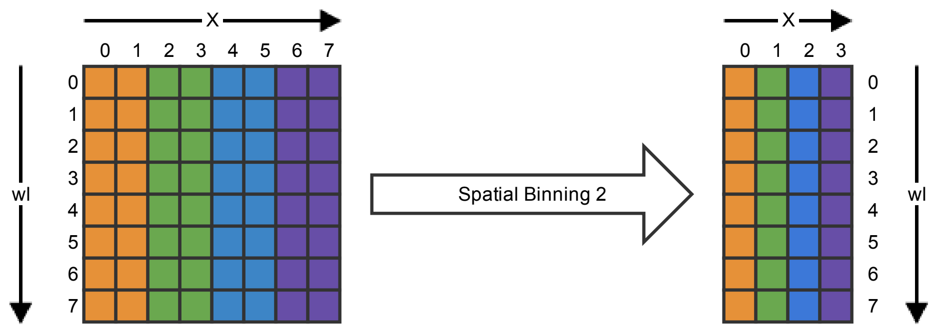


Figure 46: Spatial Binning by 2

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Spectral Binning** parameter, and select a binning value.
4. Locate the **Spatial Binning** parameter, and select a binning value.
5. Click **Apply**.

   Depending on the sensor, applying the new binning parameters may be a very quick action, or take dozens of seconds. Decreasing binning values can decrease the frame rate, if the sensor cannot handle the throughput with larger frame dimensions. If the frame rate value was changed, the parameter will have a red outline to warn you of the change. Click **Apply** again to get rid of the outline.

Enabling the Bad Pixel Replacement

This section describes how to enable the bad pixel replacement.

When enabled, the bad pixels of the sensor will be replaced with values estimated using the neighboring pixels. The implementation, and which neighboring pixels are used, is sensor specific.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **BPR** parameter.
4. Enable *BPR* by checking the check box.
5. Click **Apply**.

Depending on the sensor, applying the new *BPR* parameter may be a very quick action, or take dozens of seconds.

Defining the Non Uniformity Correction Parameters

This section describes how to define the non uniformity correction parameters.

The **NUC** parameter defines the *NUC* table that will be applied. A *NUC* table is dedicated to a certain exposure time range.

You can choose to set the parameters manually or automatically. If you choose to set the **NUC** parameter values manually, you must know the table to set.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Auto NUC** parameter.
4. To let the software set the **NUC** parameter automatically, check the **Auto NUC** checkbox, and click **Apply**.

   With **Auto NUC** enabled, the **NUC** parameter is disabled, but still displays the **NUC** value set by the application.
5. To manually set the **NUC** parameter, uncheck the **Auto NUC** checkbox.

   The **NUC** parameter is enabled.
6. Select the **NUC** value from the combo box.
7. Click **Apply**.

   Depending on the sensor, applying the new *NUC* parameter may be a very quick action, or take dozens of seconds.

Defining the Camera Trigger Mode

This section describes how to define the camera trigger mode.

The **Trigger mode** specifies if the sensor will use the Internal or External triggering method:

- Internal triggering method is the default setting for spectral cameras.
- External triggering is used when the sensor data must be synchronized with another device, such as *GPS*, or a second sensor. External triggering must always be used on airborne systems.

Proceed as follows:

1. Select **Capture View** or **Adjust View**.
2. Locate the sensor widget on the left-hand side panel.

   The sensor widget has the name of the sensor selected in sensor settings. See *Defining the Sensor Parameters*.
3. Locate the **Trigger mode** parameter.
4. Select the **Trigger mode** from the combo box.
5. Click **Apply**.

   Depending on the sensor, applying the new **Trigger mode** parameter may be a very quick action, or take dozens of seconds.

Defining the Image Correction Parameters

This section describes how to define the image correction parameters.

Image correction is used to improve the real-time visualisation of data shown in the Detector *Widget* and in the Waterfall *Widget*. The recorded data is unaffeected by image correction options.

The Image correction *Widget* is located in the right side pane of the Adjust View.

Note: The parameters are only visible when the sensor is connected. The sensor parameters need to be defined prior to performing these steps to get good image correction results.

Proceed as follows:

1. Select **Enabled**

   When **Enabled** is **True**, image correction is applied.
2. Select the **Dark Subtraction** parameter

   Pixels may not be entirely dark, even have when no light reaches the sensor. This is called the dark current level. To correct the image, dark current is subtracted from the data. The dark current reference can be acquired from the sensor or defined by the user with one of the options below:

   - The **Shutter** option has a dedicated workflow. The sensor's shutter closes and 100 frames are acquired. The frames area averaged to a single dark reference frame, and the shutter re-opens. Each pixel has its own measured dark current value.
   - Using the **Constant** option, a single dark current value is used, which is defined in the **Dark constant** field.
   - The **Thermal Calibrator** setting has a dedicated workflow. Thermal calibrators are used with thermal sensors. The thermal calibrator places the cool reference under the fore objective and acquires 100 frames. The frames are averaged to a single dark reference frame and the thermal calibrator retraces to allow imaging again. Each pixel in the frame has its own measured dark current value.
   - The **Manual** option has a dedicated workflow. The operator is asked to obstruct the fore objective. Next 100 frames are acquired and averaged to one single dark reference frame. Each pixel has its own measured dark current value.
   - The **Scanner** option has a dedicated workflow. The scanner moves to the dark reference start position and 100 frames are acquired. The frames are averaged to a single dark reference frame and the scanner retraces. Each pixel has its own measured dark current value.
3. Press **Run dark**

   This will acquire and apply the captured or defined dark current level.
4. Select the **White reference** parameter

   The white reference specifies the maximum pixel value for all pixels. Together with the dark reference, the white reference is used for *Normalization*. The white reference can be acquired from the sensor or defined by the user with one of the options below.

   - Using the **Constant** option, the constant defined in the **White constant** field will be used as the white reference.
   - The **Manual** option has a dedicated workflow. The operator is asked to place a white reference object under the fore objective. Next 100 frames are acquired and averaged to get a single white reference frame. Each pixel in the frame has its own measured maximum value.
   - The **Thermal Calibrator** option has a dedicated workflow. Thermal calibrators are used with thermal sensors. The thermal calibrator places the warm reference under the fore objective and 100 frames are acquired. The frames are averaged to a single white reference frame and the thermal calibrator retraces to allow imaging again. Each pixel in the frame has its own measured maximum value.
   - The **Scanner** option has a dedicated workflow. The scanner moves to the white reference start position and 100 frames are acquired. The frames are averaged to a single white reference frame and the scanner retraces. Each pixel in the frame has its own measured maximum value.
   - The **Radiometric** option has a dedicated workflow. The radiometric coefficients are extracted from the calibration pack. Each pixel in the frame has its own coefficient value.
5. Press **Run white**

   This will acquire and apply the captured or defined white reference level.

Performing the Focusing

This section describes how to perform the focusing.

The best focus is the least bad focus across all the different wavelengths, which makes focusing difficult by just looking at the detector *Widget*. This application provides a software tool to help with the focusing task.

The focusing tool is located in the right side pane of the Adjust View. The focusing rectangle is drawn in the Detector *Widget*, also located in the Adjust View.

Proceed as follows:

1. Place a focusing grid under the fore objective.
2. Select the focus region mode

   Right-click the Detector *Widget* and from the menu choose **Select Focus region**.
3. Paint the focus region rectangle.

   On top of the Detector *Widget*, press and hold the left mouse button, while dragging the mouse to define the focus region. Release the mouse button when satisfied with the region.
4. Click the **Run** button.
5. Find the best focus.

   Rotate the fore object to achieve an image or spatial reference with sharp contrast. The best focus value in percentage is the one closest to 100%.
6. Find the worst focus.

   Rotate in one direction to find the worst focus. The focus percentage should decrease.
7. Find the best focus again.

   Rotate to the opposite direction than you just did to get better focus percentages. Rotate until the focus percentage starts to decrease again.
8. Set the best focus.

   At this point the software knows the best focus value. Match it by slowly rotating the lens to the best focus position.

Defining the Aspect Ratio

This section describes how to define the aspect ratio.

Push-broom type imagers record one line of image at a time, when considering a standard 2D image. The X dimension is recorded for every frame, and the Y dimension is created by moving the target or the sensor, depending on the system type.

To obtain square pixels with regard to X and Y dimensions, define the speed of the target, or the frame rate.

You can also possibly configure the aspect ratio by imaging a square or a circle, and controlling the shape in the final image. *Figure 47: Aspect Ratio* illustrates the effect of a too fast, a too slow, and the correct speed, or frame rate, on the final image.

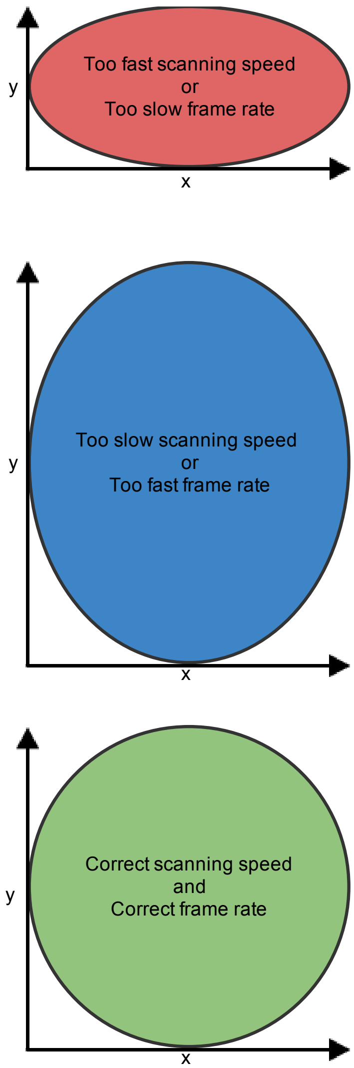


Figure 47: Aspect Ratio

With airborne systems, you cannot check the aspect ratio in this way. It must be calculated using the optics, sensor properties, and the altitude. To obtain square pixels, change the frame rate or the plane speed.

Manually

This section describes how to define the aspect ratio manually.

Square pixels for the scanned image can be achieved manually by either changing the scanning speed or the frame rate.

The steps below assume you are able to image an object.

Proceed as follows:

1. Image an object of a known size, with a simple geometry, such as a circle or a square.
   - The larger the object the better, as long as the object is within the field of view of the sensor.
   - The object should be something that can be easily seen by the sensor in use.
2. Open the recorded image in a viewer.
3. Measure the width (`X`) and the height (`Y`) of the object in pixels in the image.
4. Calculate `multiplier = X / Y`.

- If the multiplier is smaller than 1, either the scanning speed is too fast or the frame rate is too slow.
- If the multiplier is greater than 1, the scanning speed is too slow or the frame rate is too fast.
- The closer the multiplier is to 1, the closer you are to having perfect square pixels.
- If you want to change the frame rate of the sensor, multiply the current frame rate with the calculated multiplier, and set the result as the new frame rate.
- If you want to change the scanning speed, divide the current scanning speed with the calculated multiplier, and set the result as the new scanning speed.

5. Repeat the steps above from step 1, until the multiplier is close enough to 1.

Using the Field of View of the Sensor

This section describes how to define the aspect ratio using the field of view of the sensor.

Square pixels for the scanned image can be achieved by defining the field of view of the sensor.

In rotary and mirror scanner setups the field of view is represented in degrees and it refers to the horizontal field of view angle that the sensor optics see. This value differs between different types of lenses and it should be provided with the specifications on the lens.

In linear scanner setups the field of view is usually represented in millimeters and it refers to the horizontal field of view at the target position. The field of view value can be measured from the target by placing a long enough measuring stick horizontally under the target area and scanning it.

This method requires a valid calibration pack to be loaded for the sensor.

Proceed as follows:

1. Select the 'Field of view' method from the application.
2. Change the 'Field of view' value from the UI.
   - Whenever the field of view or the frame rate of the sensor changes, the scanning speed is automatically adjusted using the formula: `Scanning speed = (FOV / horizontal_pixels) * Frame rate`
3. Confirm by scanning a round object.

Using the Distance to the Target

This section describes how to define the aspect ratio using a known distance to the target.

Square pixels for the scanned image can be achieved by defining the distance to the target in millimeters. This method is only valid for linear scanner setups.

Requires a valid calibration pack to be loaded for the sensor.

Proceed as follows:

1. Select the 'Working distance' method from the application.
2. Measure the working distance and change it from the UI.
   - Whenever the distance to the target or the frame rate of the sensor changes, the scanning speed is automatically adjusted using the formula: `Scanning speed = (Detector pixel size * Spatial binning * Spectrograph magnification * (Distance - Entrance pupil position) * Frame rate) / Focal length`
3. Confirm by scanning a round object.

Defining the QA/QC Parameters

This section describes how to define the *QA/QC* parameters.

The *QA/QC* routine is an automated method to check the recorded data quality. The *QA/QC* routine is performed in the recording workflows when the dataset has been fully recorded. Three options are available:

- Disabled — The *QA/QC* will not be performed.
- Internal — See *Internal QA/QC Parameters*.

- External — See *External QA/QC Parameters*.

Internal QA/QC Parameters

This section describes how to define the internal *QA/QC* parameters.

The **Internal** *QA/QC* option provides an automated *QA/QC* procedure for checking the dataset integrity, and the white and dark references quality.

The dataset integrity checks for the following criteria:

- All the files listed on the dataset manifest exist on the disk.
- All the *RAW* files have a matching *HDR* file.
- All the *HDR* files have the required keywords, and the information is valid. The following keywords are checked for validity:

Table 1: Valid ENVI HDR criteria

Keyword	Criteria
`samples`	Must **exist**.
`bands`	Must **exist**.
`lines`	Must **exist**.
`errors`	Must be **none**.
`interleave`	Must be **bil**.
`data type`	Must be **12**.
`default bands`	Must be **within range** of the `bands` keyword information.
`himg`	Must match the `samples` keyword information.
`vimg`	Must match the `bands` keyword information..
`hroi`	Must match the `samples` keyword information.
`vroi`	Must match the `bands` keyword information.
`fps`	Must **exist**.
`tint`	Must **exist**.
`binning`	Must **exist**.
`sensorid`	Must **exist**.
`Wavelength`	The number of elements must **match** the `bands` keyword information.
`fwhm`	The number of elements must **match** the `bands` keyword information.

- All the *RAW* files sizes match the information from their associated *HDR* files.

The **internal** *QA/QC* uses **correlation** and *EMD*, to calculate the validity of each reference.

First, the white and dark references are averaged into one single frame per reference. These averaged frames are used for the **correlation** and *EMD* checks.

The **correlation** calculates the similarity between two probability distribution functions; histograms in this case. The similarity is measured by dividing the *ROI* horizontally from the middle, and taking individual correlation values of each side. This will allow us to detect **distribution anomalies**, such as **broken lamps**, and so on.

The *EMD* calculates the **minimum cost** of turning one probability distribution function into another. It is calculated over the whole averaged frame. In practice, it checks if the recorded data is close enough to the reference.

With the **Internal** *QA/QC* method selected, the parameters below are available, and must be configured:

- White reference
- Dark reference
- Timeout (ms)
- Correlation threshold
- *EMD* threshold

Proceed as follows:

1. Select the **White reference**.

   The **White reference** parameter allows you to choose an already acquired white reference file, which is known to be a good reference. This reference will be compared with the newly recorded white references within the **correlation** and *EMD* checks.

   a) Click the **White reference** parameter button to open the file dialog selector.
   b) Select the white reference file.
2. Select the **Dark reference**.

   The **Dark reference** parameter allows you to choose an already acquired dark reference file, which is known to be a good reference. This reference will be compared with the newly recorded dark references within the **correlation** and *EMD* checks.

   a) Click the **Dark reference** parameter button to open the file dialog selector.
   b) Select the dark reference file.
3. Define the **Timeout (ms)**.

   The **Timeout (ms)** parameter allows you to abort the recording workflow in case the computation, abnormally, takes too much time. The timeout is specified in milliseconds, and -1 refers to inifinite (that is, no timeout).

   a) Define the **Timeout (ms)** value in the spin box of the parameter.
4. Define the **Correlation threshold**.

   The **correlation** algorithm generates a value between -1 and 1, where **1** is a **perfect match**. Usually, a **Correlation threshold** of **0.8** is a **good** starting point value.

   a) Define the **Correlation threshold** value in the spin box of the parameter.
5. Define the ***EMD* threshold**.

   The *EMD* algorithm generates a value between 0 and infinity, where **0** is a **perfect match**. Usually, a ***EMD* threshold** of **5** is a good starting point value.

   a) Define the ***EMD* threshold** value in the spin box of the parameter.

The **internal** *QA/QC* routine generates one of the following errors:

Table 2: *QA/QC* error codes

Error name and code	Description
`siNoError (0)`	Successful *QA/QC* check.
`siQAQCDarkerThanReference` **(-1700)**	The newly recorded reference is too dark.
`siQAQCBrighterThanReference` **(-1701)**	The newly recorded reference is too bright.
`siQAQCUnevenDistribution` **(-1702)**	The newly recorded reference has uneven distribution. **Tip:** Check for shutter malfunction, broken illumination lamps or dirt on the white reference.
`siQAQCIncompatible` **(-1703)**	The newly recorded reference is incompatible with the file set in **White reference** or **Dark reference**. **Important:** The files must have the same `samples`, `bands` and `binning` values.

External QA/QC Parameters

This section describes how to define the external *QA/QC* parameters.

The **External** *QA/QC* method provides an interface for calling an external process, which is responsible for the *QA/QC*. The result can be reported back to this application as passed or failed, using a shared memory file.

This method provides flexibility for customers who want to control the *QA/QC*, while using the standard recording workflows.

With the **External** *QA/QC* method selected, the parameters below are available:

- Execute
- Start in
- Parameters
- Wait
- Timeout (ms)
- Get return value

Tip: You can use the *QA/QC* calling interface to execute a process not related to *QA/QC*. This can be helpful when integrating the software with another system.

Proceed as follows:

1. Define the **Execute** parameter.

   The External *QA/QC* method uses a uses a command line interface to call the external application. The external application's full path is required.

   The **Execute** setting allows you to select the executable path of the*QA/QC* routine using an open file type dialog.
2. Define the **Start in** parameter.

   This parameter allows you to select the start in directory for the executable path. This setting is not mandatory in most cases, and can be left blank.
3. Define the **Parameters** parameter.

   This parameter is used to specify the command line argument format to be passed to the *QA/QC* executable. Any parameters can be defined as plain text, but the patterns below will be replaced:

   - `%n`: is replaced by the manifest full path.
   - `%f`: is replaced by the manifest file name.
   - `%e`: is replaced by the manifest file extension .
   - `%d`: is replaced by the manifest directory path.
   - `%D`: is replaced by the manifest directory name (equals to dataset name).
   - `%s`: is replaced by the content of the Start in variable.
   - `%mmf`: is replaced by the path of the shared memory mapped file (*MMF*) used to retrieve the result of the external *QA/QC*.

   For example, if a dataset was recorded in `D:\data\test_dataset`, the value of the parameters above would be:

   - `%n`: `D:\data\test_dataset\manifest.xml`
   - `%f`: `manifest`
   - `%e`: `xml`
   - `%d`: `D:\data\test_dataset`
   - `%D`: `test_dataset`
   - `%s`: `D:\data\test_dataset\manifest.xml`
   - `%mmf`: `SPECIM_Workflow_QAQC_test_dataset_push_mmf`
4. Enable the **Wait** parameter.

   This parameter specifies if the acquisition workflow needs to wait for the external process to exit. If disabled, the external application process will be started detached, and this application will not monitor the execution.

5. Define the **Timeout (ms)** parameter.

   This parameter defines how long the application should wait for the *QA/QC* process to complete. The time is specified in milliseconds. A timeout of -1 corresponds to no timeout.

6. Enable the **Get return value** parameter.

   This parameter specifies if the acquisition software should read the QA/QC routine result from the shared memory mapped file, which is used to report if the QA/QC passes or fails. The external routine must write either `1` or `true` in ASCII, to report a success. Any other content is considered as failed.

   This application is responsible for creating and deleting the shared memory mapped file. The external application is responsible for opening the shared file from the path given by `%mmf`, writing `1` or `true`, and closing the file.

Operating Lumo Recorder

This section describes how to operate Lumo Recorder.

Monitoring the Traffic Lights

This section describes how to monitor the traffic lights.

The traffic lights display information about the current system status. The traffic lights include hardware status information, received data such as GPS coordinates, when a GPS sensor is connected, undefined application parameters, and so on.

The traffic lights display three levels:

- OK -- This level indicates that everything is all right. The backgroud color is **green**.
- Warning -- This level indicates that the warning threshold has been exceeded, but the condition is not an error yet. The background color is **orange**.
- Error -- This level indicates a clear error or a condition within an unacceptable range. The background color is **red**.

The fields visible in the traffic lights depend on the hardware used, and the system configuration. The fields shown can include e.g. the sensor's temperatures, the latest *GPS* coordinates, message about parameters not set, and so on.

The fields can be configured to be only visible on certain warning level.

Proceed as follows:

Ensure no red boxes are shown in the traffic lights.

Recording Data

This section describes how to record data manually.

The controls for starting and stopping recording are located at the top of the Adjust View and Capture View's right-hand side pane. The **Record** button will start acquisition, and the **Stop** button will either stop or abort the acquisition, depending on which **Capture mode** mode is selected:

- In the free mode, the acquisition will continue until the **stop** button is pressed.
- In the scripted mode, the acquisition will be stopped automatically when certain conditions are met, for example, the desired number of frames has been acquired. Clicking the **stop** button will abort the recording workflow.

Proceed as follows:

1. Define the dataset naming parameters.

   See *Defining the Dataset Naming Parameters*.
2. Select the **Workflow**.

   Also set all workflow related options, as described in *Selecting the Recording Workflow*.
3. Define the **Metadata** options.
4. Press the **Record** button.
5. Wait for data to be recorded.
6. End the recording.

   If the capture mode is free, press the **Stop** button. In the scripted mode, acquisition is ended automatically.

   Note: A message box will be shown for a few seconds after data acquisition ends, indicating if the acquisition was successful. The acquisition can fail if there were, for example, too many dropped frames during the acquisition, or if there was insufficient disk space.

Workflows

This section describes the available workflows.

Image

The **Image** recording workflow will create a dataset which will contain the image data, metadata information, manifest and thumbnail.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Record the image files
    - Open the shutter
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the hdr file.
3. Create the metadata.
4. Create the image thumbnail.
5. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
6. Create the manifest and close the dataset.
7. Queue the dataset in Lumo FileTransfer Server
8. Update the metadata if required
9. Update the name generator if required

.

Dark image

The **Dark image** recording workflow will create a dataset which will contain the dark image data, metadata information, manifest and thumbnail.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Record the image files
    - Close the shutter
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Open the shutter
    - Create the hdr file.
3. Create the metadata.
4. Create the image thumbnail.
5. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
6. Create the manifest and close the dataset.
7. Queue the dataset in Lumo FileTransfer Server
8. Update the metadata if required
9. Update the name generator if required

Image and dark image

The **Image and dark image** recording workflow will create a dataset which will contain the image data, dark image data, metadata information, manifest and thumbnail.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Record the image files
    - Open the shutter
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the image hdr file.
3. Record the dark image files.
    - Close the shutter
    - Record the dark image raw file until all scripted frame got recorded.
    - Open the shutter
    - Create the dark image hdr file.
4. Create the metadata.
5. Create the image thumbnail.
6. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
7. Create the manifest and close the dataset.
8. Queue the dataset in Lumo FileTransfer Server
9. Update the metadata if required
10. Update the name generator if required

Image with GPS Data and Dark Image

The **Image with GPS Data and Dark Image** recording workflow will create a dataset which will contain the image data, dark image data, *GPS*, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for airborne systems.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Start recording the navigation data
3. Wait for the start delay
4. Mask the triggers
5. Wait for the first PPS to star the image data recording
6. Record the image files
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the image hdr file.
7. Wait for the stop delay
8. Stop recording the navigation data
9. Record the dark image files.
    - Close the shutter

- Record the dark image raw file until all scripted frame got recorded.
- Open the shutter
- Create the dark image hdr file.

10. Create the metadata.
11. Create the image thumbnail.
12. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
13. Create the manifest and close the dataset.
14. Queue the dataset in Lumo FileTransfer Server
15. Update the metadata if required
16. Update the name generator if required

Image, Dark Image and Thermal Images

The **Image, Dark Image and Thermal Images** recording workflow will create a dataset which will contain the image data, dark image data, thermal images, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors.

The workflow steps are:

1. Open the dataset
- Save the current time.
- Create the dataset name.
- Create the dataset folder structure.

2. Record the image files
- Open the shutter.
- Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
- Create the image hdr file.

3. Record the dark image files.
- Close the shutter.
- Record the dark image raw file until all scripted frame got recorded.
- Open the shutter.
- Create the dark image hdr file.

4. Record the cool (T1) thermal image
- Move the T1 blackbody under the fore objective.
- Wait for T1 temperature to stabilize.
- Record the T1 image raw file until all scripted frame got recorded.
- Create the T1 image hdr file.

5. Record the warm (T2) thermal image
- Move the T2 blackbody under the fore objective.
- Wait for T2 temperature to stabilize
- Record the T2 image raw file until all scripted frame got recorded.
- Create the T2 image hdr file.

6. Home the blackbodies to allow imaging the scene again.
7. Create the metadata.
8. Create the image thumbnail.
9. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
10. Create the manifest and close the dataset.
11. Queue the dataset in Lumo FileTransfer Server
12. Update the metadata if required
13. Update the name generator if required

Image with GPS Data, Dark Image and Thermal Images

The **Image with GPS Data, Dark Image and Thermal Images** recording workflow will create a dataset which will contain the image data, dark image data, thermal images, *GPS*, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors in airborne systems.

The workflow steps are:

1. Open the dataset
   - Save the current time.
   - Create the dataset name.
   - Create the dataset folder structure.
2. Start recording the navigation data
3. Wait for the start delay
4. Mask the triggers
5. Wait for the first PPS to star the image data recording
6. Record the image files
   - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
   - Create the image hdr file.
7. Wait for the stop delay.
8. Stop recording the navigation data.
9. Record the dark image files.
   - Close the shutter
   - Record the dark image raw file until all scripted frame got recorded.
   - Open the shutter
   - Create the dark image hdr file.
10. Record the cool (T1) thermal image
    - Move the T1 blackbody under the fore objective.
    - Wait for T1 temperature to stabilize.
    - Record the T1 image raw file until all scripted frame got recorded.
    - Create the T1 image hdr file.
11. Record the warm (T2) thermal image
    - Move the T2 blackbody under the fore objective.
    - Wait for T2 temperature to stabilize
    - Record the T2 image raw file until all scripted frame got recorded.
    - Create the T2 image hdr file.
12. Home the blackbodies to allow imaging the scene again.
13. Create the metadata.
14. Create the image thumbnail.
15. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
16. Create the manifest and close the dataset.
17. Queue the dataset in Lumo FileTransfer Server
18. Update the metadata if required
19. Update the name generator if required

.

Image with Embedded Dark Image

The **Image with Embedded Dark Image** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, metadata information, manifest and thumbnail.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Record the image files
    - Open the shutter
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the image hdr file.
3. Record the dark data.
    - Re-open the image file
    - Close the shutter
    - Record the dark data at the end of the image raw file until all scripted frame got recorded.
    - Open the shutter
    - Update the image hdr file.
4. Create the metadata.
5. Create the image thumbnail.
6. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
7. Create the manifest and close the dataset.
8. Queue the dataset in Lumo FileTransfer Server
9. Update the metadata if required
10. Update the name generator if required

Image with GPS Data and Embedded Dark Image

The **Image with GPS Data and Embedded Dark Image** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, *GPS*, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for airborne systems.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Start recording the navigation data
3. Wait for the start delay
4. Mask the triggers
5. Wait for the first PPS to star the image data recording
6. Record the image files
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the image hdr file.
7. Wait for the stop delay
8. Stop recording the navigation data
9. Record the dark data.
    - Re-open the image file
    - Close the shutter
    - Record the dark data at the end of the image raw file until all scripted frame got recorded.

- Open the shutter
- Update the image hdr file.

10. Create the metadata.
11. Create the image thumbnail.
12. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
13. Create the manifest and close the dataset.
14. Queue the dataset in Lumo FileTransfer Server
15. Update the metadata if required
16. Update the name generator if required

Image with Embedded Dark Image and Thermal Images

The **Image with Embedded Dark Image and Thermal Images** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, thermal images, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors.

The workflow steps are:

1. Open the dataset
   - Save the current time.
   - Create the dataset name.
   - Create the dataset folder structure.
2. Record the image files
   - Open the shutter.
   - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
   - Create the image hdr file.
3. Record the dark data.
   - Re-open the image file
   - Close the shutter
   - Record the dark data at the end of the image raw file until all scripted frame got recorded.
   - Open the shutter
   - Update the image hdr file.
4. Record the cool (T1) thermal image
   - Move the T1 blackbody under the fore objective.
   - Wait for T1 temperature to stabilize.
   - Record the T1 image raw file until all scripted frame got recorded.
   - Create the T1 image hdr file.
5. Record the warm (T2) thermal image
   - Move the T2 blackbody under the fore objective.
   - Wait for T2 temperature to stabilize
   - Record the T2 image raw file until all scripted frame got recorded.
   - Create the T2 image hdr file.
6. Home the blackbodies to allow imaging the scene again.
7. Create the metadata.
8. Create the image thumbnail.
9. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
10. Create the manifest and close the dataset.
11. Queue the dataset in Lumo FileTransfer Server
12. Update the metadata if required

13. Update the name generator if required

Image with GPS Data, Embedded Dark Image and Thermal Images

The **Image with GPS Data, Dark Image and Thermal Images** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, thermal images, *GPS*, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors in airborne systems.

The workflow steps are:

1. Open the dataset
    - Save the current time.
    - Create the dataset name.
    - Create the dataset folder structure.
2. Start recording the navigation data
3. Wait for the start delay
4. Mask the triggers
5. Wait for the first PPS to star the image data recording
6. Record the image files
    - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
    - Create the image hdr file.
7. Wait for the stop delay.
8. Stop recording the navigation data.
9. Record the dark data.
    - Re-open the image file
    - Close the shutter
    - Record the dark data at the end of the image raw file until all scripted frame got recorded.
    - Open the shutter
    - Update the image hdr file.
10. Record the cool (T1) thermal image
    - Move the T1 blackbody under the fore objective.
    - Wait for T1 temperature to stabilize.
    - Record the T1 image raw file until all scripted frame got recorded.
    - Create the T1 image hdr file.
11. Record the warm (T2) thermal image
    - Move the T2 blackbody under the fore objective.
    - Wait for T2 temperature to stabilize
    - Record the T2 image raw file until all scripted frame got recorded.
    - Create the T2 image hdr file.
12. Home the blackbodies to allow imaging the scene again.
13. Create the metadata.
14. Create the image thumbnail.
15. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
16. Create the manifest and close the dataset.
17. Queue the dataset in Lumo FileTransfer Server
18. Update the metadata if required
19. Update the name generator if required

Thermal Images

The **Thermal Images** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, thermal images, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors in airborne systems.

The workflow steps are:

1. Open the dataset
   - Save the current time.
   - Create the dataset name.
   - Create the dataset folder structure.
2. Record the cool (T1) thermal image
   - Move the T1 blackbody under the fore objective.
   - Wait for T1 temperature to stabilize.
   - Record the T1 image raw file until all scripted frame got recorded.
   - Create the T1 image hdr file.
3. Record the warm (T2) thermal image
   - Move the T2 blackbody under the fore objective.
   - Wait for T2 temperature to stabilize
   - Record the T2 image raw file until all scripted frame got recorded.
   - Create the T2 image hdr file.
4. Home the blackbodies to allow imaging the scene again.
5. Create the metadata.
6. Create the image thumbnail.
7. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
8. Create the manifest and close the dataset.
9. Queue the dataset in Lumo FileTransfer Server
10. Update the metadata if required
11. Update the name generator if required

Thermal Images with Embedded Dark

The **Thermal Images with Embedded Dark** recording workflow will create a dataset which will contain the image data files with the dark data added at the end of the image raw file, thermal images, metadata information, manifest and thumbnail.

Note: This workflow is dedicated for thermal sensors in airborne systems.

The workflow steps are:

1. Open the dataset
   - Save the current time.
   - Create the dataset name.
   - Create the dataset folder structure.
2. Wait for the start delay
3. Mask the triggers
4. Record the image files
   - Record the image raw file until the operator stops the recording or the all scripted frame got recorded.
   - Create the image hdr file.
5. Wait for the stop delay.
6. Record the dark data.

- Re-open the image file
- Close the shutter
- Record the dark data at the end of the image raw file until all scripted frame got recorded.
- Open the shutter
- Update the image hdr file.

7. Record the cool (T1) thermal image

- Move the T1 blackbody under the fore objective.
- Wait for T1 temperature to stabilize.
- Record the T1 image raw file until all scripted frame got recorded.
- Create the T1 image hdr file.

8. Record the warm (T2) thermal image

- Move the T2 blackbody under the fore objective.
- Wait for T2 temperature to stabilize
- Record the T2 image raw file until all scripted frame got recorded.
- Create the T2 image hdr file.

9. Home the blackbodies to allow imaging the scene again.
10. Create the metadata.
11. Create the image thumbnail.
12. Extract the radiometric calibration frame, if available, and place a copy in the dataset.
13. Create the manifest and close the dataset.
14. Queue the dataset in Lumo FileTransfer Server
15. Update the metadata if required
16. Update the name generator if required

Plugins Manager

Overview

Lumo plugins are software extension modules that provide custom functionality, which the application itself does not support. A plugin always consists of a single master *.lua* file that contains the necessary interface required for a Lumo plugin. A plugin can also load other *.lua* files or C libraries, if needed. The master *.lua* file is loaded from the Lumo application's Plugins manager.

Using Plugins

This section describes how to use Lumo plugins.

The basic controls for Lumo plugins include:

- *Loading Plugins*
- *Unloading Plugins*
- *Enabling Plugins*
- *Disabling Plugins*
- *Changing Parameters*

A more detailed description of Lumo plugins can be found in *Overview*.

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Load a new plugin by selecting **Add plugin**, and locate the plugin *lua* file.
3. Modify the plugin parameters, if needed. All plugins do not necessarily provide custom parameters.
4. Enable the plugin by selecting the **Power** button. The button lights up if the plugin is enabled.
5. The plugin is now enabled and running.

   The plugin interface provides functionality for the plugin to provide a scrolling progress bar to the **Plugins** section whenever it is processing a task.
6. **Lumo Plugin Manager** remembers the state of each plugin between application restarts - an enabled plugin will be automatically enabled after a restart.

Loading Plugins

This section describes how to load a Lumo plugin.

The controls for loading a Lumo plugin are located in the **Plugins** section of the *Setup View* (see *Using Plugins*). All loaded plugins will be listed in this section after being successfully loaded. The loaded plugins are **Disabled** by default, and you must enable them before use (see *Enabling Plugins*).

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Select **Add plugin**.
3. Locate the master *.lua* file related to the plugin. Usually, plugins are placed in separate sub-folders within the **plugins** folder of a Lumo application.
4. Select **Open** after finding the correct plugin.

   If a plugin is loaded successfully, it will be added to the list of loaded plugins.

Unloading Plugins

This section describes how to unload a Lumo plugin.

The controls for unloading a Lumo plugin are located in the **Plugins** section of the *Setup View* (see *Using Plugins*). All loaded plugins are listed in this section, and can be separately unloaded. If a plugin is **Active** while being unloaded, it is first disabled by the **Lumo Plugins Manager** and only then unloaded.

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Find the correct plugin from the list of all available plugins.
3. If you want to forcefully unload the plugin, regardless of its current state, select **X**.

   To gracefully unload plugins, always wait for the plugin to stop processing by making sure the progress bar is not active on the plugin.
4. After a plugin has been successfully unloaded, it is removed from the list of loaded plugins.

Enabling Plugins

This section describes how to enable a Lumo plugin.

The controls for enabling a Lumo plugin are located in the **Plugins** section of the *Setup View* (see *Using Plugins*). The loaded plugins are **disabled** by default, and you must enable them before use.

The different states for an enabled plugins are:

- **Waiting**
  - Refers to a state where the plugin is waiting for either a device to be ready for usage, or an event to be triggered, until the plugin enters the **Active** state.
- **Active**
  - Refers to a state where the plugin is currently running. Depending on the plugin logics, the plugin can be idle or processing which is indicated by a scrolling progress bar next to the **Active** indicator.
- **Error**
  - Refers to an error state, which usually indicates an error within the plugin code. In the case of an error, an error string is shown in the **Extended view** of the plugin.

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Find the correct plugin from the list of all available plugins.
3. Check the state of the **Power** button for the plugin and if the button is not lit, select it to enable the plugin.

   If a plugin is successfully enabled, it will change its **Disabled** state to either **Waiting** or **Active**, depending on the plugin implementation. In the case of an error, the state changes to **Error**.

Disabling Plugins

This section describes how to disable a Lumo plugin.

The controls for disabling a Lumo plugin are located in the **Plugins** section of the *Setup View* (see *Using Plugins*). Disabling an plugin immediately stops any running plugin processes.

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Find the correct plugin from the list of all available plugins.
3. Check the state of the **Power** button for the plugin, and if the button is lit, select it to disable the plugin.

   A successfully disabled plugin will automatically change its state to **Disabled**.

Changing Parameters

This section describes how to change Lumo plugin parameters.

The controls for changing Lumo plugin parameters are located in the **Plugins** section of the *Setup View* (see *Using Plugins*). When a plugin is loaded, it can generate plugin specific parameters to the **Plugins** section. These parameters can then be used by the plugin when processing, for example, when choosing the processing output data type from the GUI, which then refers to the data output of the plugin. These parameters can be shown or hidden, by using the **triangle** toggle button in the **Plugins** section. You can change the plugin parameters regardless of the plugin state.

Proceed as follows:

1. Open the **Plugins** section of the *Setup View*.
2. Find the correct plugin from the list of all available plugins.
3. Select **triangle** to show the plugin parameters.
4. Modify the desired parameters.

ENVI Header File Keyword Summary

This section provides a summary for ENVI header file keywords.

File description		
ENVI	identifier string for ENVI file format header	
description	image description string	
file type	ENVI-defined file type, check filetype.txt	
Sensor description		
sensor type	sensor identifier string	
GPS/IMU and timing information		
acquisition date	data file acquisition date, mm-dd-yyyy	
GPS Start Time	start time of GPS/IMU data recording	
GPS Stop Time	stop time of GPS/ IMU data recording	
GPS Starting point	starting location of GPS/ IMU data recording	
GPS Ending point	stopping location of GPS/ IMU data recording	
NavSync Timing = {XXXX, YYYY}	XXXX	delay in [ms] before the first NavSync message #999 was read in the navigation data stream after the data acquisition started
	YYYY	NavSync time read from the sync message #999
NavSync Extra	the number of extra sync messages in the navigation data	
qpfTiming = {XXXX, YYYY}	XXXX	delay measured by query performance timer in [ms] before the first image frame is acquired for the frame grabber
	YYYY	Measured flight line data acquisition time in [ms]
First sensor or dual sensor settings (In AisaFENIX sensor1=VNIR)		
fps_set	set data acquisition frame rate	
fps	true data acquisition frame rate	
fps_qpf	frame rate estimated by the QPF timer	
sensorid	sensor serial number	
tint1	integration time set to sensor	
binning = {X, Y}	X	sensor spectral total binning (HW+SW)
	Y	sensor spatial total binning (HW+SW)
hw_binning = {X, Y}	X	sensor spectral hardware (HW) binning
	Y	sensor spatial hardware (HW) binning
sw_binning = {X, Y}	X	sensor spectral software (SW) binning
	Y	sensor spatial software (SW) binning
vroi = {Y1, Y2}	full spectral vertical region of interest	
hroi = {X1, X2}	full spatial horizontal region of interest	

vimg = {Y1, Y2}	sensor1 spectral image area for CaligeoPRO to process, subset of vroi	
himg = {X1, X2}	sensor1 spatial image area for CaligeoPRO to process, subset of hroi	
fodis = {X1, X2}	sensor1 spatial pixel range used by fodis sensor	
temperature = {t1,t2,t3,t4}	sensor specific temperatures (t1-t4) for up to 4 points.	
Second sensor settings (In AisaFENIX sensor2=SWIR)		
sensorid2	second sensor serial number	
tint2	integration time set to sensor2	
binning2 = {X, Y}	X	sensor2 spectral total binning (HW+SW)
	Y	sensor2 spatial total binning (HW+SW)
hw_binning2 = {X, Y}	X	sensor2 spectral hardware (HW) binning
	Y	sensor2 spatial hardware (HW) binning
sw_binning2 = {X, Y}	X	sensor2 spectral software (SW) binning
	Y	sensor2 spatial software (SW) binning
vimg2 = {Y1, Y2}	sensor2 spectral image area for CaligeoPRO to process, subset of vroi	
himg2 = {X1, X2}	sensor2 spatial image area for CaligeoPRO to process, subset of hroi	
fodis2 = {X1, X2}	sensor2 spatial pixel range used by fodis sensor	
Spectral band information		
default bands = {R, G, B}	default band numbers for R, G, and B channels in ENVI images if only one number is given, used as a gray scale channel	
wavelength	center wavelength values for each band in the image	
fwhm	full-width-half-maximum values for each band in the image	
Image data information		
header offset	number of bytes used for embedded header information before image data in data file	
data type	1	8-bit byte
	2	16-bit signed integer
	3	32-bit signed long integer
	4	32-bit floating point
	5	64-bit double-precision floating point
	6	2x32-bit complex
	9	2x64-bit double-precision complex
	12	16-bit unsigned integer
	13	32-bit unsigned long integer
	14	64-bit signed long integer
	15	64-bit unsigned long integer

byte order	0	least significant byte first
	1	most significant byte first
interleave	data file interleave format: BSQ / BIP / BIL	
lines	number of image frames in data file	
bands	number of spectral bands in the image frame	
samples	number of spatial pixels in the image frame	
x start	x-coordinate of the upper-left corner pixel	
y start	y-coordinate of the upper-left corner pixel	
autodarkstartline	starting line of the acquired dark frame in the end of data file	
errors	none	no dropped frames, that is, no .log file
	recover	frames dropped, data recoverable
	terminal	frames dropped, unable to recover

GPS / Scanner Interface

Position and 1PPS information are received from the GPS unit. The 1PPS pulse is used to measure the nav sync – timing message. The nav sync –timing value is inserted to the GPS data flow, and data is forwarded to the PC through the USB interface (RS232 virtual serial port).

When the nav sync timing measurement is started:

- The trigger pulse state is saved.
- Trigger pulses are counted.
- The time delay between the falling edge of the 1PPS and the rising edge of trigger1 pulse is measured, and saved to the register.
- Nav sync-timing message is generated and sent to PC.
- The insertion place is detected when a quite time is detected in GPS data line.

The same GPS interface can also be used with the scanner unit. When a time-stamp measure is not started, the 1PPS line is ignored, and the incoming RS232 line from the GPS/Scanner is directly connected to the PC without any nav sync timing function.

The PC interface is implemented by a USB. The PC receives GPS messages and timing information through the USB interface. Messages from PC to GPS are transparent to the FPGA.

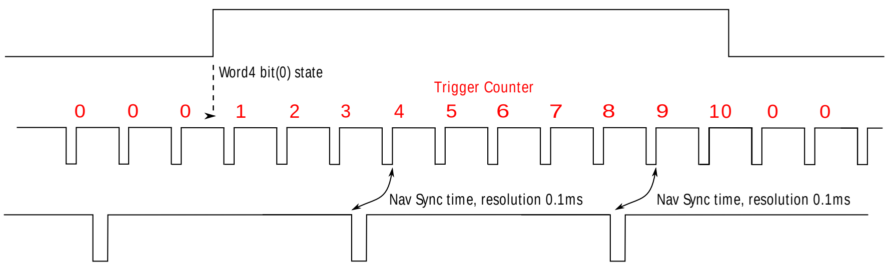


Figure 48: NavSync Diagram

Table 3: Nav Sync – Timing Message

#	Example	Meaning
Word1	0x81FF	sync
Word2	0x03E6	Message id #998
Word3	0x0002	Data word count: 2
Word4	0x800x	Flags, assuming no errors, last bit: trigger state beginning of measurement
		Trigger pulse is low when measure is started = 0x8000 • 0x8... = no errors
		Trigger pulse is high when measure is started = 0x8001 • 0x8... = no errors
Word5	0xhhll	Header check sum
Word6	0xtttt	16-bit delay, resolution 0.1 ms
Word7	0xnnnn	16-bit trigger counter
Word8	0xhhll	Data check sum

Message is sent low byte first, high byte after that.

Table 4: Check Sum Values

Header check sum High byte (hh)	MSB(word1) XOR MSB(word2) XOR ... MSB(word4)
Header check sum Low byte (ll)	LSB(word1) XOR LSB(word2) XOR ... LSB(word4)
Data check sum High byte (hh)	MSB(word6) XOR MSB(word7)
Data check sum Low byte (ll)	LSB(word6) XOR LSB(word7)

Configuring the SCB GPS Parameters

This section describes how to configure the SCB GPS parameters.

Multiple acquisition computers can share the same camera trigger pulses, GPS data and PPS signals. This is done through chaining the Sensor control boards. This requires linking the acquisition computers with chaining cables via the Chain In and Chain Out ports, and configuring the software to match the hardware setup. The following steps guide you through configuring the software.

Proceed as follows:

1. Locate the **SCB widget** in **Setup ViewSCB X**, where **X** refers to the GPS index.
2. Set the **Chaining mode**

   The chaining mode needs to match the **Chain In**/**Chain Out** cabling configurations of the acquitision computers.

   1. On the computer with the GPS unit, the chaining cable is connected to the **Chain out** connector. Set the **Chaining mode** also to **Chain out** from dropdown menu.
   2. If there are more than two computers in the setup, then chaining cables are connected to both **Chain in** and **Chain out** ports of the computer in the middle of the chain. Set **Chaining Mode** to **Chain in and out**.
   3. The last computer in the chain only has the **Chain in** port connected. Set **Chaining Mode** to **Chain in**.

Glossary

B

BPR

Bad pixel replacement.

C

CCW

Counterclockwise.

CW

Clockwise.

D

DN

Data number.

E

EMD

Earth mover's distance.

G

GPS

Global positioning system.

GUI

Graphical user interface.

H

HDR

The ENVI header file format.

HSI

Hyper spectral imager.

I

INS

Inertial Navigation System.

IP

Internet Protocol

M

MMF

Memory mapped file.

N

NIC

Network interface card.

Normalization

The scaling of *pixel* values from the original range between dark and white reference values to the full intensity range of a pixel. (Also known as dynamic range expansion.)

NUC

Non-uniformity correction.

Q

QA/QC

Quality assurance, quality control.

R

RAW

The camera output data format.

RGB

Red, Green and Blue.

ROI

Region of interest.

S

SATA

Serial ATA, a computer bus interface that connects host bus adapters to mass storage devices such as hard disk drives and optical drives.

SSD

Solid state drive.

SSP

Specim Sensor Profile.

U

URL

Uniform Resource Locator.

W

Widget

A **widget** is an element of interaction in a graphical user interface (*GUI*), such as a button or a scroll bar.

For more information, see *http://en.wikipedia.org/wiki/Widget_%28GUI%29*.

Workflow

A sequence of steps, which can be systematically reproduced.

X

XML

Extensible Markup Language.

Appendix

ENVI Header Files

An ENVI header file contains information required to read an image data file. The header file has the same name as the image data file, with the file extension `.hdr`. The header file keywords are presented in the table below.

Header keyword	Description
File description	
`ENVI`	An identifier string for the ENVI file format header.
`description`	An image description string.
`file type`	An ENVI-defined file type, check `filetype.txt`.
AISA sensor description	
`sensor type`	An AISA sensor identifier string.
`sensorid`	An AISA sensor serial number.
GNSS/IMU information	
`acquisition date`	The data file acquisition date.
`GPS Start Time`	The start time of GNSS/IMU data recording.
`GPS Stop Time`	The stop time of GNSS/IMU data recording.
`GPS Starting point`	The starting location of GNSS/IMU data recording.
`GPS Ending point`	The stopping location of GNSS/IMU data recording.
`NavSync Timing = {XXXX, YYYY}`	• `XXXX` = Delay in [ms] before the NavSync message was read in the navigation data stream after the data acquisition started. • `YYYY` = NavSync time read from the sync message.
`NavSync Extra`	The number of extra sync messages in the navigation data.
Data acquisition information	
`fps`	The data acquisition frame rate.
`tint`	The integration time used in the data acquisition.
`tint1`	The same as tint for the first sensor on dual grabbing setup.
`tint2`	The same as tint for the second sensor on dual grabbing setup.
`temperature`	The measurement values from temperature sensors, depends on system.
`binning = {X, Y}`	• `X` = Sensor spectral binning. • `Y` = Sensor spatial binning.
`qpfTiming = {XXXX, YYYY}`	• `XXXX` = Additional delay in [ms] before the first image frame is acquired for National Instruments grabber. • `YYYY` = Flight line data acquisition time in [ms].
`errors`	• `none` = no dropped frames

Header keyword	Description
	• `recover` = frames dropped, data recoverable • `terminal` = frames dropped, unable to recover
Spectral band information	
`default bands = {R, G, B}`	The default band numbers for R, G, and B channels in ENVI images. If only one number is given, used as a gray scale channel.
`wavelength`	The center wavelength values for each band in the image.
`fwhm`	The full-width-half-maximum values for each band in the image.
Image data information	
`header offset`	The number of bytes used for embedded header information before image data in data file.
`data type`	• 1 = 8-bit byte • 2 = 16-bit signed integer • 3 = 32-bit signed long integer • 4 = 32-bit floating point • 5 = 64-bit double-precision floating point • 6 = 2x32-bit complex • 9 = 2x64-bit double-precision complex • 12 = 16-bit unsigned integer • 13 = 32-bit unsigned long integer • 14 = 64-bit signed long integer • 15 = 64-bit unsigned long integer
`byte order`	• 0 = least significant byte first • 1 = most significant byte first
`interleave`	The data file interleave format: BSQ / BIP / BIL.
`lines`	The number of image frames in data file.
`bands`	The number of spectral bands in the image frame.
`samples`	The number of spatial pixels in the image frame.
`x start`	The x-coordinate of the upper-left corner pixel.
`y start`	The y-coordinate of the upper-left corner pixel.
`vroi = {Y1, Y2}`	The vertical region of interest, the full spectral image area.
`hroi = {X1, X2}`	The horizontal region of interest, the full spatial image area.
`vimg = {Y1, Y2}`	The spectral image area for Caligeo to process, subset of vroi.
`vimg1`	The same as vimg for first sensor on dual grabbing setup.
`vimg2`	The same as vimg for second sensor on dual grabbing setup.

Header keyword	Description
`himg = {X1, X2}`	The spatial image area for Caligeo to process, subset of hroi.
`himg1`	The same as himg for first sensor on dual grabbing setup.
`himg2`	The same as himg for second sensor on dual grabbing setup.
`fodis = {X1, X2}`	The spatial pixel range used by fodis sensor.
`fodis2 = {X1, X2}`	The same as fodis for second sensor on dual grabbing setup.
`autodarkstartline`	The starting line of the acquired dark frame in the end of data file.
aisaOWL keywords	
`Image_start`	The first band of image area.
`Image_stop`	The last band of image area.

Specim Dataset

The Specim dataset is a folder structure containing all data and calibration files related to a capture. One dataset is created per flight line, scan or normal capture.

This structure helps with file management and simplifies tasks like batch processing.

Dataset support is implemented in all Lumo applications.

Note:

Keep the dataset original data intact. Store processed results and other files added after recording to a new subfolder and / or files.

When adding new contents to the datasets, update the `manifest.xml` file to reflect the content of the dataset.

Automatic file import can be conveniently done by giving the absolute path of the `manifest.xml` file. All absolute paths for all the capture files in the dataset can be built by combining the absolute path of the dataset folder and the relative paths parsed from the manifest.

At the root of the dataset, there are two directories, the `manifest.xml` file, and a thumbnail image.

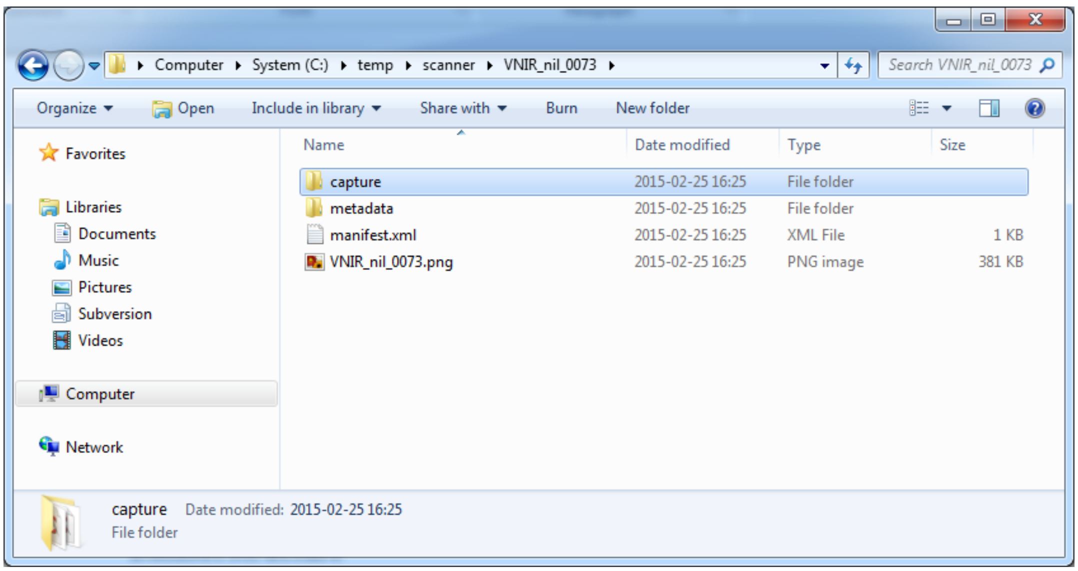


Figure 49: Dataset Root Folder

The Capture Folder

The `capture` directory contains all the data files related to one acquisition. The file extensions are:

- **.hdr** for the header files
- **.raw** for the files containing the data
- **.log** for the dropped frame (optional)
- **.nav** for navigation data

The `capture` directory is depicted in the figure below:

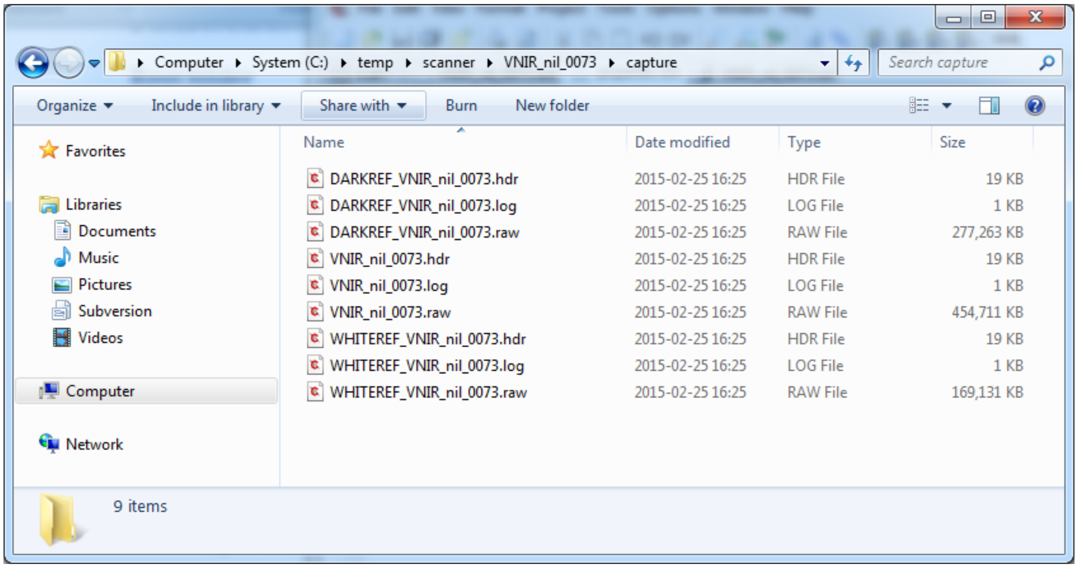


Figure 50: Capture Directory

To be able to differentiate the different capture files, prefixes are used. The files without a prefix and with the dataset name always are the image files or the navigation data. The others are references. The prefixes are presented in the table below:

Table 5: File Prefixes

Prefix	File type	Example
`none`	Image data	• `pencil_090512-080027.hdr` • `pencil_090512-080027.raw` • `pencil_090512-080027.log`
`none`	GPS data	• `pencil_090512-080027.nav`
`DARKREF`	Dark reference files	• `DARKREF_pencil_090512-080027.hdr` • `DARKREF_pencil_090512-080027.raw` • `DARKREF_pencil_090512-080027.log`
`WHITEREF`	White reference files	• `WHITEREF_pencil_090512-080027.hdr` • `WHITEREF_pencil_090512-080027.raw` • `WHITEREF_pencil_090512-080027.log`
`T1`	Cool reference files (OWL sensor)	• `T1_pencil_090512-080027.hdr` • `T1_pencil_090512-080027.raw` • `T1_pencil_090512-080027.log`
`T2`	Warm reference files (OWL sensor)	• `T2_pencil_090512-080027.hdr` • `T2_pencil_090512-080027.raw` • `T2_pencil_090512-080027.log`

The Metadata Folder

The `metadata` folder contains:

- An XML file (*.xml) with the default sensor metadata as well as the user defined metadata.

- An XML stylesheet (*.xsl) for viewing the metadata XML file in a web browser.

The `metadata` directory is depicted in the figure below:

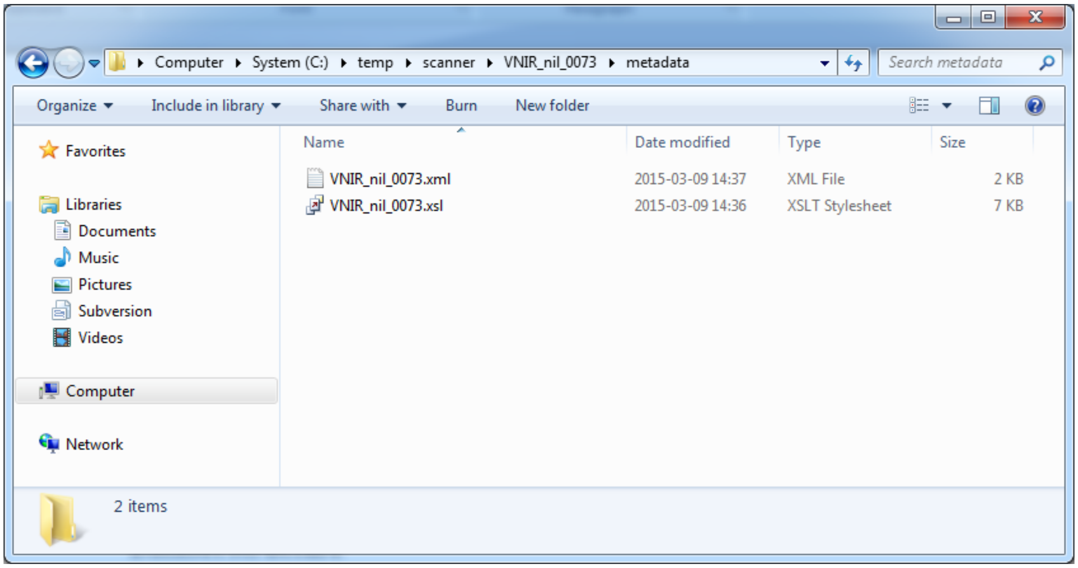


Figure 51: The `metadata` Directory

The metadata XML File as opened in a Web browser, is depiceted in the figure below:

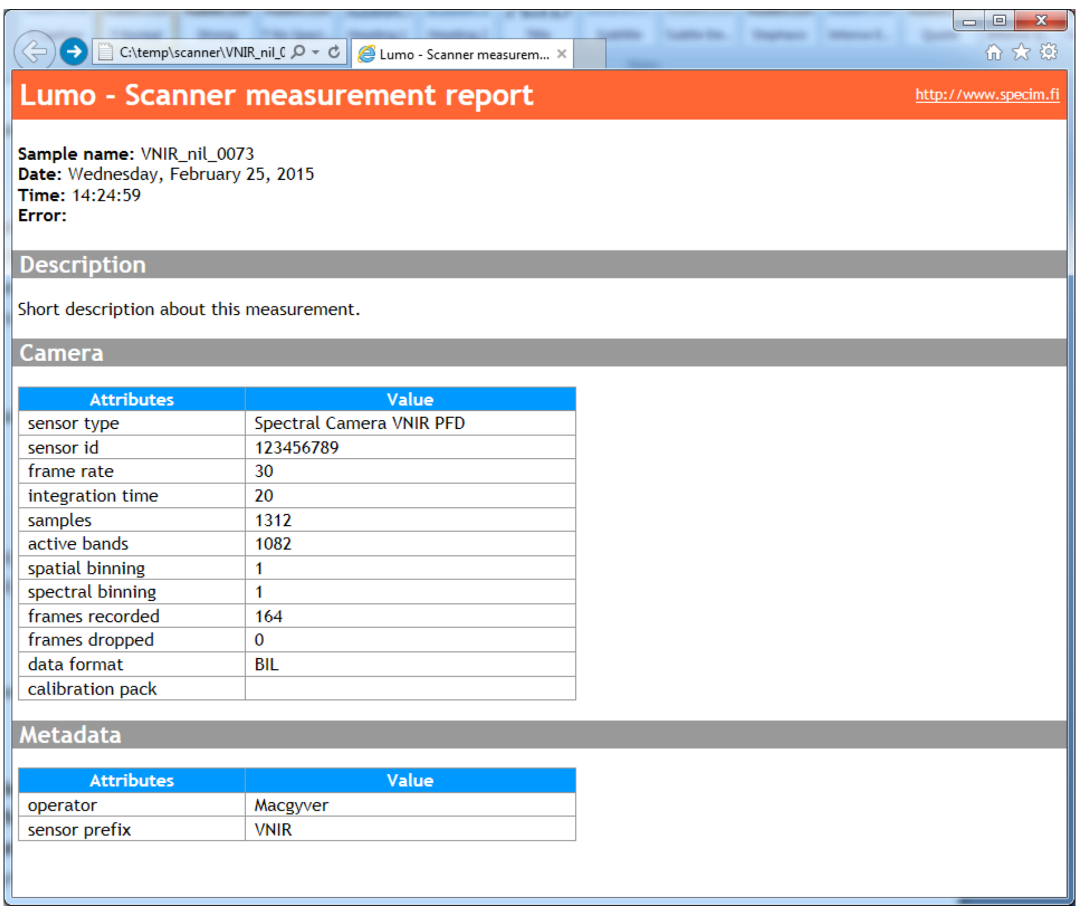
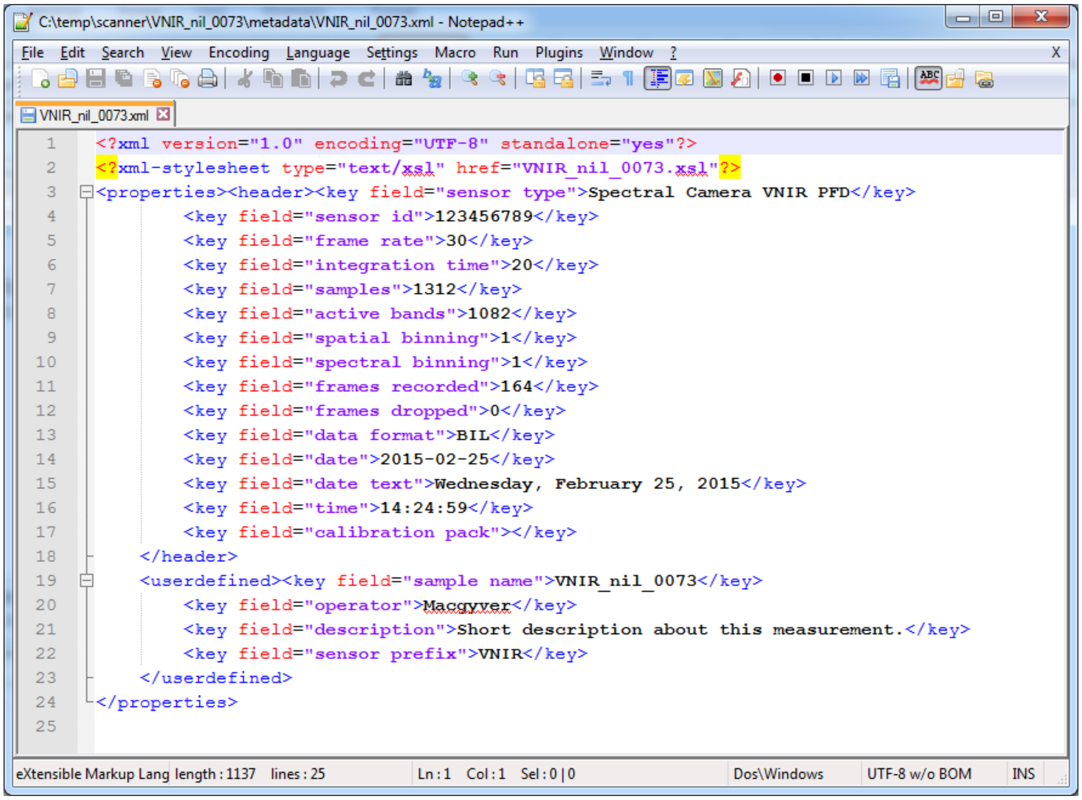


Figure 52: The Metadata XML File in a Web Browser

The metadata consists of a simple and easy-to-parse XML tree structure. The metadata XML file as opened in a text editor, is depiceted in the figure below:




```xml
<?xml version="1.0" encoding="UTF-8" standalone="yes"?>
<?xml-stylesheet type="text/xsl" href="VNIR_nil_0073.xsl"?>
<properties><header><key field="sensor type">Spectral Camera VNIR PFD</key>
        <key field="sensor id">123456789</key>
        <key field="frame rate">30</key>
        <key field="integration time">20</key>
        <key field="samples">1312</key>
        <key field="active bands">1082</key>
        <key field="spatial binning">1</key>
        <key field="spectral binning">1</key>
        <key field="frames recorded">164</key>
        <key field="frames dropped">0</key>
        <key field="data format">BIL</key>
        <key field="date">2015-02-25</key>
        <key field="date text">Wednesday, February 25, 2015</key>
        <key field="time">14:24:59</key>
        <key field="calibration pack"></key>
    </header>
    <userdefined><key field="sample name">VNIR_nil_0073</key>
        <key field="operator">Macgyver</key>
        <key field="description">Short description about this measurement.</key>
        <key field="sensor prefix">VNIR</key>
    </userdefined>
</properties>
```

Figure 53: The Metadata XML File in a Text Editor

Manifest File

The `manifest.xml` enumerates all the files in the dataset, their relative paths within the dataset, types and extensions.

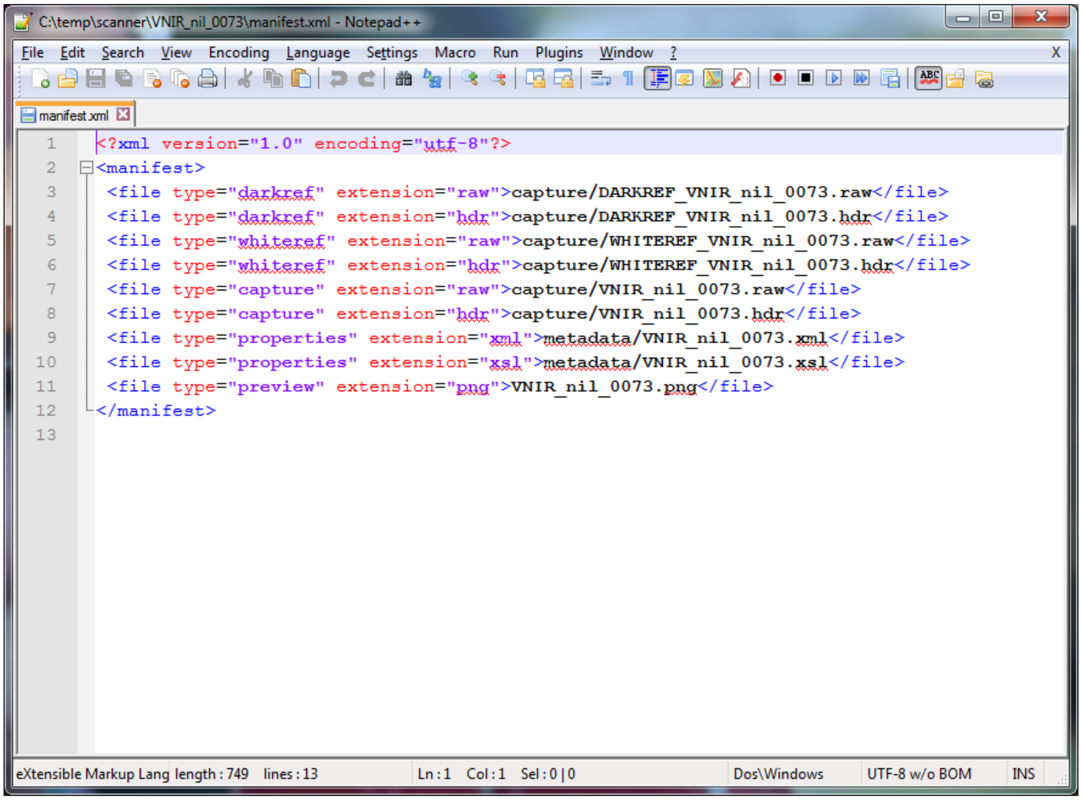


Figure 54: `manifest.xml` Example

The manifest file types are:

- **darkref** for dark reference files
- **whiteref** for white reference files
- **capture** for image files
- **t1** for cool reference files
- **t2** for warm reference files
- **gps** for navigation data file
- **properties** for metadata files
- **preview** for thumbnail file

Thumbnail

The thumbnail image is created using the RGB band set during the recording. The color scaling is done by finding the minimum and maximum value across all pixels of the RGB bands. The image isn't normalized and is provided for quick identification of the dataset only.

Figure 55: Thumbnail Example