VERSION 1.5.0.3

SPECTROLYTIC GMBH

Innovative Solutions in Spectroscopy

SphinxSuite User Manual

SPECTROLYTIC GMBH

SphinxSuite User Manual

Please note: the information contained in this document subject to change without further notification. Spectrolytic reserves the right to alter the performance and any resulting specification. Spectrolytic may choose not to supply any devices as a commercial product. No responsibility is accepted for any consequential loss incurred.

Proprietary content property of Spectrolytic GmbH. No copying, distribution or other use without prior written permission of Spectrolytic GmbH.

Spectrolytic GmbH.
Bergstrasse 14
92533 Wernberg-Köblitz
Germany
Phone +49 (0) 960 499 567
Website: www.spectrolytic.com
Email: info@spectrolytic.com

Spectrolytic

TABLE OF CONTENTS

TABLE OF FIGURES

TABLE OF TABLES

1 SYSTEM REQUIREMENTS

- Windows XP or Windows 7 or Windows 8.x
- 250 MB RAM
- At least 300 MB of Hard Disk Space (Run-Time included)
- RJ45 port for communication over Ethernet
- USB 2.0 port or higher for communication over USB (full and low speed)
- Bluetooth 2.1 or higher Adapter for communication over Bluetooth (for availability refer to the technical specification of your device)

2 INSTALLATION

2.1 SPHINXSUITE SOFTWARE

SphinxSuite was developed using National Instruments LabVIEW and needs the LabVIEW run time engine (RTE) to function. Two software versions are available: one with a smaller size that doesn't have the RTE for those who already have it, and one that does. These are available for download from www.comline-elektronik.de/downloads/sphinxsuite/

- If you have downloaded SphinxSuite from our online portal in the *Customer Service Home* section (https://secure.softwarekey.com/solo/customers/), the LabVIEW RTE is not included. To run your SphinxSuite software you will need to download the LabVIEW RTE from www.comline-elektronik.de/downloads/sphinxsuite/
- For a software compatibility overview of SphinxSuite and its runtime engines, refer to the file compatibility.txt in the download section
- Unzip the downloaded file and click …SphinxSuite Installer\Volume\setup.exe to start the installation process
- *Please note:* You need administrator rights to install the software
- The device doesn't need to be connected to the computer during installation

2.2 PREPARE FIRST START

After the installation of SphinxSuite the software needs to be prepared for the first start. Therefore a dialog will appear automatically showing the progress (Figure 1). This dialog shows the initialization of each SphinxSuite Toolkit. After all steps are performed, all ✖ crosses are replaced by ✓ checkmarks.

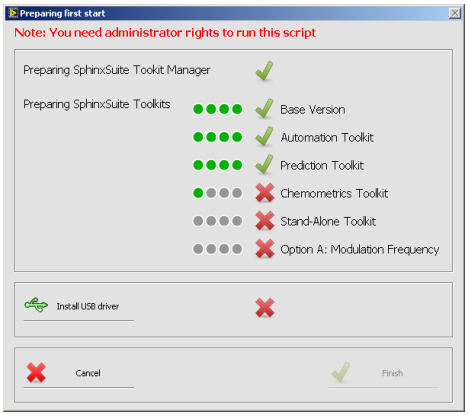


Figure 1: Prepare First Start

You now need to install SphinxSuite USB drivers to enable communication via USB. Click ***Install USB driver*** to start installing the device drivers. A Windows security dialog will appear (Figure 2). Choose ***Install this driver software anyway*** to install USB drivers. If you previously installed USB drivers, you can skip this step.

If you want to install the IRSphinx USB drivers later you can run the installer manually from: *"Installation Path\SphinxSuite\Drivers\IRSphinx USB Driver.exe"* or under ***Help > Install USB Drivers…*** in the software menu bar.

If an error occurred when preparing first start of SphinxSuite, you can open the dialog manually. Therefore navigate to your SphinxSuite installation path (e.g. *Program Files\SphinxSuite*) and execute the program *"Prepare First Launch.exe"* as administrator. If you do not run it as administrator, the program cannot be executed correctly.

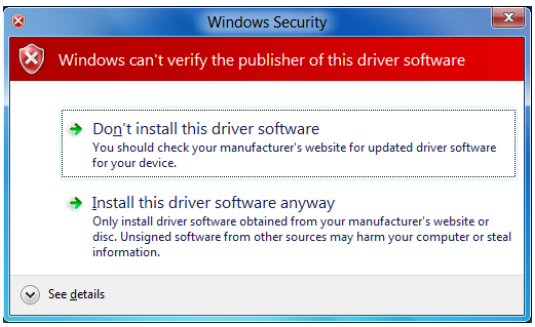


Figure 2: Windows Security Dialog

3 ACTIVATING SPHINXSUITE

You need to activate your **SPHINXSUITE BASE VERSION** and **SPHINXSUITE TOOLKITS** in order to use them. After the first installation of SphinxSuite you can evaluate the **BASE VERSION** for 14 days before activation is required. You can activate your software automatically online by entering your License ID and password when an internet connection is available (section 3.1). If your measurement computer is not connected to the internet you can activate your license manually from another computer connected to the internet (section 3.2). If your trial period has not expired you can evaluate the software (section 3.3) Figure 3 shows the activation dialog.

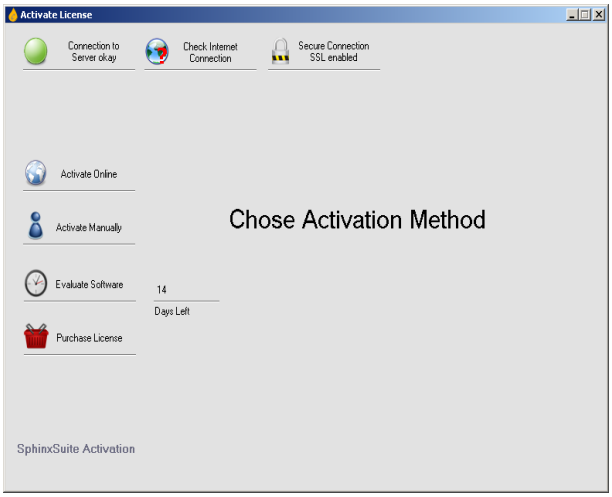


Figure 3: Activation Dialog

3.1 ACTIVATE ONLINE

When a connection to the activation server is available (see indicator on top of the dialog in Figure 3) you can activate your license online. Select ***Activate Online*** in the dialog and enter your License ID and password. After confirming the entered data the license status will be checked and, if valid, the software will be activated.

Note

A License ID can only be used once. Incorrectly entering your License ID will result in the loss of your License ID!

Example: If you enter the License ID of the "Automation Toolkit" when trying to activate "Chemometrics Toolkit" the "Chemometrics" License remains valid but the "Automation" License becomes invalid. In this scenario neither the "Automation" or "Chemometrics" toolkits would be activated.

3.2 ACTIVATE MANUALLY

If your computer is not connected to the internet you can activate your software manually by clicking on ***Activate Manually***. (Figure 4) .You will need to enter your registration details, click ***Okay*** then click on ***Next Step***. The software then generates two activation codes: *User Code 1* and *User Code 2*, which you need to activate your software. At this point you'll have two ways to submit your activation codes:

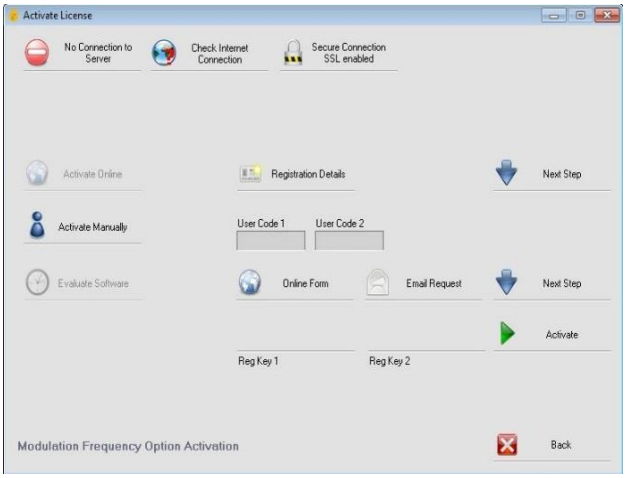


Figure 4: Activate License

Online Form (internet address can be copied and entered on a different computer): Enter the license ID and password of your module. After your login has been verified, you can enter *User Code 1* and *User Code 2*. Once you have submitted your activation codes your registration keys (*RegKey 1* and *RegKey 2*) will be generated. Enter these codes in the SphinxSuite activation dialog to activate your license. If *RegKey 2* is not generated, leave the dialogue blank.

Email Request: The previously entered registration details will be shown in this dialog. Enter your additional comments, if needed, before clicking ***Save Activation File*** to save the entered data to your disk. When saving the file to disk your default mail program will initialize. Attach the previously generated activation file and send it to software@irsphinx.com. After providing your License ID and activation codes your registration codes will be sent to you. Enter these codes in the SphinxSuite activation dialog, click ***Activate*** and confirm your data to activate your license.

3.3 EVALUATE SOFTWARE

If your trial period has not expired, you can start SphinxSuite or test a toolkit by clicking ***Evaluate***. An indicator shows the remaining days until an activation is required. SphinxSuite can be trialed for a period of 14 days however toolkits do not have a trial period by default. If you want to try a specific toolkit before buying it please contact software@irsphinx.com and provide your Customer ID.

Note

After expiration of a trial period the following actions will occur (dependent on module in trial mode):

SphinxSuite Base Version	You will get disconnected from the device Activation dialog appears
Automation Toolkit	Automated test sequence will be stopped Connection to IRSphinx remains established

Prediction Toolkit	Online prediction will be deactivated
Chemometrics Toolkit	Classification Model Builder or Predictor will be closed
Stand-Alone Toolkit	not available as trial
Option A: Modulation Frequency	Modulation frequency is set to 8 Hz

3.4 PURCHASING LICENSES

If you do not own a license for a module you can order a one by clicking ***Purchase License***. SphinxSuite Base Version. (Figure 5)

3.5 TOOLKIT MANAGER

When you first install SphinxSuite no toolkits are active as SphinxSuite is distributed as trial with a 14 days period. You can open the toolkits manager to get an overview of all available toolkits. In the dialog (Figure 5) you can activate, deactivate and request licenses. To open the toolkit manager, click ***Toolkits > Configure Toolkits…*** in the menu bar. You can find more information about the modules by choosing ***Toolkits Overview*** in the lower left of the dialog.

The dialog can be separated into the following parts:

A Module names
B Current module states
C Selected module
D Options for selected module

When choosing ***Activation Details*** a dialog will show the License ID, license state, registration data and remaining trial days.

Click ***Activate License*** to open the activation dialog (see Figure 4).

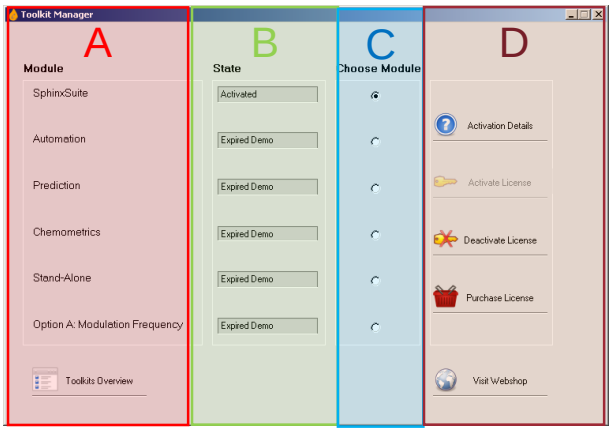


Figure 5: Toolkits Manager

If a module is already activated and you want to deactivate your license in order to use it on a different computer, click ***Deactivate License*** to open the deactivation dialog (Figure 6).

Carry out the following steps to deactivate your license:

Step 1: Request your custom deactivation codes by clicking ***Request Deactivation*** to bring up the deactivation form. *User Code 1* and *User Code 2* will be generated automatically. Save the deactivation file to your hard disk. You should then send this file to software@irsphinx.com.

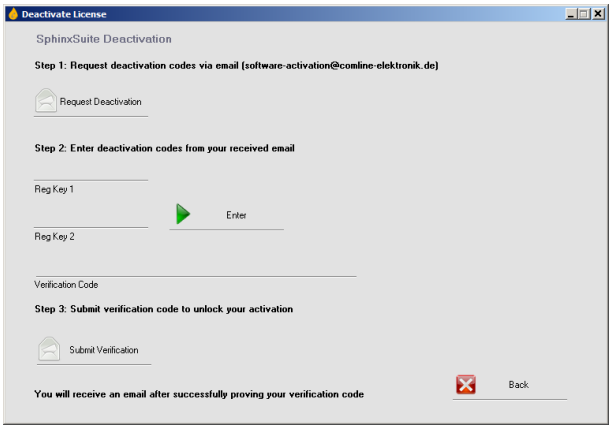


Figure 6: Deactivate License

Step 2: After validating your request you will receive an email with your deactivation codes (*Reg Key 1* and/or *Reg Key 2*). Enter the codes into the deactivation code dialog and click ***Enter*** to deactivate your license. After the deactivation operation has completed a verification code will be generated. You will need this code to reactivate your license later.

Step 3: Click ***Submit Verification*** and save the verification file to your disk. Send it to software@irsphinx.com and your license will be re-activated. You will receive an email after reactivating your license.

Note
Write down your unique verification code after proceeding step 2. Your verification code can only be generated once after you have entered your deactivation codes. If you cannot provide a matching verification code after deactivation you will lose your license.

4 SphinxSuite Basics

4.1 User Interface

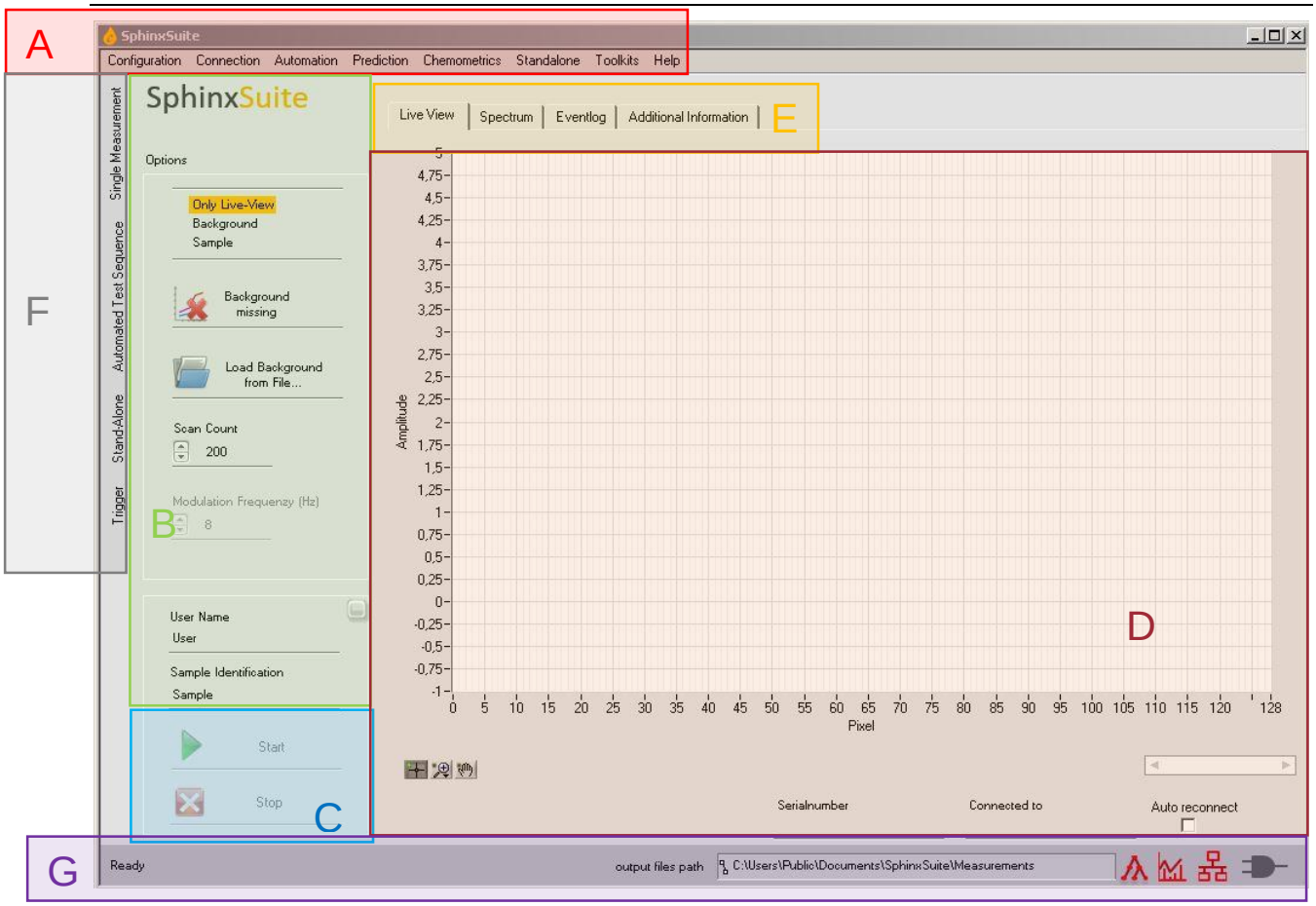


Figure 7: SphinxSuite User Interface

Figure 7 shows the user interface of SphinxSuite software. It can be separated into the following parts:

- A Menu Bar with configuration options (will be extended with additional toolkits)
- B Measurement options and import/export configuration
- C Start & Stop measurement buttons
- D Graphical display
- E Data selection tabs
- F Measurement selection (will be extended with additional toolkits)
- G Status bar: If you unplug a portable device from its power supply, the operation symbol changes from power supply operation to battery operation and shows the state of charge.

4.2 CONNECTING TO A DEVICE

You have three options to connect to the IRSphinx device: Ethernet, USB and, if you are using a portable device, Bluetooth.

4.2.1 Connection to a Device via Ethernet

If your device is connected to your local network or directly to your computer via an Ethernet cable, you can find your device in the menu bar under ***Connection > Ethernet > Find Devices…*** (Figure 8) or, alternatively, click the connection state symbol in the status bar. Select ***Refresh list*** in the dialog box to show the current operating state, serial number and IP-address of all connected IRSphinx devices. If an IRSphinx device is not responding refer to the hardware manual in order to set up your network correctly. Select the desired device and click ***Connect to device*** or double-click the selection to connect to the device. In the Table 1 below you can see an overview of the different operational states of the device.

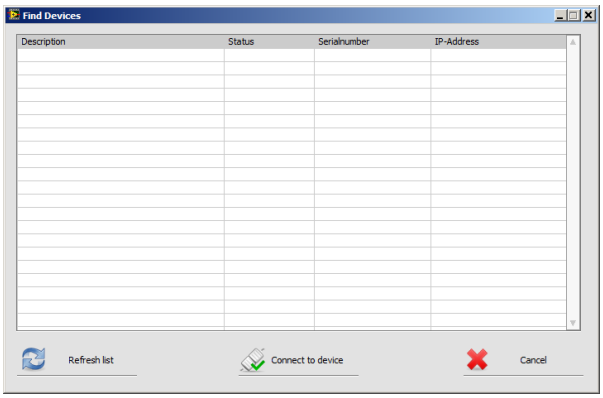


Figure 8: Find Ethernet Devices

State	Description	Connection possible?
Available	The device is in an idle state and is waiting for a connection.	YES
Busy	The device is connected to another computer.	NO
Stand Alone	The device is running in stand-alone mode. You can connect to the device to stop the stand-alone sequence (section 9).	YES
Unknown State	Your version of SphinxSuite is not compatible with the connected device.	NO

Table 1: Operation State

After the connection was established successfully, the connection state symbol switches to . Click on this symbol to get more information about the connection or to disconnect from device. Quick connect to the previous device after connection was lost or disconnected in the menu bar under ***Connection > Ethernet > Quick Connect to Previous Device.***

4.2.2 Connecting to a Device via USB

When you have connected IRSphinx to your computer via USB cable, select ***Connection > USB…*** to establish a connection between your computer and device (Figure 9). Select ***Refresh list*** in the dialog to identify all IRSphinx devices connected via USB. Click ***Connect to device*** or double-click your selected device to connect. After a connection is successfully established, the connection state symbol will switch from red to green. Click on this symbol for more information about the connection or to disconnect from device.

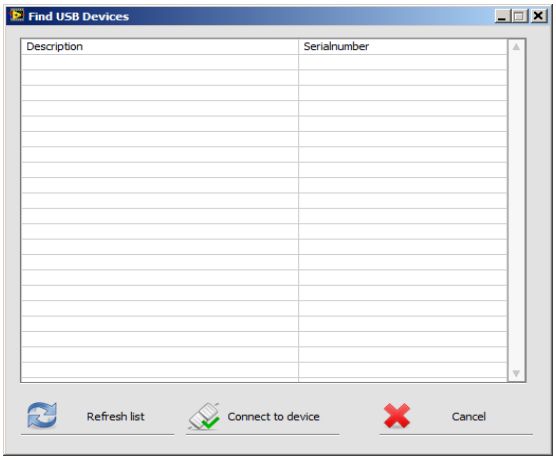


Figure 9: Find USB Devices

Note
*Please make sure you have installed the IRSphinx USB drivers (*section 2.2*) before attempting to connect via USB*

4.2.3 Connection to a Device via Bluetooth

To connect IRSphinx to your device using Bluetooth, you will first need to pair the device with your operating system. For detailed instructions please refer to Windows help by using the following links:

Windows XP: http://support.microsoft.com/kb/883259/en-us

Windows 7: http://windows.microsoft.com/en-us/windows7/add-a-bluetooth-enabled-device-to-your-computer

Windows 8: http://windows.microsoft.com/en-gb/windows-8/why-isnt-windows-finding-device

When you are asked for a pairing code, enter 0000 and follow the onscreen instructions.

Note
A Bluetooth connection can only be established with IRSphinx portable devices. In addition, when a Bluetooth connection is established, the Automation Toolkit and Stand-Alone Toolkit are disabled. Disconnect from Bluetooth and reconnect via Ethernet or USB to enable those features.

Open the connection dialog by clicking on ***Connection > Bluetooth*** which will open an *Installed Bluetooth Devices* dialog (Figure 10). (The timeout is set to 2000ms by default) Click ***Refresh List*** to obtain a list of all previously paired Bluetooth devices. The device table will then be updated to provide the names and addresses of all previously connected devices. You can then select a device and press ***Connect to device*** to establish a connection.

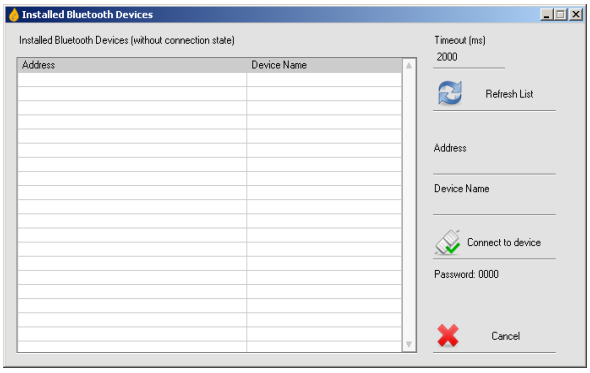


Figure 10: Installed Bluetooth Devices

Note
The Installed Bluetooth Devices dialog only shows devices that have already been paired to SphinxSuite. The dialog does not tell you if you can connect to the device or if the device is powered on.

4.2.4 Disconnecting from a Device

To disconnect from a device, select ***Connection > Disconnect From Device*** or click the connection state symbol and choose ***Disconnect From Device*** in the dialog. When the connection is successfully closed, the connection state symbol switches to . The device will now wait for a new connection from a host computer. If the stand-alone mode (section 9) has been started before, the device will run this mode as configured.

4.2.5 Shutting Down a Device

To shut down a device, select ***Connection > Disconnect And Shutdown Device*** When the device has shut down, the connection state symbol switches to . Depending on the firmware version of the device the following steps will be taken:

- Firmware version 0.0.8.9 and lower: The device will not completely be powered off. It is still running in power-saving mode. In the *Find Devices* dialog it appears as "*Stand By*". A connection to the device cannot be established. Restart the device to enable a new connection.
- Firmware version 0.0.9.0 and higher: The device will completely powered off. Press the *Power* push button on the device to restart it.

This option is not available when connected via Bluetooth.

4.2.6 Auto reconnect to a device

In the main window you will find a check box for ***Auto reconnect***. If you wish to reconnect to your device after the connection was lost then activate this check box. (Figure 11)

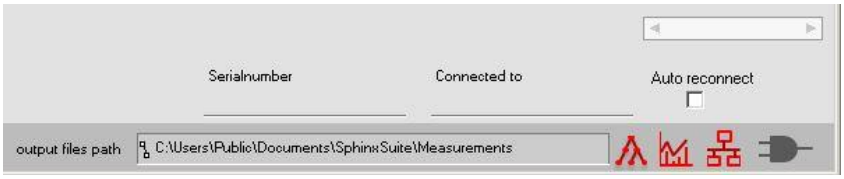


Figure 11: Auto reconnect check box

After connection is lost there will pop up a window where you can see the last Serial number, IP address and connection type. The program is trying to re connect to the last device and you will see a counter how often the program has tried to re connect and a status bar which shows the status of the next connection check. All 10 seconds the software is doing a reconnection attempt. (Figure 12) Clicking on ***Quit to reconnect*** will bring you back to the main window.

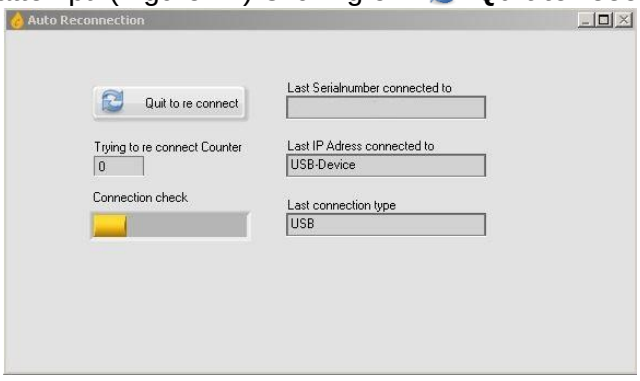


Figure 12: Auto reconnect

4.3 MAKING MEASUREMENTS

4.3.1 Generating a Sample Spectrum

To obtain the infrared spectrum of a sample it is necessary to make two measurements:

- A “background” measurement which is recorded when there’s a known material in the spectrometers optical path (e.g. air; see section 4.3.2).
- A “sample” measurement which is recorded when a sample is placed in the spectrometers optical path (e.g. oil, wine; see section 4.3.3).

A sample’s transmission spectrum can be calculated by dividing its sample measurement by a background measurement. A sample spectrum, unlike a sample measurement, will be unaffected by the frequency response of the spectrometer’s IR emitter and detector.

The signal to noise (S:N) of your measurements can be improved by recording the average of multiple spectral scans. The S:N of a measurement is directly proportional to the square root of the number of scans (NoS) averaged. For example, increasing the NoS from 5 to 20 will double S:N, however to double it again the NoS would need to increase from 20 to 80. Matching the NoS used for the background and sample measurements is the most time efficient way to minimize the S:N of a sample spectrum.

4.3.2 Recording a Background Measurement

The way that a background is measured depends on the type of IRSphinx device you are using:

- Transmission Mode IRSphinx (stationary):
  - Remove the cuvette from the device
  - Take a background measurement without a sample in the light path
  - The background measurement can be used for later sample measurement and does not need to be measured every time
  - To load a previously measured background, click ***Load Background from File…*** and SphinxSuite **File Loader** will open
  - Mount the cuvette to the device and continue with sample measurements
- ATR Mode IRSphinx (portable):
  - Clean the ATR crystal with a lint free tissue, making sure that it’s dry.
  - Take a background measurement without sample present on the crystal
  - Continue with the sample measurement

Approach within SphinxSuite:

1. In the options section: choose *"Background"*
2. Enter desired scan count, tester's name and sample identification.
3. Click ***Start*** to start the measurement. (Figure 13)
4. After the light sources have warmed up (status bar string: *"Warm Up Infrared Sources"*) the status bar will show a progress bar and the estimated time remaining for the measurement to complete.
5. When the measurement has finished and you checked the option ***Configuration > Measurement Complete Acoustic Signal*** in the menu bar a signal indicates the end of measurement.
6. After the background measurement has finished it is displayed graphically under the "Live View" tab.

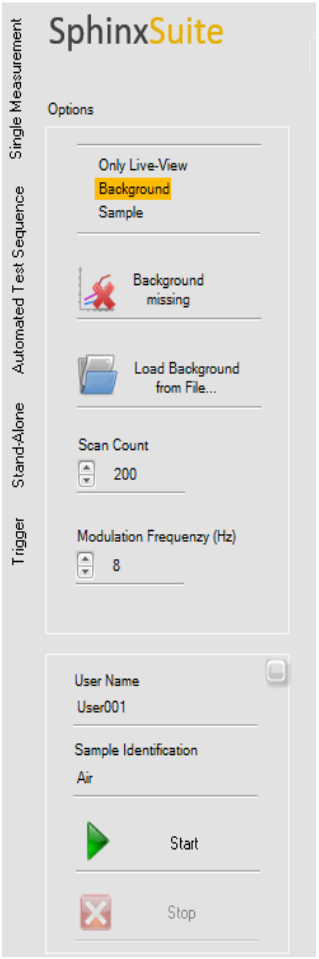


Figure 13: Measurement Options

If you want to cancel the measurement, click ***Stop***. The disabled options will be re-enabled to configure and start a new measurement. *Canceling a measurement is not available when connected via Bluetooth.*

4.3.3 Recording a Sample Measurement

1. In the options section: choose *"Sample"*
2. Enter desired scan count, tester's name and Sample identification
3. Click ***Start*** to start the measurement
4. After the light sources have warmed up (status bar string: *"Warm Up Infrared Sources"*) the status bar will show a progress bar and the estimated time remaining for the measurement to complete.
5. When the measurement has finished and you checked the option ***Configuration > Measurement Complete Acoustic Signal*** in the menu bar a signal indicates the end of measurement.
6. After the sample measurement has finished, the sample spectrum will be displayed. This is computed using the most recently acquired background spectrum.

If you want to cancel the measurement, click ***Stop***. Any options which are disabled will be re-enabled to configure and start a new measurement. *Canceling a measurement is not available when connected via Bluetooth.*

4.3.4 Viewing the Raw Detector Response

If you wish to view your spectrometer's raw detector response you can do so by selecting *"Only Live-View"* in the options menu. You will need to stop the live view by clicking ***Stop*** in order to record a background or sample measurement. In firmware version 0.0.8.9 or lower the "*Live-View*" mode will be stopped after 10 minutes.

4.4 OTHER OPTIONS

4.4.1 File Loader

The SphinxSuite **File Loader** is used to load files from your computer. Figure 14 shows the user interface of the **File Loader**. The screen can be separated into the following parts:

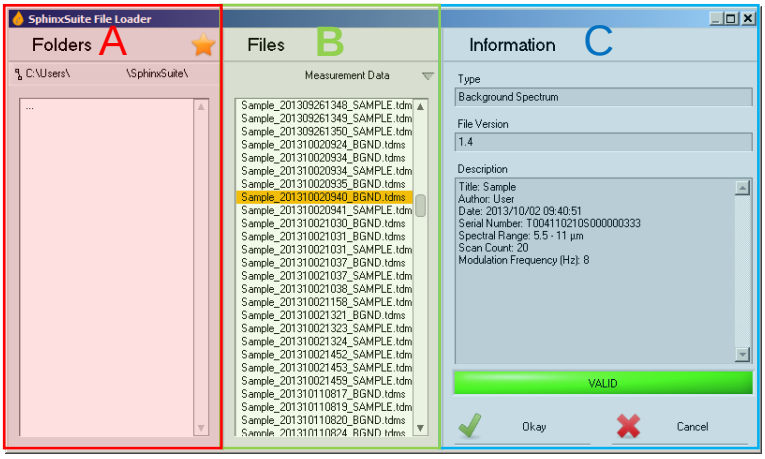


Figure 14: File Loader

A Folder Selection
Navigate through the folder list with your mouse: double-click an entry to open the folder or ***"..."*** to step out of the folder. You can also manually enter a folder path.

B File Selection
You can select a file by left clicking on it. The file filter is set automatically but can be changed using the filter configuration click the down-button. When importing files, you can load multiple files at once. Hold CTRL or Shift while choosing your samples to select multiple files.

C Data Information
If the file format is supported by SphinxSuite, information about the selected file is displayed in this section.

If the file type is appropriate for the chosen import function, the indicator changes from invalid to valid and data can be loaded with ***Okay***.

If you would like to be able to quickly access to your current folder, you can add it to your favorite list. Right-click on the folder you wish to add and select the option "*Add to Favorites*". Enter the name of the entry that should be shown in the favorites list. To show your favorites, click the favorite's button. To remove an entry from the favorites, right-click the favorites list and choose "*Remove selection from favorites*". If you would like to return to the folder directory, you can do so by clicking on the list button. Table 2 shows the existing file extensions supported from SphinxSuite.

File Extension	File	Application
tdms	Technical Management Data Streaming	Background Spectrum Infrared Spectrum Prediction Report Local Database
csv	Comma-Separated Values	Background Spectrum Infrared Spectrum Parameter Concentrations Model Builder Exports
sts	SphinxSuite Test Sequence	Automated Test Sequence
clc	SphinxSuite Classification Class	Classification Model Builder
cmdl	SphinxSuite Classification Model	Sample Classification

cqb	SphinxSuite Quantification Boundaries	Quantification Model Builder
cqm	SphinxSuite Quantification Model	Quantification Model Builder Quantification Predictor Online Quantification Prediction
cprj	SphinxSuite Quantification Project	Quantification Model Builder
ccm	SphinxSuite Channel Fitting Model	Quantification Channel Fitting Model Builder
six	SphinxSuite Similarity Model	Similarity Model Builder

Table 2: File extensions

4.4.2 Spectrum Viewer

If you would like to quickly compare your sample spectra, you can open your most recently acquired spectra in a new window. To do so, right-click the spectrum graph and select "*Add to Spectrum Viewer"* in the context menu. The most recently acquired spectrum will be stacked alongside any previously undocked spectra (Figure 15). This window can also be reached from the menu bar: ***Toolkits > Spectrum Viewer….*** You can import the last spectra when you click on the Button ***Import last Measurement***. Or you can add previously saved spectra to the window by selecting ***Add Measurement Data*** to open the SphinxSuite **File Loader**.

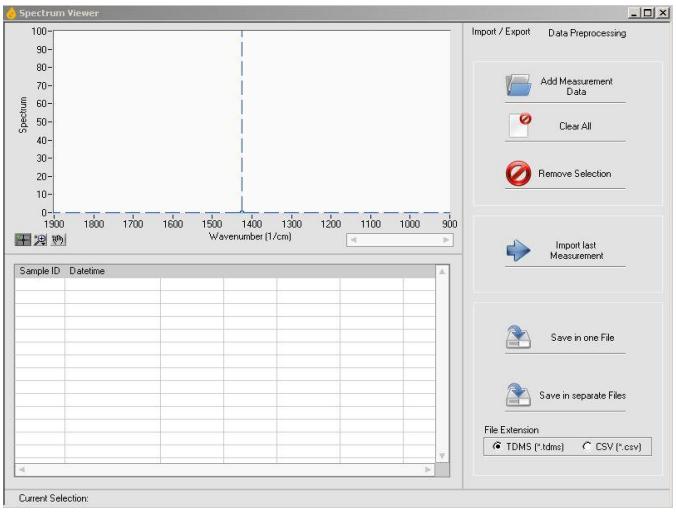


Figure 15: Spectrum Viewer

The table on the bottom of the Spectrum Viewer contains the names and acquisition time of all loaded spectra. Selecting an individual sample and the sample will be highlighted. You can delete one or more samples by selecting them in the table and clicking ***Remove selection***. You can quickly delete all samples by selecting ***Clear all***.

You can leave the window opened while taking further measurements. When you want to add a new transmittance spectrum the window will be closed and re-opened automatically. If you close the dialog your data will remain in the working space and will only get deleted when selecting ***Clear all***.

4.4.3 Output Files

The default path for measurement data is:
"C:\Users\Public\Documents\SphinxSuite\Measurements "
To change your output files folder, click ***Configuration > Output Files…*** in the menu bar. You can also right-click the output files folder indicator in the status bar and select "*Change Folder…"* to open the output files dialog (Figure 16). Select *"Explore…"* if you want to open the output files folder in the Windows Explorer.

Choose between the default output folder and a custom directory. The program automatically checks write permissions in the chosen folder if you do not have administrator rights.

You can choose between two output file formats: TDMS (Technical Data Management Streaming) and CSV (Comma Separated Values) format and a PDF format for generated reports. You can save to all formats by simultaneously having the outputs selected. To generate a report as a PDF you need also to activate TDMS format.

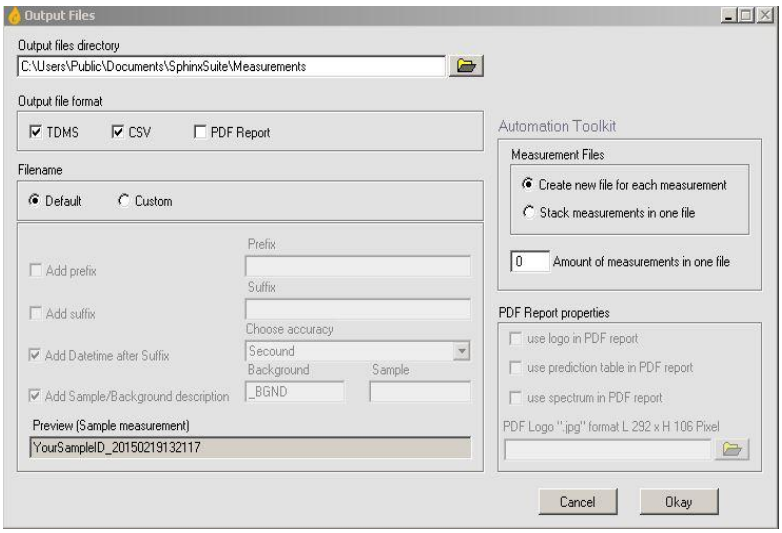


Figure 16: Outputfiles

Additionally you can configure the filename of your exported measurement data. The main part of the file name is the entered Sample ID in the main window. You can add a customized prefix, suffix, date of the measurement and a Sample/Background description for a better distinction of created files. You can choose between four different accuracies of the date: day, hour, minute and second. It is appended after the suffix you entered above. Under *Preview* you can see an example of your configuration. The Sample ID is there set to *"YourSampleID".* By default, the date is added with accuracy *"Second"* and a Background measurement is assigned with the description *"_BGND"*

You can select several options which should be shown in the generated PDF report. You can add a logo in the PDF report by activating the check box and choosing a JPG file in suitable dimensions like L292xH106 Pixel. You can also add a prediction table and the spectrum graph if you activate the check boxes.

If you own the **AUTOMATION TOOLKIT** (section 5) you can save all measurements from an automated test sequence to one file to help you keep track of your data. You can enter how many measurements will be stored before a new file is created under *"Split File after Measurements"*.

The following example shows which files will be saved in which directory.
C:\Users\Public\Documents\SphinxSuite…

- … Measurements

File Name	Meaning
YourSampleName_timestamp.tdms	Sample Measurement
YourSampleName_timestamp.csv	Sample Measurement
YourSampleName_timestamp_BGND.tdms	BGND Measurement
YourSampleName_timestamp_BGND.csv	BGND Measurement
TR_Analysis report_Timestamp.pdf	Trend Analysis Tool Report
TR_YourSampleName_Timestamp.pdf	PDF Report Sample Measurement
PR_YourModelName_timestamp.csv	Online Prediction Report
Prediction_Report_timestamp.tdms	Online Prediction Report

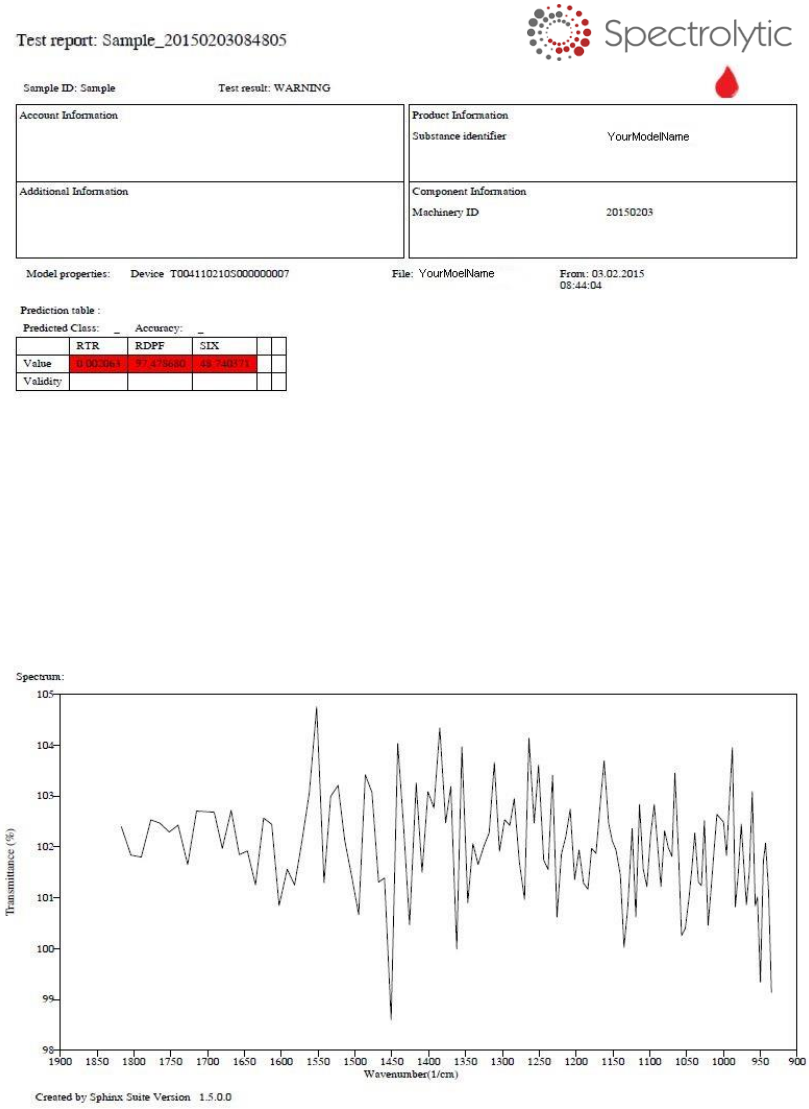
- ... Files\Saved_Templates

File Name	Meaning
Default.rgt	Default Template
YourTemplateName.rgt	Your Template

- … Database

File Name	Meaning
PR_Database.tdms	Prediction Database

The following example shows how a PDF Report looks like if you have selected PDF Report with Logo, Prediction table and Spectrum.


Test report: Sample_20150203084805

Spectrolytic

Sample ID: Sample Test result: WARNING

Account Information	Product Information Substance identifier: YourModelName
Additional Information	Component Information Machinery ID: 20150203

Model properties: Device T00411021OS000000007 File: YourModelName From: 03.02.2015 08:44:04

Prediction table :

Predicted Class: _ Accuracy: _

	RTR	RDPF	SIX
Value	0.003063	97.479680	48.740371
Validity			

4.4.4 Measurement Complete Acoustic Signal

If the option ***Configuration >Measurement Complete Acoustic Signal*** is set then a noise will sound after the measurement has completed.

4.4.5 Expiration of Background

The emissivity and responsivity of your spectrometers emitter and detector can subtly shift over time which can slightly perturb your sample spectra. This effect can be mitigated by periodically recording a fresh background measurement. To help remind you that you need to update your background you have the option set a background expiration alarm. To do so, select ***Configuration > Expiration of Background…*** and set a time before you are given a background expiration warning. (Figure 17)

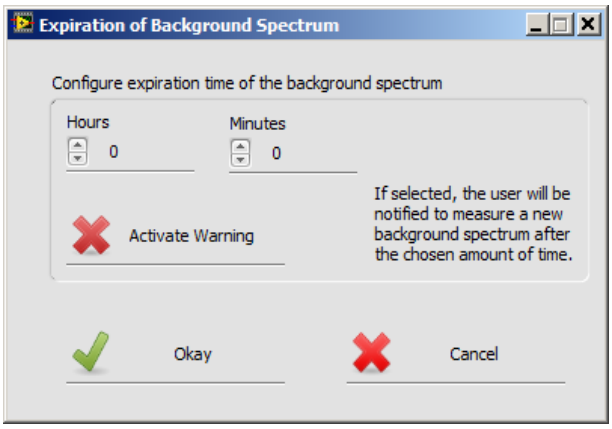


Figure 17: Expiration of Background

When the background has expired you will then get a notification and the indicator ***Background loaded*** changes to ***Background missing***. You can activate this function by changing ***Activate Warning*** to ***Activate Warning*** and confirm it with ***Okay***.

4.4.6 Device Configuration

You can open the device configuration dialog in the menu bar under ***Configuration > Configure Device…***. The device dialogue will give you the following information (Figure 18):

- Serial Number
- Firmware Version
- Spectral Range
- Current connection address or description

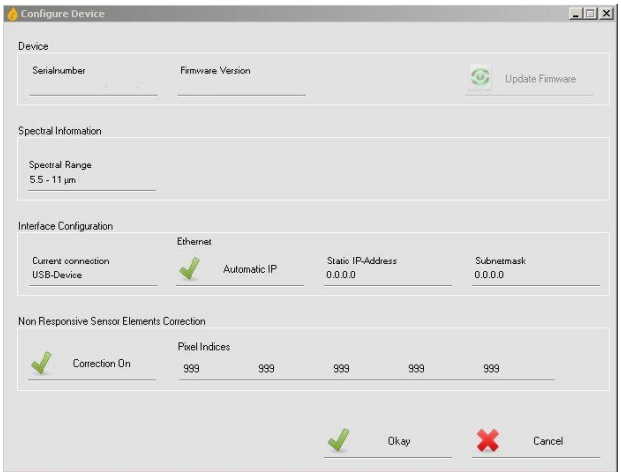


Figure 18: Configure device

The dialog will also give you information about the network configuration of your Ethernet interface. Choose between static and dynamic IP-address and configure your static IP-address. The current state is updated automatically.

Due to the way the sensor unit is manufactured the measurement data might display some non-responsive pixel elements. These can be compensated in the section *"Non Responsive Sensor Elements Correction".* Enter the index of the non-responsive pixel you identified in the data display (D) and switch the indicator to ***Correction On***. Enter the value *"999"* in the data field to ignore a previously entered value. Click ***Okay*** to save changes and send it to the device.

Note
If you are connected to the device via Ethernet and change Ethernet network configuration, you need to reboot (switch off and on again) the device to apply changes.

4.4.7 Eventlog

You can select the ***Eventlog*** tab in the data display to get an overview of SphinxSuite's most recent activity. To reset the table, right click on it and select ***Reset Eventlog***. You are also able to export the table to the clipboard or an Excel file by right-clicking on it and selecting either ***Export > Export Data to Clipboard*** or ***Export > Export Data to Excel***. (Figure 19)

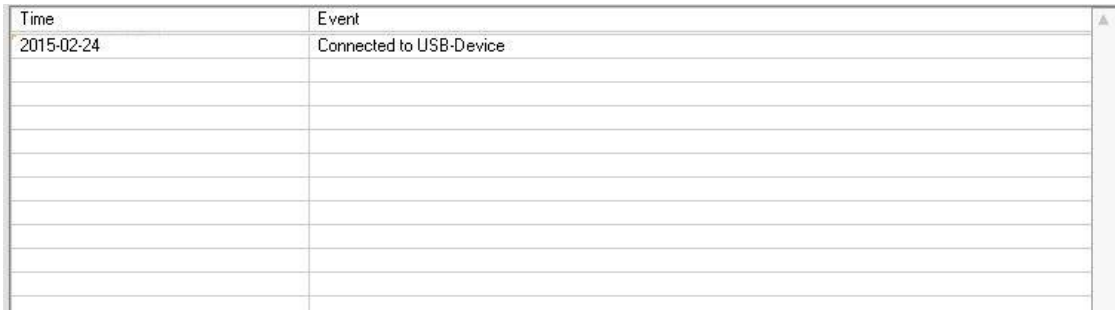


Figure 19: Eventlog

If an error occurs during SphinxSuite's execution (e.g. the connection to device timed out) a ⚠ warning sign will appear in the lower right of the user interface. Clicking on this symbol takes you to the Eventlog where you can get more details about the error.

4.4.8 Datasheets

After a connection has been established there will pop up a warning window dependent on which device you are connected to.
For Transmission devices you will get the message:
Cuvette window made of Zinc Selenide (ZnSe) or Zinc Sulfide (ZnS)
Please read safety data sheet before use!
And for portable devices you will get the message:
ATR crystal made of Zinc Selenide (ZnSe) or Zinc Sulfide (ZnS)
Please read safety data sheet before use! (Figure 20)

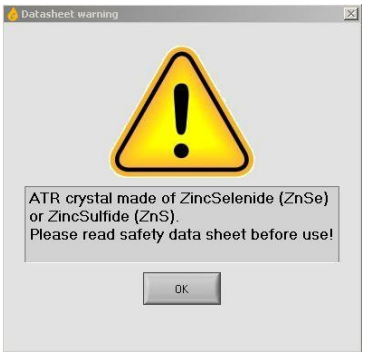


Figure 20: Warning Window

That's just to make sure that you were pointed to read the safety datasheet.

This window is coming up every time you connect to a device, to change this you have to navigate to **Help > Datasheets > activate datasheet warning window**. In the upcoming window you can deselect the check box *Show window on startup*.

To view SphinxSuite datasheets you have to select **Help > Datasheets > Show datasheets**. In the window there is a dropdown menu were some datasheets can be selected. After selection the PDF file will be opened. (Figure 21)

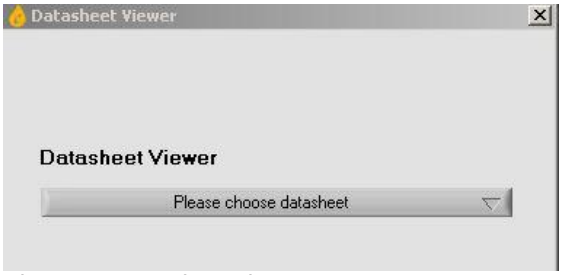


Figure 21: Datasheet viewer

5 AUTOMATION TOOLKIT

5.1 OVERVIEW

The Automation Toolkit allows you to automate your data acquisition process by creating fully automated test sequences adapted to your application. This toolkit can also be used to manually control external hardware peripherals when using in conjunction with the *IRSphinx Extension Board (only available for industrial Transmission Devices)*. Also, you can start and stop measurements by an externally controlled trigger to integrate your IRSphinx spectrometer into a predefined process chain.

The following extensions of the default user interface of Sphinxsuite relate to the **AUTOMATION TOOLKIT**: The menu bar receives an additional entry "*Automation"* and the measurement type selection tabs are extended by *"Automated Test Sequence"* and *"Trigger"*. When selecting *"Automated Test Sequence"*, the options section provides options for automated test sequences and current sequence progress. Selecting the *"Trigger"* tab displays configuration options and status indicators for the *Trigger Mode* (section 5.3).

5.2 AUTOMATED TEST SEQUENCES

The test sequence editor is used to customize your application-dependent test sequences. To start the test sequence editor, switch to tab ***Automated Test Sequence*** and then select ***Create or edit sequence***.

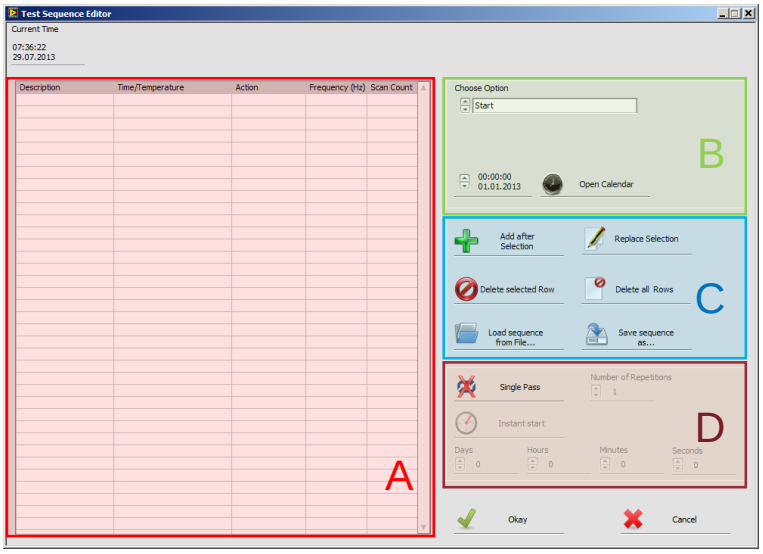


Figure 22: Test Sequence Editor

Figure 22 shows the test sequence editor dialog and can be separated into the following parts:

A Sequence table with all positions
B Choose option to be added to the sequence (A) and its parameter setting
C Tools to modify the current test sequence
D Change repetition count and delay between sequences

5.2.1 Create a test sequence

Step 1: In the options section (B) choose the option you want to add to the sequence and set the desired parameters. The available options and parameters are summarized in the Table 3 beyond.

Option	Description	Parameters
Start	Set the start time and date of the test sequence. This must be the first stage of the sequence.	Timestamp
Wait	This stage waits until the time interval has expired or a specific date is reached before proceeding to the next stage.	Timestamp Interval (HH, MM, SS)
Pump	Start or stop pump (*IRSphinx Extension Board* needed)	On Off
Valve	Open or close valve (*IRSphinx Extension Board* needed)	On Off
Measure Background	Measure a background spectrum (Modulation frequency available with Option A (section 10)	Modulation frequency (Hz) Scan Count
Measure Sample	Measure sample transmission spectrum (Modulation frequency available with Option A (section 10)	Modulation frequency (Hz) Scan Count

Table 3: Test sequence variables

Step 2: Click ***ADD*** to add a sequence stage after the last element of the sequence.

Repeat Steps 1 and 2 to until your test sequence is completed. To remove a stage from the sequence, left-click into the desired stage and choose ***Delete selected Row***. Select ***Delete all Rows*** to completely clear the sequence. To change an entry in the sequence select the desired stage, choose new option (B) and click ***Change selection to chosen option***.

You can import and export your sequences using the controls ***Load sequence from file…*** and ***Save sequence as…*** .

In section D you can configure your test sequence to repeat. Choose between single and multiple passes and set the number of repetitions. You can also select the starting time of a repeated sequence: If ***Instant start*** is chosen, the sequence will be restarted immediately after the last sequence stage was executed. To restart the sequence with a delay relative to the previous starting point, select ***Delayed start*** and set the delay time.

After you have finished configuring your test sequence, click ***Okay*** to apply or ***Cancel*** to discard all changes.

5.2.2 Start and stop a test sequence

To start a sequence select ***Automated Test Sequence*** in the selection tabs and press ***Start*** (C). The indicator will change from ***Sequence stopped*** to ***Sequence running*** and two sliders will show the current position and loop number of the test sequence. (Figure 23)

To get more information about the active sequence select ***Show current test sequence***. The dialog shows the test sequence with the current position highlighted and updated dates.

To stop an automated test sequence press ***Stop*** and confirm your choice. You can now decide whether to stop the sequence immediately or after the current loop. If you lose connection to your device and reconnect, the automated test sequence will go on running.

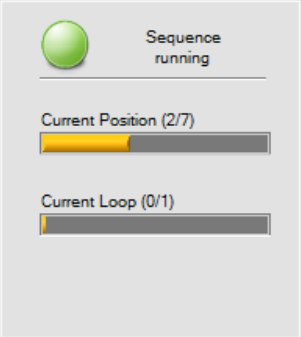


Figure 23: Test Sequence running

5.3 TRIGGER MODE

Trigger mode must be used in combination with the *IRSphinx Extension Board* (only available for industrial Transmission Devices). Please refer to the extension boards' documentation for technical specifications and trigger setup requirements. SphinxSuite's trigger mode is designed to help you integrate your IRSphinx spectrometer into a predefined process chain.

You can use trigger mode to start and stop measurements using an externally triggered source. The data collected using this mode will be stored as though you were making manual measurements. The data can also be used with the prediction toolkit for online process control.

To initialize the trigger mode module, navigate to the ***Trigger Mode*** tab under measurement type selection. To configure and start ***Trigger Mode*** select ***Configure Trigger Mode*** to activate a configuration window (Figure 24).

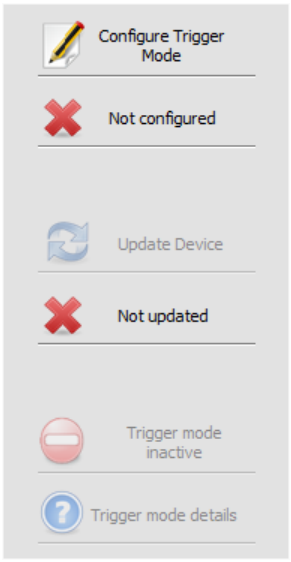


Figure 24: Trigger Mode

You can select if a measurement starts or stops at rising or falling edge and set amount of "end" edges that will be ignored before the measurement ends. The preview window displays the current trigger settings. The green rectangle shows measurement time (Figure 26 show an example of trigger configurations).

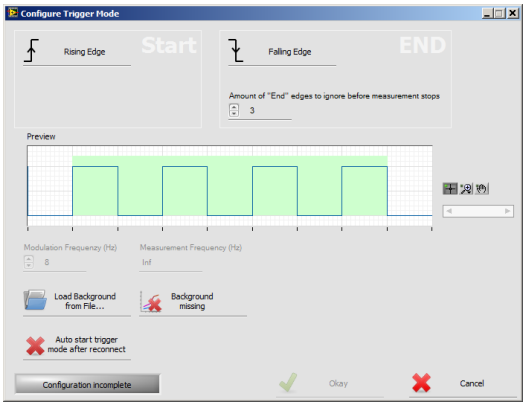


Figure 26: Configure Trigger Mode

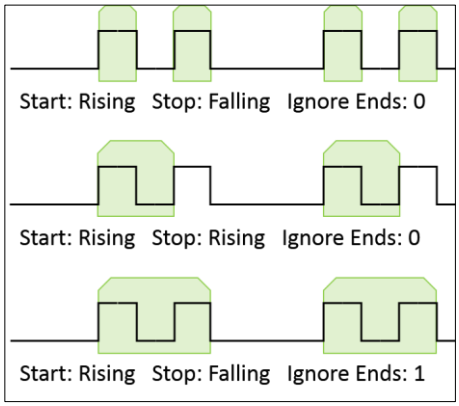


Figure 25: Edges Trigger Mode

If the *Modulation Frequency Option* is licensed and you are connected to a device, you can change the modulation frequency from its default 8Hz value.

When trigger mode is fully configured the indicator on the bottom left of the configuration window will read "*Configuration Complete*". You should then click ***Okay*** to save the configuration and go back to the main window. You will then need to transfer your configuration to your device by clicking on ***Update Device.*** An indicator (✓/✗) will show you if transfer has been successful.

Once you have updated your device you can define the name of the measurement run and the tester under "*Sample Identification*" and "*User Name*". Clicking Start activates the trigger mode and will cause the indicator ***Trigger Mode inactive*** to switch to ***Trigger Mode active***. You will also be able to view the configuration, starting date, sample identification and tester's name by selecting "Trigger mode details" when Trigger mode is active.

Note
▪ In order to record a transmission spectrum a background file must be loaded ▪ If you have set the auto start option to active (section 5.5) and "*Auto start trigger mode after reconnect*", trigger mode will start automatically when you open SphinxSuite.

5.4 Controlling External Hardware

Select ***Automation > Control Peripheral…*** in the menu bar to open the dialog to control your external hardware (Figure 27). The following symbols indicate the current state of your hardware peripheral:

- Valve opened:
- Valve closed:
- Pump started:
- Pump stopped:

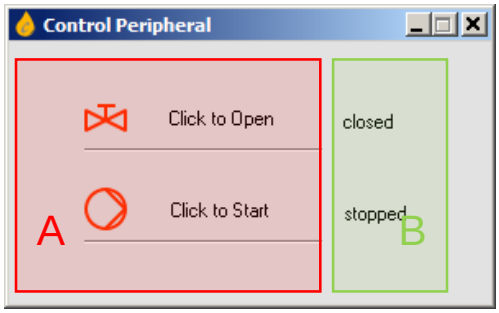


Figure 27: Controlling External Hardware

Status text in the right half of the dialog (B) shows the current state of your hardware.
You can activate the controls in section A to change the hardware state. Once activated, the controls will be disabled until the device acknowledges the command. Afterwards the controls are re-enabled and the hardware state is updated.

Note
To control external hardware you need the IRSphinx Extension Board which is only available for industrial Transmission Devices

5.5 Auto Start

Use the auto start option mode ***Automation > Autostart Mode (only Ethernet)*** to automatically connect to the device when SphinxSuite is started. The connection can only be established via Ethernet. In the first step the device with the serial number of the last successful connection will be searched in the network. Then the IP-address is determined and a connection to this IP-address will be established. This ensures the device can be found if a new IP-address was assigned by your DHCP server. If you have already configured an automated test sequence it will be started automatically.

5.6 STACK MEASUREMENTS IN ONE FILE

To keep track of your measurement data when measuring more samples at a time, you can append new measurement data to an existing file. This is especially helpful when running automated test sequences. Set the option in the output files dialog (section 4.4.3) before starting a measurement. The file will be created after first measurement, with new data appended after each measurement.

6 CHEMOMETRICS TOOLKIT

6.1 OVERVIEW

The **CHEMOMETRICS TOOLKIT** allows you to build customized models for predicting chemical concentrations or classifying samples. The toolkit contains a model builder and a predictor for each option. With the help of the model builder you can easily create chemometric and classification models, which can be used for the predictor. The predictor applies the model to spectra loaded from your hard drive and evaluates classes or concentrations. The models can be used in combination with the **PREDICTION TOOLKIT** (separately available) to carry out predictions during runtime of sample measurements.

The user interface gets extended by the entry ***Chemometrics*** in the menu bar.

6.2 CREATE SAMPLE CLASSIFICATION MODEL

To open the classification model builder, select ***Chemometrics > Classification > Create new Classification Model*** in the menu bar. Figure 28 shows the dialog to create classification models.

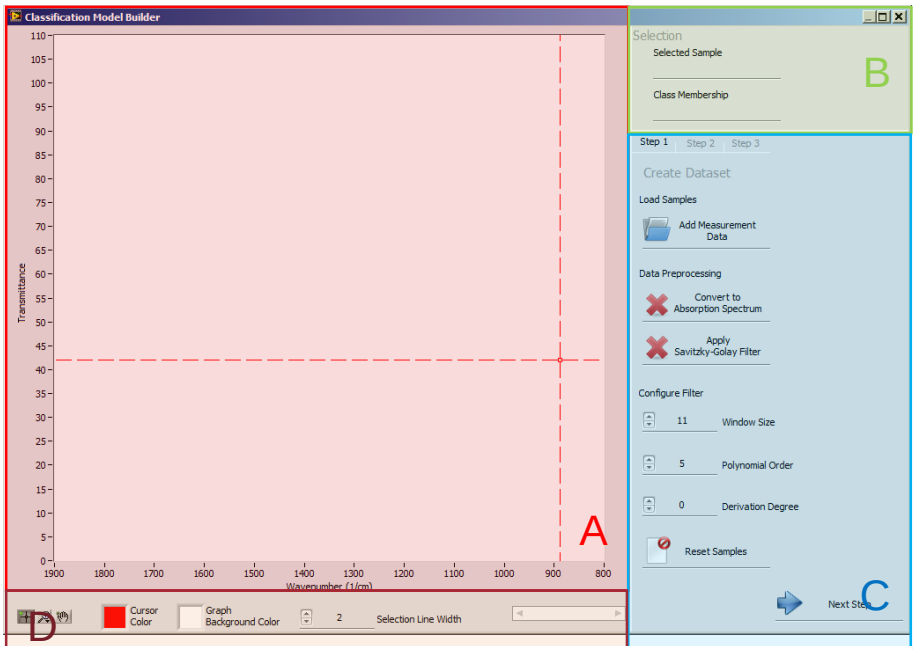


Figure 28: Classification Model Builder

The screen can be separated into the following parts:

A Overview of loaded spectra
B Spectral information
C Options to configure the classification model
D Additional options to change the data's appearance

Follow these steps to create a classification model:

Step 1: Import measurement files

Click ***Add Measurement Data*** to open the **File Loader** and load your measurement data. Imported files will be displayed in the data display (A). You have two data preprocessing options to optimize class differentiation: Conversion to absorbance spectrum and Savitzky-Golay filtering. To apply an option click on the ✖ indicator and it changes to ✓. (Figure 29)

Savitzky-Golay filtering can be applied to reduce spectral noise in measured data. The values of a chosen window size will be approximated by a polynomial of the chosen order. A derivation can be calculated in order to eliminate baseline shift.

Click ***Reset Samples*** to remove imported data from memory.

When you have finished loading the desired data click ***Next Step*** to continue to step 2. You can go back to this dialog later and change preprocessing options or import additional measurement data.

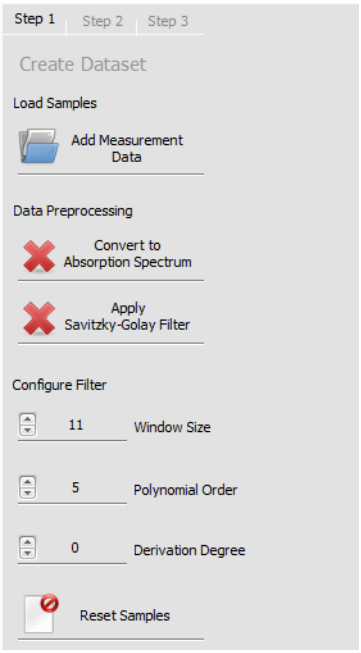


Figure 29: Classification Model Builder Step 1

Step 2: Create classes and assign them to samples

Enter class name and set class color in the ***Navigate Classes*** section (E). The background of the section changes to white if a class is activated. This is done automatically if a class name or class color is set. By default the current class is grayed out and not activated.

Choose a sample in the data display that should be added to the class: drag the crosshair, using the mouse, within the data display to highlight a spectrum. You can also navigate with the up and down keys. The selection area (B) on the upper right of the screen shows the sample description of current selection and the assigned class. Change the cursor and selection color as well as background color of the data display in the lower left of the screen (D).

To add the selected spectrum to the class currently showed in ***Navigate Classes***, click ***Add Selection to Class***. The current sample name will be added to the ***Class Samples*** list and the spectrum in the data display will be given the class color. Repeat this approach to add additional spectra to a class. (Figure 30)

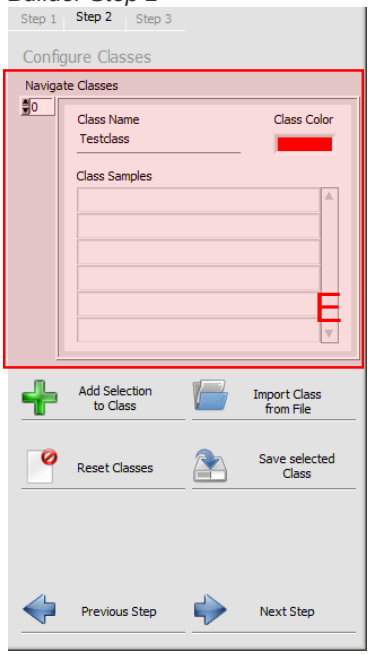


Figure 30: Classification Model Builder Step 2

You can navigate between classes with the increment and decrement buttons on the left side of the ***Navigate Classes*** section or by entering the index of the class (starting from 0).

To save the currently displayed class click ***Save selected Class***. Open the **File Loader** with ***Import Class from File*** and choose the desired classes to import. The data included in the classes are saved without data preprocessing. This enables you to load classes independently

from previously chosen preprocessing options. Data preprocessing is carried out after importing classes.

To reset all classes and reset data display click ***Reset Classes***. Imported data from step 1 and data preprocessing options will be reset.

You can go to ***Previous Step*** to add additional files and change data preprocessing options or continue with ***Next Step***.

Step 3: Build Model

Clicking **Build Model** creates the classification model. The model will be automatically generated and optimized by the toolkit. You can save your model in order to use it later with the **PREDICTION TOOLKIT**.

Click ***Okay*** loads your model and takes you back to the main window. Clicking ***Cancel*** discards unsaved changes and takes you back to the main window.

Note
Prediction of classes is only available for known classes. You have to cover all varieties of classes during construction of the classification model. A mixture of different classes or unknown classes won't deliver accurate results. Classification accuracy may then show a high value even for unknown and incorrectly predicted classes.

Best practice to build model:

1. Split your data set in two parts: training and test data. Training data is used to create the model and test data is used to evaluate the model. The recommended ratio of training data to test data is 3:1
2. Load training data into the classifier and activate both data preprocessing methods with default values
3. Assign classes to desired samples and save created classes in order to re-import them later
4. When finished assigning all classes, go back to Step 1
5. Change data preprocessing parameters to get a larger variance in dataset. I.e.: The better you can separate different classes just by looking at them, the better results will be returned by the model
6. Build your model
7. Evaluate the created model with the Class Predictor (section 6.3) in combination with your test data
8. If evaluation returns bad predictions recreate your model with different preprocessing options

6.3 PREDICT CLASSES

To open the class predictor, select ***Chemometrics >Classification > Class Predictor…*** in the menu bar. Figure 31 shows the dialog to predict classification models. It is structured similarly to the dialog of the model builder. Only the options section (C) differs.

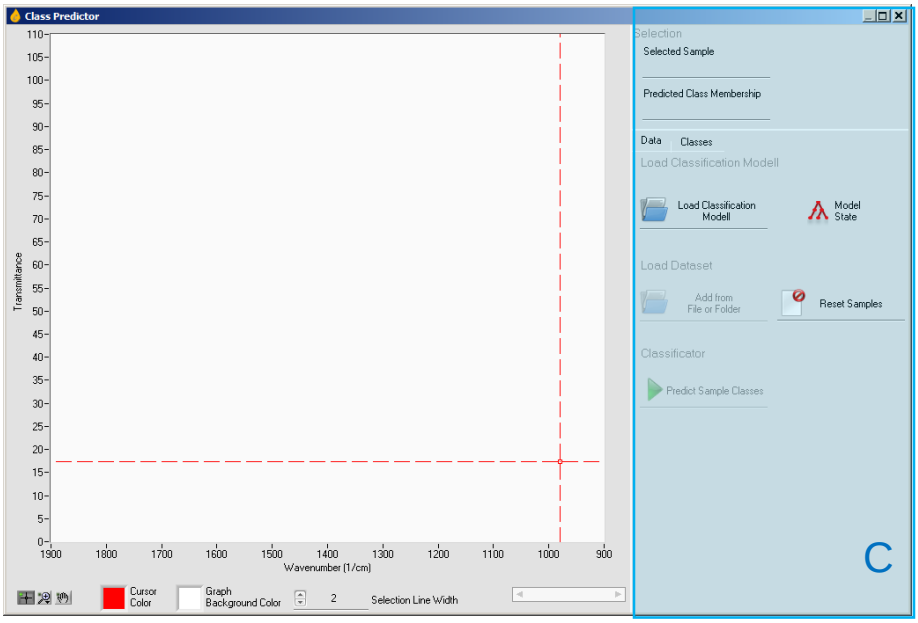


Figure 31: Class Predictor

Note
Prediction of classes is only available for known classes. You have to cover all varieties of classes during construction of the classification model. A mixture of different classes or unknown classes won't deliver accurate results. Classification accuracy may then show a high value even for unknown and incorrectly predicted classes.

To predict classes of your data you first have to click ***Load Classification Model***. If you have created a classification model in the current session, it is automatically loaded. The model state indicator / shows if a model is loaded and **PREDICTION TOOLKIT** is activated.

Note
Loading a new model with class predictor will be applied to the main window as well. If you activated the "Classify Sample" function that comes with PREDICTION TOOLKIT*, the new model will be used to classify measurement data.*

After loading your classification data you can import your sample data. Click ***Add from File or Folder*** and the **File Loader** will open. ***Reset Samples*** will delete all files added.

Click on ***Predict Sample Classes*** to apply the model and classify your data. The ***Classes*** tab will display analysed data: sample identification, predicted class and classification accuracy. You can double-click on a sample to show the classification accuracy.

6.4 CREATE QUANTIFICATION MODEL PLS1

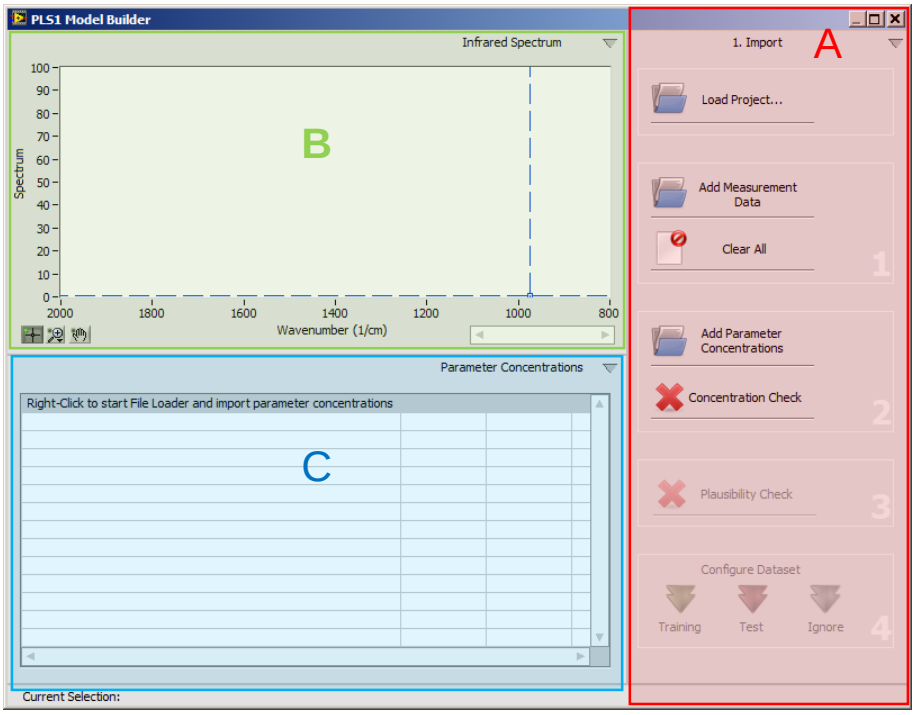


Figure 32: Create Quantification Model

Navigate to ***Chemometrics > Quantification > Create new Quantification Model > PLS1 Regression…*** in the menu bar to open the Model Builder dialog (Figure 32). The window can be divided into 3 parts:

On the right pane current options are displayed: *Import*, *Data Preprocessing*, *Configuration*, *Build Model* and *Export*. On the left pane data is visualized. It can be divided into upper and lower pane (the splitter between the panes can be moved).Select desired visualization from dropdown menu:

- *Infrared Spectrum (with data preprocessing applied)*
- *Parameter Concentrations (for Calibration or Validation)*
- *RMSE (Root Mean Squared Error)*
- *Correlation (R-Square)*
- *Predicted vs. Measured Plot*
- *Prediction Overview Table*
- *Regression Coefficients*
- *Principal Components*

Method for Creating a Model:

6.4.1 Import Sample Spectra

To build a model you first need to import your spectral data. To do so, click on ***Add Measurement Data*** and the infrared spectra will be visualized automatically. You can only import spectra with unique sample names that have been recorded by a single device. For more configuration options, right click on the infrared spectrum to show a context menu:

- Overview Sample Identification: Displays a list containing the names of all loaded spectra and the time they were measured. Duplicate entries are highlighted automatically
- Edit Sample Identification: Edit the sample ID of the selected spectrum
- Remove Selected Sample: Remove the currently selected spectrum
- Colors…: Opens a dialog where you can set the color of the mouse, selected spectrum and selection line width or the background color of the spectrum viewer,

Click ***Clear all*** to remove all previously imported spectra from the model builder.

6.4.2 Import Parameter Concentrations

You need to add parameter concentrations to supplement your sample spectra in order to build your model. You will need to list your parameters concentrations in a .csv file with the following format: (Table 4)

#INFORMATION					File Header	Do not Change
Data Type: Parameter Concentrations						
File Version: 1.0						
#DATA						
Sample Identification	A	B	C	…	Column Header	Comma-separated Values
Sample_1	1	2	3	…	Samples	
Sample_2	4	5	6	…		
Sample_3	7	8	9	…		
…	…	…	…	…		

Table 4: Parameter Concentration File

The following characters are allowed as column separators: “tab”, “comma (,)” and “period (.)”. You should replace Sample_1, etc. with your sample names, A, B, C, etc. with your parameter names and 1, 2, 3, etc. with your parameter values. Once the .csv calibration file is finished click on ***Add Parameter Concentrations*** to load it.

6.4.3 Check Parameter Concentrations

After importing the parameter concentrations file, run ***Concentration Check*** which will highlight any errors with your parameter data. In this window click on ***Check Plausibility*** to detect duplicate sample IDs, empty entries and edit the sample identification and concentration values. Next click ***Check Numerics*** to convert the loaded parameter entries into numerical values. If necessary, you can then change the way that SphinxSuite reads periods (.) or commas (,) to ensure your files load correctly. In order to build a functional model your parameters need to have a variance. You need to click on ***Check Variance*** to ensure this variance is present in your data. If you want to delete a column or row from the list, first select a cell in the column or row. The right-click and choose "*Remove column*" or "*Remove row*" from the context menu. (Figure 33)

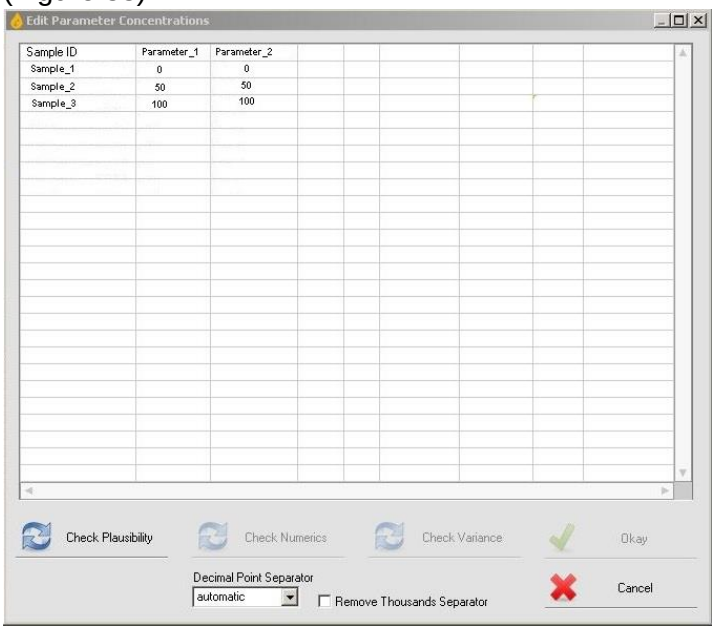


Figure 33: Parameter concentration

You can also enter parameter concentrations manually without importing a file. You have to make sure that the first row contains "Sample Identification" and your parameter names (in this case A, B, C, …) and the first column contains the sample names (here: Sample_1, Sample_2, Sample_3,… while Sample_1 is the name of a spectra you loaded). An example is shown inTable 5.

Sample Identification	A	B	C	…
Sample_1				…
Sample_2				…
Sample_3				…
…	…	…	…	…

Table 5: Manually enter parameter concentrations

When all checks are carried out, click on the ***Okay*** button to apply the changes. The indicator in the model builder window will switch to ***Concentration Check.*** Right click the *Parameter Concentrations* table and select "Export Data to CSV" from the context menu if you want to save the table to your computer.

6.4.4 Plausibility Check

Clicking on ***Plausibility Check*** opens a dialogue that matches the sample names of imported spectra to your parameter concentrations. Any spectra or parameters which are missing will be marked in red and can be removed by clicking on ***Remove missing***. Once you have removed any missing spectra or parameters click on ***Reset*** to reset all changes or you can click ***Okay*** to continue and then carry out a ***Plausibility Check***. The plausibility check can only be performed if the Concentration check has been carried out successfully and all spectra have unique sample IDs. (Figure 34)

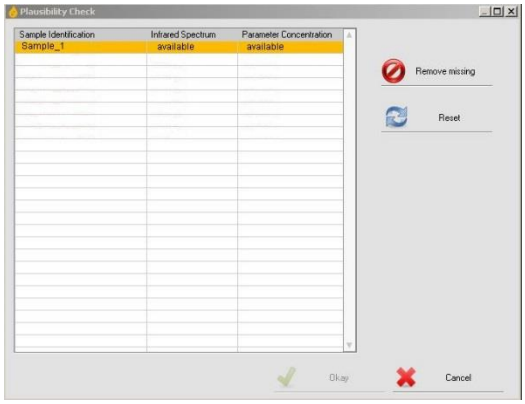


Figure 34: Plausibility Check

6.4.5 Configure data set

When you build your model you will have three options as to how your data is used. A sample can be designated as “training data”, “test data” or simply ignored. To build a reliable, robust model you should always use the most training samples you can. However, there is a drawback – if you don’t have any “test samples”, then it’s impossible to estimate the predictive capacity of your model. Prior to model building, a compromise must be made regarding the proportion of your data used for the training of your model. We recommend that one third of your data is selected as “test data” and that these samples should have evenly spread parameter values. You can define which samples are used to “train” or “test” your model by highlighting the desired samples and selecting ***Training data***, ***Test Data*** or ***Ignored***. (Figure 32 Section A)

6.4.6 Data Preprocessing

Pre-processing your data can significantly enhance the predictive capacity of your final model. Three types of operation can be carried out on your data:

- Absorbance
- Savitzky Golay Filter (Settings: Window Size, Polynomial Order, Derivation Degree)
- MSC: Multiplicative Scatter Correction

To apply the pre-processing operation to your data select "**2. Data Preprocessing"** in the options dropdown menu, choose the operation from the dropdown menu and click ***Add to List***. To change the order of your pre-processing methods you can click on up and down. (Figure 35)

Figure 35: Data Preprocessing

6.4.7 Model Configuration

The configuration stage is where the actual model is computed. To build a parameter model, select "**3. Configuration**" in the options menu and select the parameter of interest in the dropdown list. You then need to choose the maximum number of principle components to use – normally it is easiest to set this to the number of training samples minus one. You then need to select whether or not you wish to center the spectral data before building the model – normally it is best to select this option. You then need to press ***Compute*** to generate the model. Calculation of the model will stop when the maximum number of principle components have been calculated, or when 99.99% of the spectral information is described. (Figure 36)

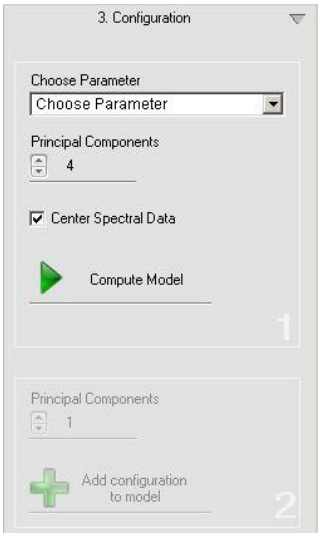


Figure 36: Configure Parameter

You can display the *RMSE*, *Correlation (R-Square)* of your "Test Data" and "Training Data" as a function of total principal components in the lower window by selecting the appropriate option in the lower dropdown menu. You should select the number of principal components your model incorporates to be the lowest number which minimizes the RMSE and maximizes R-Square of your test samples. If there is no minimum, you should choose the value where increasing the number of principal components does not significantly improve the RMSE of your test data. You can view the principal components and associated regression coefficients individually by changing the "*Principal Components*" entry (in box "2") to the component of interest.

To set up a new parameter, return to step 1 where you can add new spectra, parameter concentrations, change data set/preprocessing options and configure the regression model independently from previously configured parameters. Right click the *Prediction Overview* table and select "Export to CSV" from the context menu to save the table of measured and predicted parameter values to your computer.

To build the model with your configuration click ***Add configuration to model***.

6.4.8 Build Model

The build model window can give you an overview of the configuration options you have selected for each of your model parameters. You can use this window to remove a parameter from your model by selecting it and clicking ***Remove Parameter***. When you are happy with your models parameters click ***Build Model*** to reach the final "***Export***" section. (Figure 37)

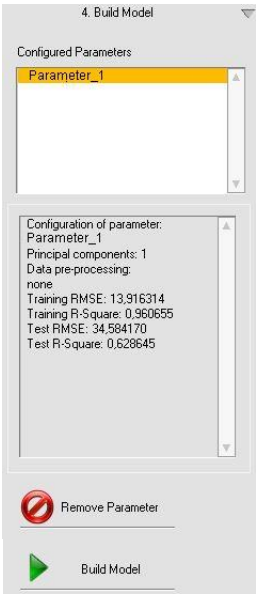


Figure 37: Build Model

6.4.9 Export

If you would like your model to sub divide your models parameter concentrations as “good”, “warning” or “bad”, you need to define concentration boundaries. By default, no boundaries are defined and all parameters are rated as "*Bad*" at any concentration. To change this, click ***Configure Quality Boundaries*** to open the configuration dialog (Figure 38).

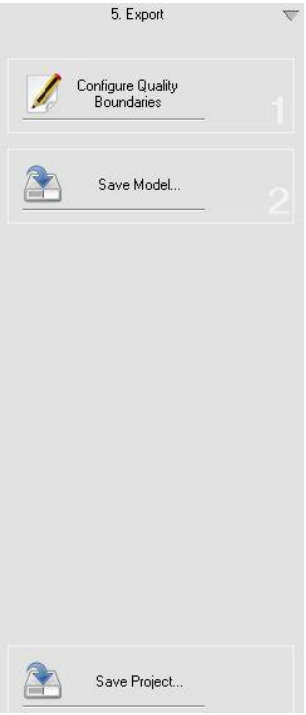


Figure 39: Export Model

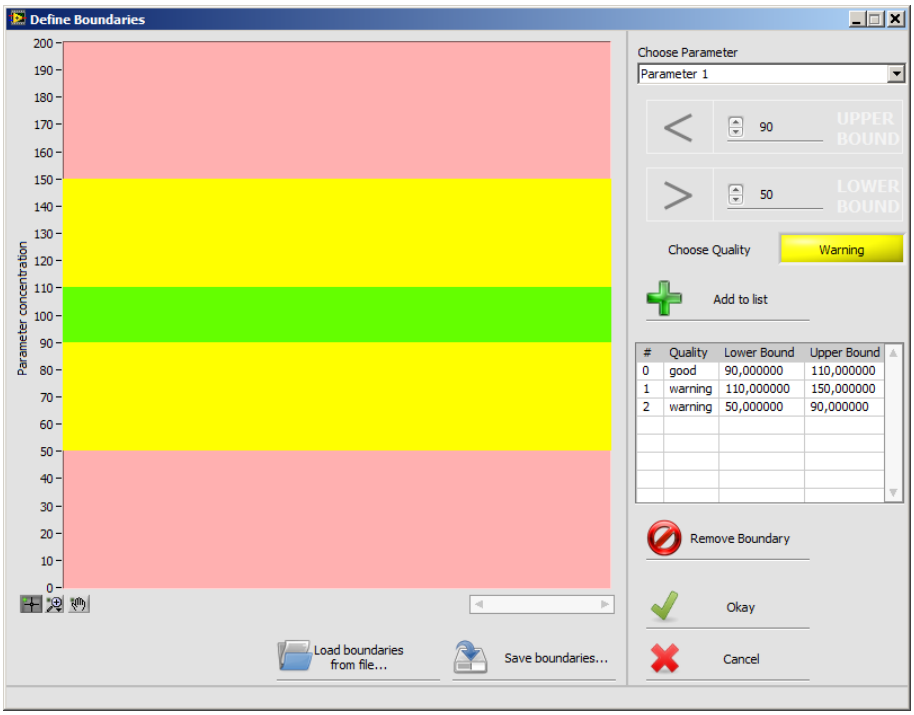


Figure 38: Configuration of Boundaries

You can select a parameter from a dropdown menu in the upper right of the window and select the boundary selection rules with the < (*greater),* > (lower than), ⩽ (*greater or equal to) and* ⩾ (less than or equal to) symbols. If you accidently enter a lower boundary as higher than an upper boundary, or vice-versa, the rule will be discarded. Similarly, if you set your boundary ranges to overlap you will be shown a warning dialogue and the rule won’t be applied. Once a boundary range has been selected you can apply it by clicking ***Add to list***. Boundaries can be defined as *Good* or *Warning* and undefined concentration ranges are treated as *Bad.* Once you have finished, click ***Okay*** to apply the changes or ***Cancel*** to discard new boundaries and keep previously defined ones. There is the option to alter these boundaries by using the **Boundary Builder** which can be found by clicking **Chemometrics > Boundary Builder**.

You can also save your boundaries to enable you to import into future projects. After you finished configuration of quality boundaries you can save your model by clicking on ***Save Model… .*** The model can be used with the **Predictor** which is bundled with the **CHEMOMETRICS TOOLKIT**, or the Online-Prediction Tool with the **PREDICTION TOOLKIT**. Click ***Save Project…*** to save all previously taken steps as a project file that can be imported later if modifications of the model are necessary. (Figure 39)

6.4.10 Load Project

If you have saved a project before (section 6.4.9), you can import it to section "**1. Import"** by clicking on ***Load Project.*** This opens the "**4. Build Model**" window which displays all the parameters the model describes.

If you would like to view the spectral and parameter data that has been used to generate the parameter model, highlight the model and select "*Transfer Data to Import Memory*" from the context menu. The infrared spectra, parameter concentrations, training and test data set, data preprocessing options and regression configuration will then be displayed onscreen. The training/test sample and pre-processing options can be then configured independently for each parameter. If you decide to make any changes, click ***Add configuration to Model*** and choose *Replace* to apply them to the model.

6.5 Create Quantification Model Channel Fitting

Note

Channel Fitting is just for Transmission devices.

Navigate to ***Chemometrics > Quantification > Create new Quantification Model > Channel Fitting…*** in the menu bar to open the Model Builder dialog. (Figure 40)

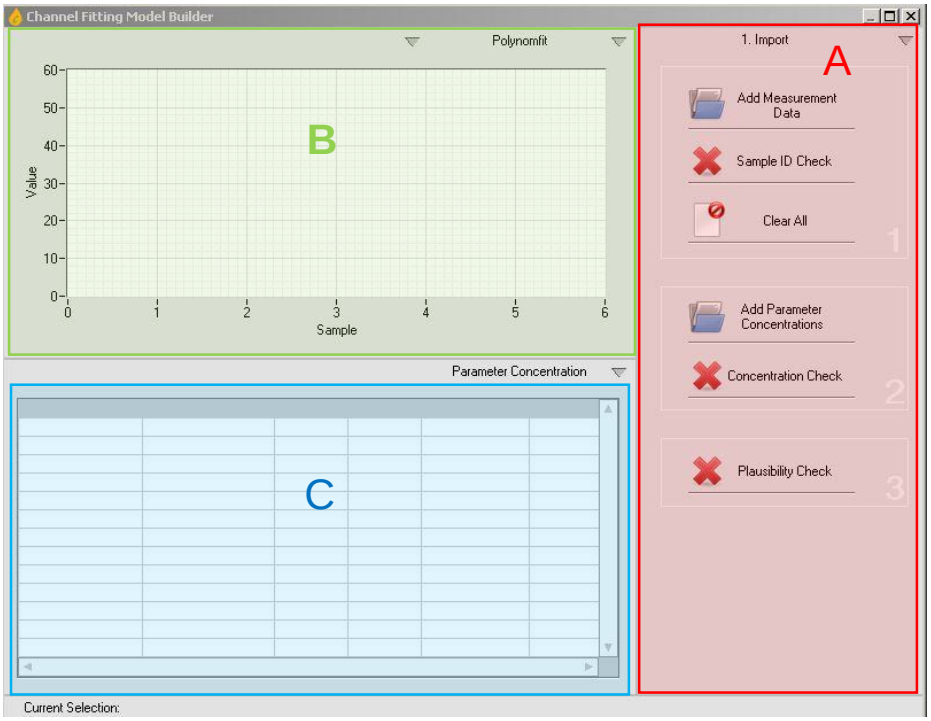


Figure 40: Create Channel Fitting Model

The window can be divided into 3 parts: on the right pane current options are displayed: Import, Channel Preprocessing, Configuration and Export. On the left pane data is visualized. It can be divided into upper and lower pane. Select desired visualization from dropdown menu:

- Channel Values
- Polynomfit
- Parameter Concentration

Method for Creating a Model:

6.5.1 Import Sample Spectra

To build a model you first need to import your spectral data. To do so, click on ***Add Measurement Data*** and the infrared spectra will be visualized automatically. You can only import spectra with unique sample names that have been recorded by a single device. Click ***Clear all*** to remove all previously imported spectra from the model builder.

Click on ***Sample ID Check*** and in the upcoming window clock ***Check Plausibility*** if all Sample IDs are valid you can go on and click ***Okay*** the icon from ***Sample ID Check*** changes from to

6.5.2 Import Parameter Concentrations

Do this the same way as point 5.4.2.

6.5.3 Check Parameter Concentrations

Do this the same way as point 5.4.3.

6.5.4 Plausibility Check

Do this the same way as point 5.4.4.

6.5.5 Channel Preprocessing

Pre-processing your data can significantly enhance the predictive capacity of your final model. In the Channel Selection you can select your channels (the channels values in µm are set automatically due to your added measurement data) and you can do some mathematical operations with your selected channel. You can also select if you want to convert your data to absorption. In the preview you will see the mathematical function of your selection. When you wish to add your selection then please enter a name in the Enter Description field and click on ***Add to List***. The Channel Values table will automatically be updated. (Figure 41)

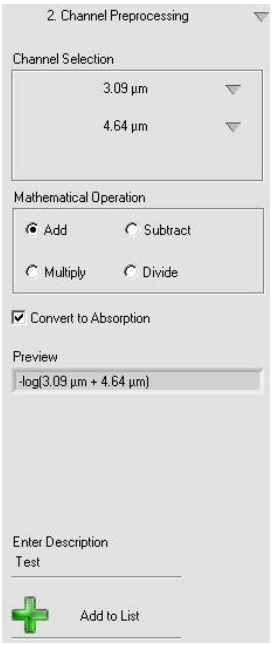


Figure 41: Channel Preprocessing

6.5.6 Model Configuration

In the configuration window you can select your channel and parameter and when you click on ***Compute Model*** you will get a result for RMSE and R-Square. You can change the polynomial order and the shown extrapolation (%) and compute again till you get the results you are looking for. After this enter a description and click ***Add configuration to model***. (Figure 42)

6.5.7 Export

Do this the same way as point 5.4.9

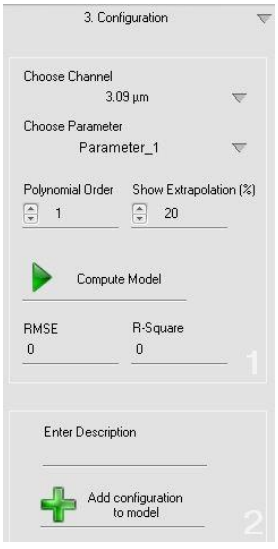


Figure 42: Channel Configuration

6.6 PREDICT CHEMICAL CONCENTRATIONS PLS1 REGRESSION OR CHANNEL FITTING

The quantification predictor is used to predict user defined parameter values from infrared spectra using a model generated by the **CHEMOMETRICS TOOLKIT**. It can also be used to determine predict the "quality" of a parameter using previously defined boundary rules. To open the quantification predictor click on ***Chemometrics > Quantification > Quantification Predictor… .*** This opens the "predictor" window (Figure 43), which can be divided into two parts:

- Right pane: Displays current options
- Left pane: Visualizes the available parameters, infrared spectra, predicted values, Prediction Graph. The visualized parameter can be selected using the windows dropdown menu

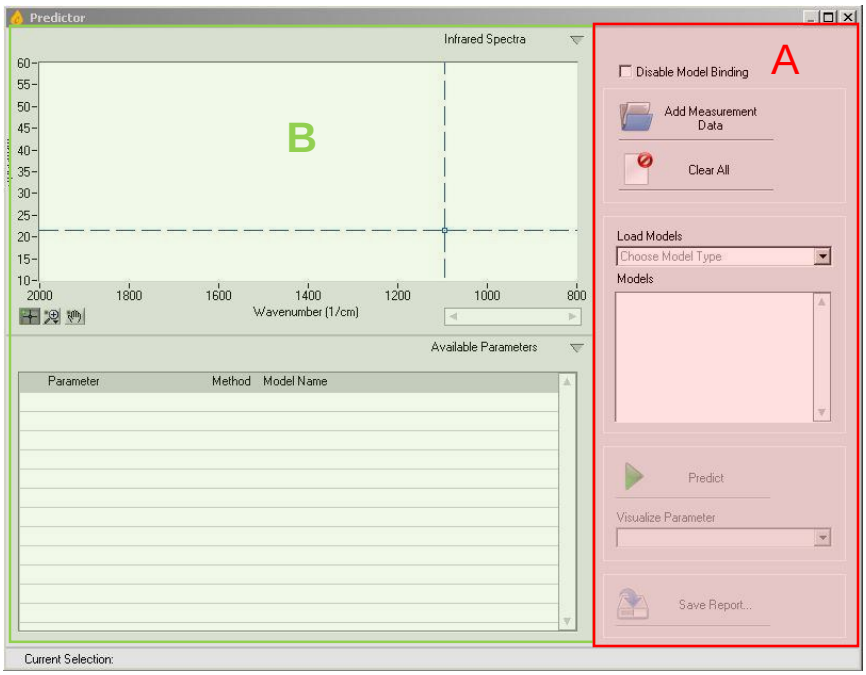


Figure 43: Quantification Predictor

Using the Predictor:

6.6.1 Adding Measurement Data

To load your spectra click on ***Add Measurement Data*** and select your files within the *File Loader*. You will only be able to open files generated using a single device. If you activate ***Disable Model Binding***, you will be able to open files generated with different devices. Your imported spectra will be shown in the *Infrared Spectra* pane, you can select individual spectra with your mouse to display information about it in the status bar. You can select *Colors…* in the right click context menu to change background color, cursor color and selection line width of the spectrum.

6.6.2 Load a Model and Parameters

To load a prediction model created by **SPECTROLYTIC** or with **CHEMOMETRICS TOOLKIT** you should select its model type PLS1 Regression or Channel Fitting from the dropdown menu to open the **File Loader**. Each model is bound to the device used to record the sample spectra used to build it. Successfully loaded models are added to the *Models* list and the *Available Parameters* of the model are shown in a selection list in the lower pane. The selection list contains all the parameters that have been configured during model building, including the model type and the model name. In this checklist you can select the parameters you want to use for prediction. If you would like to remove a model you have loaded, select it in the *Models* list and right click to open a context menu, then select *Remove Selected Model.*

6.6.3 Prediction and Visualization

After loading your spectra and selecting the parameters in the Available Parameters list, you would like to measure, click ***Predict*** to begin prediction. This will enable the *Visualize Parameter* dropdown menu to display all the available parameters. Additionally, a *Predicted Values* table will be shown in the lower pane with Sample IDs, and the infrared spectra's measurement time and predicted parameter values. The parameter values are colored red, yellow and green depending on the boundaries that have been defined in the model. The parameter that has been selected in the *Visualize Parameter* list will be displayed in the *Prediction Graph* in the upper pane If no boundaries have been defined, all values are interpreted as *bad* and spectra will be shown as red. You can left-click a column header to sort the table for ascending values. The *Prediction Graph* will then automatically be reordered as the table.

6.6.4 Export

You can export the values your model has predicted by clicking ***Save Report…*** to open the file dialog. This will save your data as a CSV file (which can be opened by Microsoft Excel) in which the predicted concentration values and their corresponding parameter rating -1 = bad; 0 = warning; 1 = good will be saved.

6.7 CREATE SIMILARITY MODEL

Creating a Similarity Model requires a spectrum of a “good sample”, this will be the same that all the other samples will be compared to, and “bad samples”

To build a Similarity Model, go to ***Chemometrics > Similarity > Create new Similarity Model… .*** and the Similarity Model Builder will be presented which can be divided into two parts. (Figure 44)

- Right pane: Displays current options
- Left pane: Visualizes the available parameters, infrared spectra, predicted values, Prediction Graph. The visualized parameter can be selected using the windows dropdown menu.

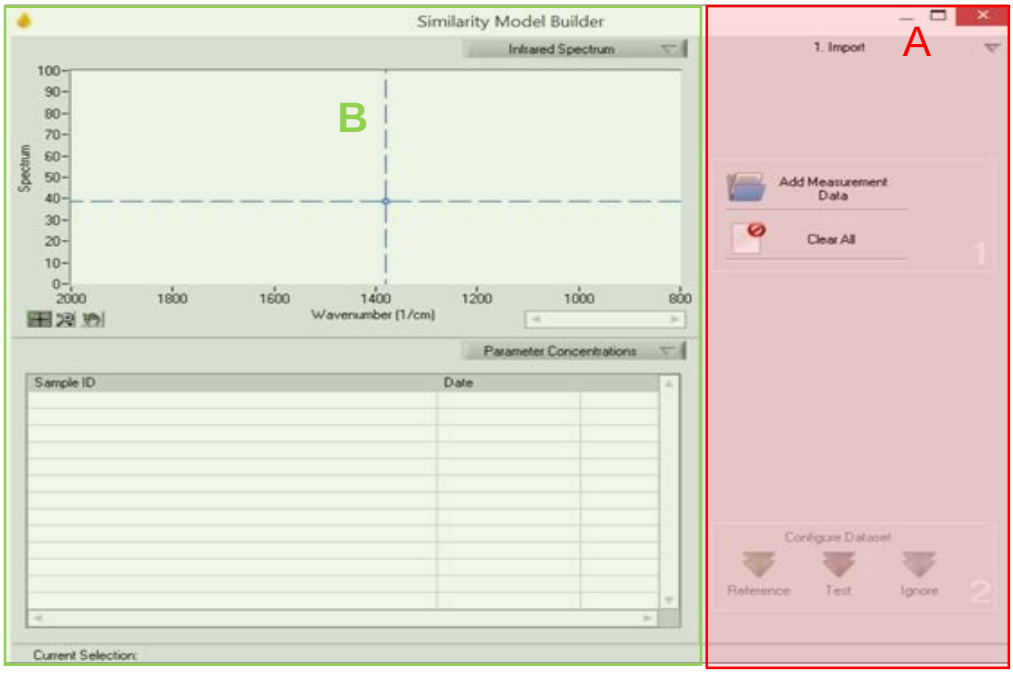


Figure 44: Create Similarity Model

Click on ***Add Measurement Data*** which opens the File Loader and select the spectra of a “good” and “bad” samples. Once they are selected, press ***Okay***

The spectra of the “good” and “bad” samples will be shown in the Similarity Model Builder Window. Select the “good” spectra and click on ***Reference*** which will highlight it in green. Then click on the “bad” spectra and press ***Test***, causing it to highlight red. There is to reject any spectra you do not wish to include in the mode by selecting ***Ignore***.

Click on the tab in the top right of the window, and select **Data Preprocessing**. This will show the preprocessing dialogue allowing you to select any preprocessing technique you wish to use.

- Absorbance
- Savitzky Golay Filter (Settings: Window Size, Polynomial Order, Derivation Degree)
- MSC: Multiplicative Scatter Correction

Then click ***Add to List***. The order the preprocessing techniques are implemented can be adjusted by highlighting a technique and using the up and down arrows. (Figure 45)

Figure 45: Data Preprocessing

Following adding these preprocessing techniques select **Configuration** from the top right hand tab and set the parameter you wish to use for the Similarity Model. There are three options: RTR (Baseline sensitive), RDPF (Peak sensitive) and SIX (overall evaluation). After you have chosen your parameter press ***Compute Model***, and the comparison result will appear under the relevant column. The training spectra will have a result of 100.00000 as it is the ideal "good" sample, and the test "bad" spectra will be less. If the result is ok you can click ***Add configuration to Model***. Repeat this for all parameters you want to have in your model to predict later on. (Figure 46)

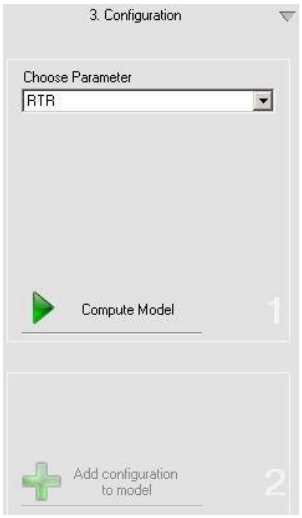


Figure 46: Similarity configuration

Move on the next stage **Build Model**. You will get some information of your parameters which you added to your model. And you have the option to remove any previously configured parameters by selecting it and pressing ***Remove Parameter***. Or, if complete click on ***Build Model***, and you will be taken to the final step – **Export**. (Section 6.4.8)

If you would like your model to sub divide your models parameter concentrations as "good", "warning" or "bad", you need to define concentration boundaries. By default, no boundaries are defined and all parameters are rated as "*Bad*" at any concentration. To change this, click ***Configure Quality Boundaries***. This dialog is exactly the same as Point 6.4.9.

6.8 PREDICT SIMILARITY INDEX

The similarity predictor is used to predict user defined parameter values from infrared spectra using a model generated by the **CHEMOMETRICS TOOLKIT**. It can also be used to determine predict the "quality" of a parameter using previously defined boundary rules. To open the similarity predictor click on ***Chemometrics > Similarity > Similarity Predictor…*** This opens the "predictor" window, which can be divided into two parts: (Figure 47)

- Right pane: Displays current options
- Left pane: Visualizes the available parameters, infrared spectra, predicted values, Prediction Graph. The parameter visualized can be selected using the windows dropdown menu

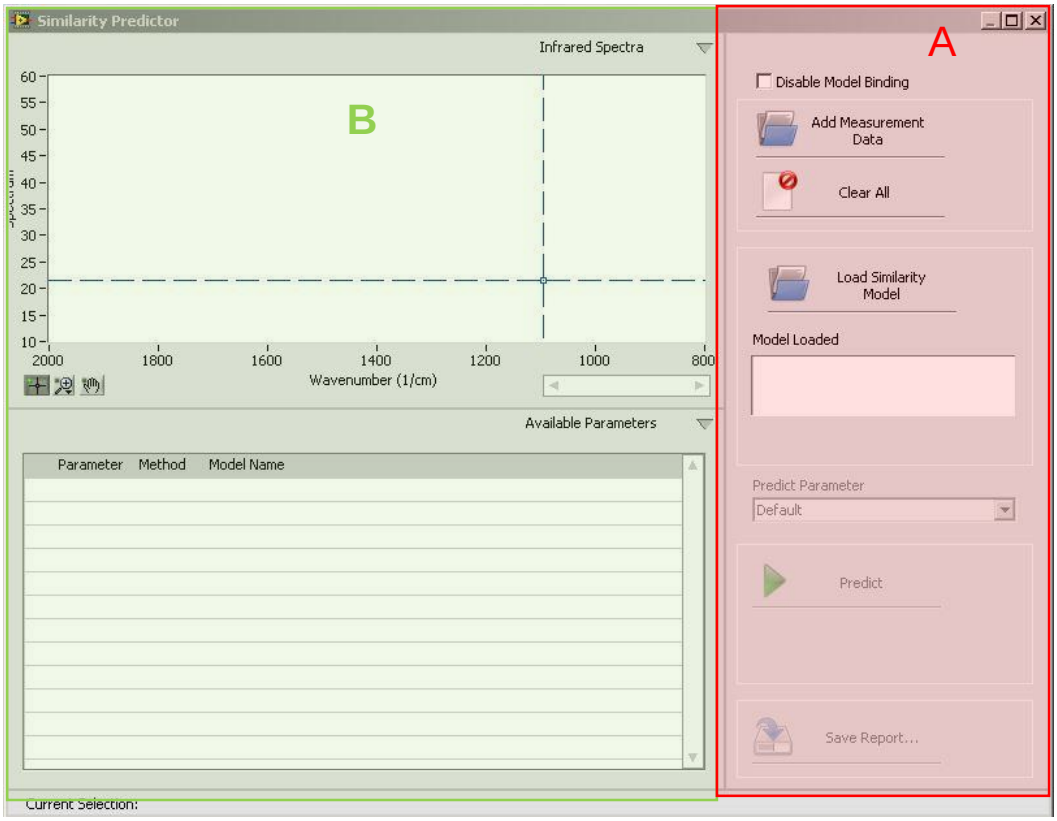


Figure 47: Chemometrics Predict Similarity

6.8.1 Adding Measurement Data

To load your spectra click on ***Add Measurement Data*** and select your files within the *File Loader*. You will only be able to open files generated using a single device. If you activate ***Disable Model Binding***, you will be able to open files generated with different devices. Your imported spectra will be shown in the *Infrared Spectra* pane, you can select individual spectra with your mouse to display information about it in the status bar. You can select *Colors…* in the right click context menu to change background color, cursor color and selection line width of the spectrum. To remove your spectra click ***Clear All***.

6.8.2 Load a Model and Parameters

To load a prediction similarity model created by **SPECTROLYTIC** or with **CHEMOMETRICS TOOLKIT** you can click on ***Load Similarity Model*** to open the **File Loader**. Each model is bound to the device used to record the sample spectra used to build it. If you activate the check box *Disable Model Binding* you can load spectra from a different device. Successfully loaded models are added to the *Models* list and the *Available Parameters* of the model are shown in a selection list in the lower pane. The selection list contains all the parameters that have been configured during model building, including the model type and the model name. In this checklist you can select the parameters you want to use for prediction. If you would like to remove a model you have loaded, select it in the *Models* list and right click to open a context menu, then select *Remove Selected Model.*

6.8.3 Prediction and Visualization

After loading your spectra and selecting the parameters this will enable the *Predict Parameter* dropdown menu to display all the available parameters you would like to measure, click ***Predict*** to begin prediction. Additionally, a *Predicted Values* table will be shown in the lower pane with Sample IDs, and the infrared spectra's measurement time and predicted parameter values. The parameter values are colored red, yellow and green depending on the boundaries that have been defined in the model. The parameter that has been selected in the *Predict Parameter* list will be displayed in the *Prediction Graph* in the upper pane If no boundaries have been defined, all values are interpreted as *bad* and spectra will be shown as red. You can left-click a column header to sort the table for ascending values. The *Prediction Graph* will then automatically be reordered as the table.

6.8.4 Export

You can export the values your model has predicted by clicking ***Save Report…*** to open the file dialog. This will save your data as a CSV file (which can be opened by Microsoft Excel) in which the predicted concentration values and their corresponding parameter rating -1 = bad; 0 = warning; 1 = good will be saved.

6.9 CHEMOMETRICS BOUNDARY BUILDER

There is the option to alter boundaries by using the **Boundary Builder** which can be found by clicking **Chemometrics > Boundary Builder**.

In the dropdown menu Load Model to edit Boundaries you can select the type of model and then the File loader will open. After loading the model you can click ***Reset*** if you want to load different model or you can click on ***Edit Boundaries*** to edit the boundaries for the selected model like described in section 6.4.9. In the Model information window you will get some details of the model you loaded. And in the table below you will get an information if the boundary was found for the parameters in the model. After you finished to edit the boundaries you can click on ***Save as new Model***. It will be created a new model, the old one will net be replaced. (Figure 48)

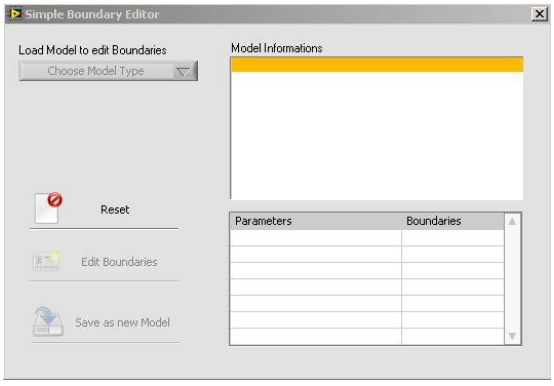


Figure 48: Simple Boundary Editor

7 PREDICTION TOOLKIT

7.1 OVERVIEW

The **PREDICTION TOOLKIT** was built to help you convert your sample spectra into quantifiable, application specific measurements during run-time, i.e. right after a measurement finished. The toolkit can be used to predict the concentration of sample components e.g. determining the concentration of water in an oil. It can also be used to predict if your unknown sample is similar to your master sample. Alternatively, you can assign your spectra to classes e.g. to determine whether a milk sample came from a cow or a goat.

For each type of measurement you will need a chemometrics model that mathematically relates your sample spectra to the parameters and/or classes you are interested in. You can use the **PREDICTION TOOLKIT** to load your own models (created with **CHEMOMETRICS TOOLKIT**) or you can download pre-built models from Spectrolytic GmbH.

If you own the **PREDICTION TOOLKIT** then "*Prediction"* will be visible in the menu bar which you can use to load your chosen model and set choose whether to analyze or not analyze measurements.

7.2 CLASSIFICATION OF SAMPLES

To automatically classify a sample after a measurement you need to check the menu entry ***Prediction > Prediction State > Classify Sample***. To enable the entry, you have to load a classification model by selecting ***Prediction > Load Model > Classification Model*** in the menu bar. You can also click on the classification state indicator to open **File Loader** and load a classification model. If successful, the indicator switches to . After loading a model and setting ***Classify Sample***, the data display will have a ***Classifier*** tab (Figure 49) which gives an overview of the predicted classes as they were defined in the model builder. For more information about how classification works and how to build a classification model, refer to section 6.2

The transmission spectrum is shown in section A, the table B shows the sample name, predicted class and classification accuracy. Double click an entry in the table to highlight the spectrum in the overview and show classification accuracy of the sample in the graph (C). Accuracy is shown as a bar plot. The X-Axis represents the class index as previously defined when building the model. Hover your mouse over the graph to get the corresponding names of classes. A greater difference between the bars shows a better prediction accuracy of

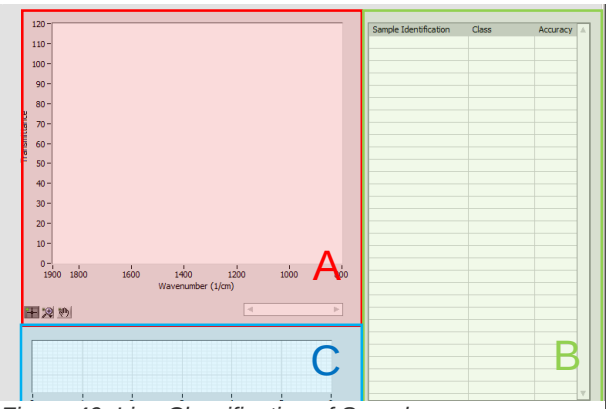


Figure 49: Live Classification of Samples

the sample. When bars become closely spaced, the classification must be treated with caution. The most accurate class is represented by the highest bar.

Right-click on the classification table and select "*Save Data"* to export classification results as csv document. If the auto start and ***Classify Sample*** option are used together, the classification model will automatically be loaded during program start up. The samples will be predicted after each measurement if you are running an automated test sequence.

7.3 Prediction of chemical concentrations PLS1 Regression or Channel Fitting

To automatically quantify a sample after a measurement you need to check the menu entry ***Prediction > Prediction State > Quantify Sample***. To enable the entry, you have to load a Quantification model by selecting ***Prediction > Load Model > Quantification Model*** in the menu bar. You can also click on the quantification state indicator and select quantification to open the **Configure Online Prediction** window (Figure 50). You have to select which model type (PLS1 Regression or Channel Fitting) you want to predict and chose the model type from the dropdown menu. After that the File Loader will open where you have to load your model and select your parameters which you want to predict. If successful, the indicator switches to. After setting ***Quantify Sample***, the data display will have a ***Quantification*** tab which gives an overview of the predicted parameter as they were defined in the model builder. For more information about how quantification works and how to build a quantification model, refer to section 6.4 – 6.5.

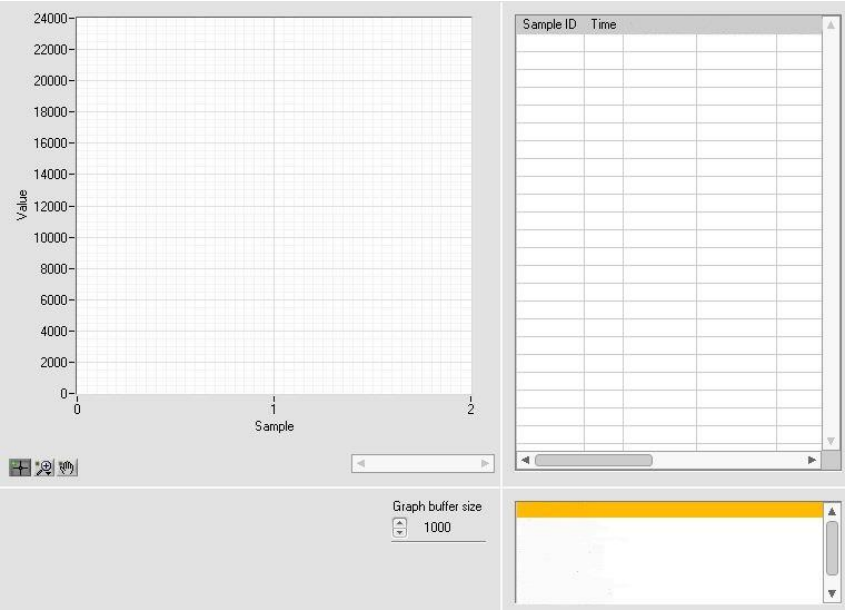


Figure 50: Live Quantification of Sample

7.4 PREDICTION OF SIMILARITY INDEX

To automatically check the similarity of a sample after a measurement you need to check the menu entry ***Prediction > Prediction State > Similarity Sample***. To enable the entry, you have to load a Similarity model by selecting ***Prediction > Load Model > Similarity Model*** in the menu bar. You can also click on the quantification state indicator and select similarity to open the **Configure Similarity Prediction** window (Figure 51). You have to select **Load Model**, what will open the File Loader where you have to load your model and select your parameters which you want to predict. If successful, the indicator switches to. After setting ***Similarity Sample***, the data display will have a ***Similarity*** tab which gives an overview of the predicted parameter as they were defined in the model builder. For more information about how quantification works and how to build a Similarity model, refer to section 6.7

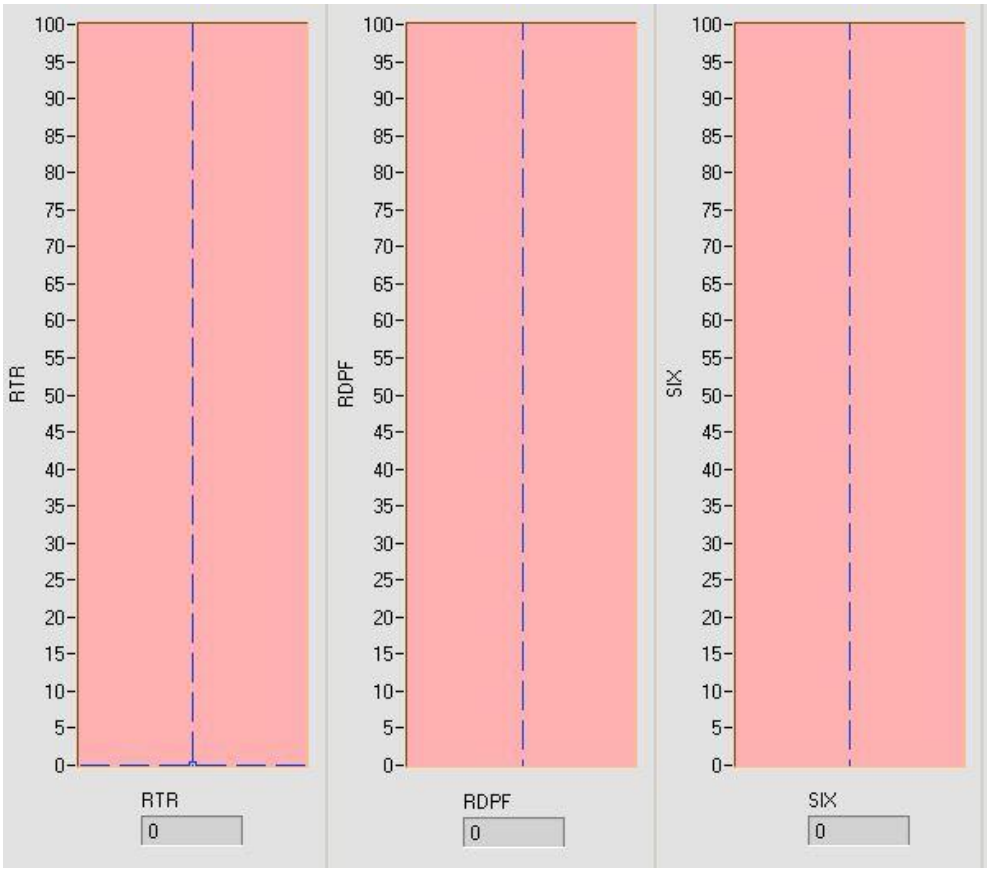


Figure 51: Live Similarity Prediction

8 SIMPLE MODE

8.1 ADDITIONAL INFORMATION AND TEMPLATES

The Additional Information tab (Figure 52) can be selected in the data display, or by using the button in the measurement options section near User Name. The Additional Information tab allows account information, product information, additional information and component information to be entered. When saving as templates, they can be used in both, the Standard Mode and the Simple Mode interface. The information you enter there will be stored in the measurement output files as well. When pressing the cell adjacent to ***Substance Identifier*** the name of the latest model you loaded in the ***Prediction > Load Model >...*** will be inserted. This you will need to use the templates for the Simple Mode.

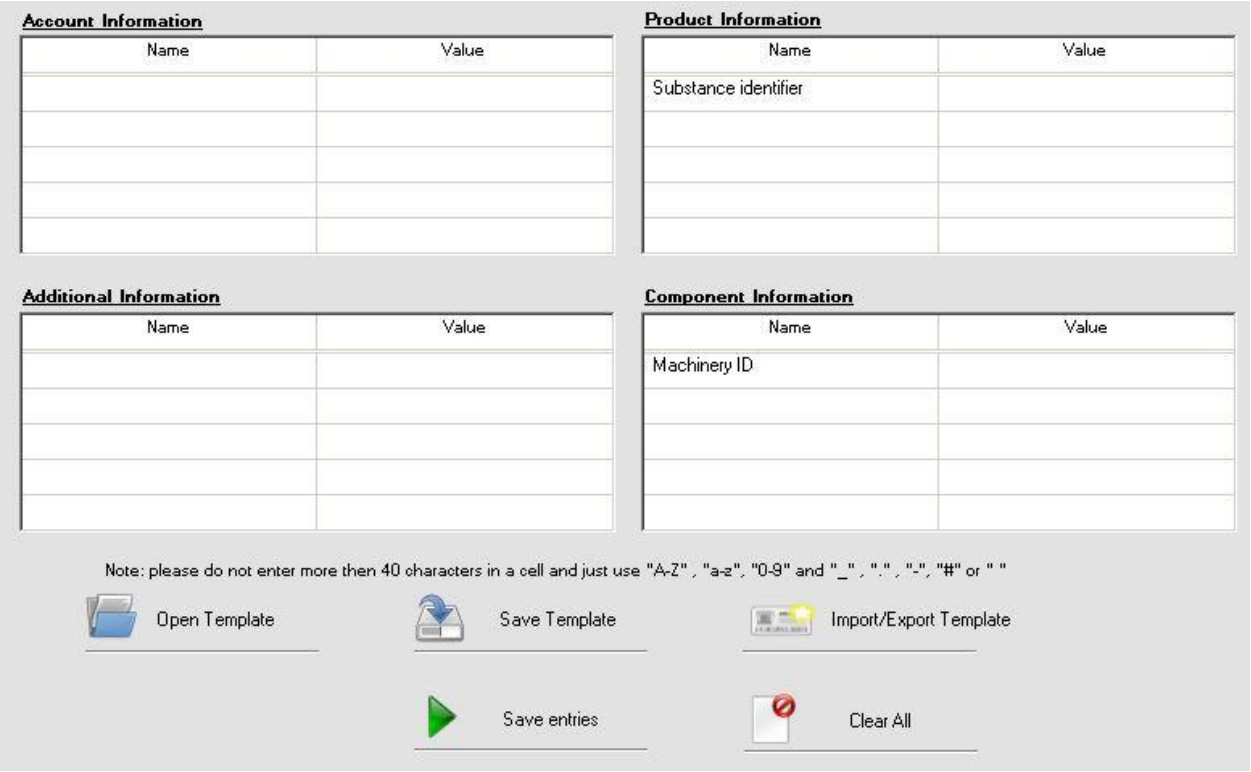


Figure 52: Additional Information

Right click in a cell in column Value will bring up a dropdown menu. When there is some information in this cell, it will be copied to the dropdown menu. If you want to delete an entry in the dropdown menu just right click again on the specific cell and select *delete from dropdown.* All entries in the dropdown are also available in the Simple Mode when you save it as template. With this templates you can quick and easily change your specific additional information from measurement to measurement. To create a template you need to insert your specific additional information and then click ***Save entries***, after that click ***Save Template***. It will come up a dialog (Figure 53) if you want to use your template as Standard

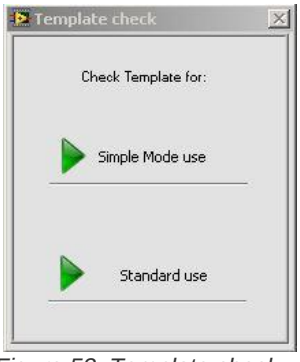


Figure 53: Template check

use or Simple Mode use. (For Simple Mode use you need a model loaded). Then you have to enter the name for your template. To open Templates which you have created click ***Open Template*** and if there has been a model saved it will also load the model in the online prediction tool so that you just need to click ***Save entries*** and you can start your measurement. You can delete all your information by clicking ***Clear All***. When you created templates and want them to share then you can click ***Import/Export Template***. The Edit Template window will be opened (Figure 54) where you can delete templates by selecting the template in the Existing Template list and then click ***Delete Template***. If you want to export your template then select ***Select Export Folder***. Just navigate to your destination folder and click select folder. The Button ***Export Template now*** will be enabled and if you click it your data will be saved in a new folder with the format: YourModelName_export_data at your destination. In this folder you will find the template file (rgt format) and the specific model file. Copy this folder to your other system where SphinxSuite is installed and navigate there to the same Edit Template window where you can click ***Import Template***. Navigate to your imported folder and click on select folder. After that you will be asked where you want to save your model (Please don't move models to a different location after a template is created, you will get an error if you try to load that template again because the assignment is lost. Use always Import and Export template to move files which belong to your template)

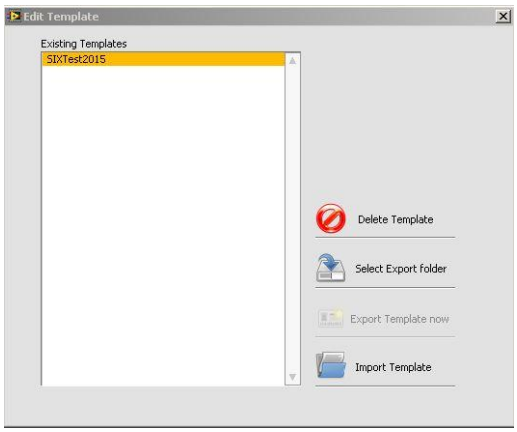


Figure 54: Edit Template

8.2 SIMPLE MODE

Note
Simple Mode is part of the Prediction Toolkit

Simple Mode is a simplified version of the SphinxSuite interface which can be used by untrained users. It's split into two parts, an administrator part where the measurement is set up, and a simple part which is to be used by untrained users.

- **Administrator part**

First of all you have to create a Template (section 8.1) and you have to do the settings for the output files (section 4.4.3)

Once Simple Mode is started you need to enter a password to come back to the main window. This password you can edit in the menu bar ***Prediction > Simple Mode > Simple Mode Password***. (Figure 55) Enter your old password and then the new one and then re-enter the new password. Click on ***Confirm*** if your settings are correct or click ***Cancel*** if you don't want to change your password. You can click ***Reset to system default*** to set the password back to its default = SphinxSuite

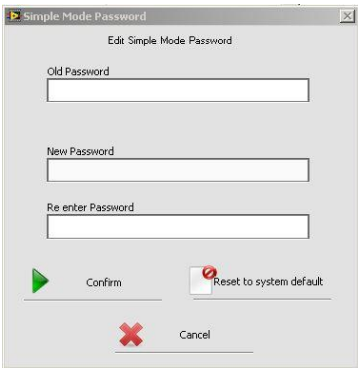


Figure 55: Simple Mode Password

You can also set a connection as standard for the Simple Mode. After Simple Mode is started the software tries to connect to your selected connection. To do so go to the menu ***Prediction > Simple Mode >Simple Mode Auto Connect***. (Figure 56)

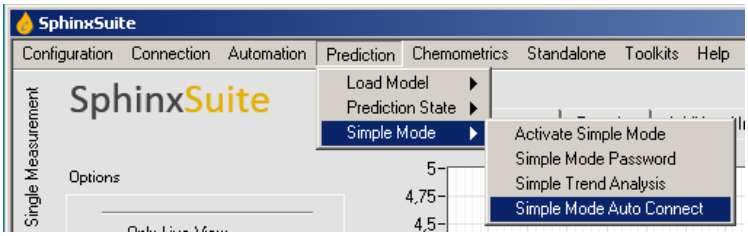


Figure 56 Simple Mode Auto Connect

The following window will only appear if you are connected to a device. (Figure 57)

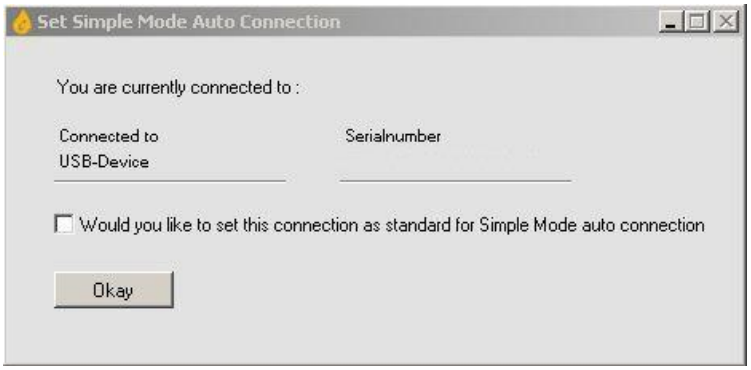


Figure 57 Set Simple Mode Auto Connection

Here you will see the Serial number and which device you are currently connected to. If you activate the checkbox then your currently connected device will be set as standard for the Simple Mode.

After all settings are complete you can select ***Prediction > Simple Mode > Activate Simple Mode*** this will activate the Simple Mode. The program will be closed and you have to restart it. Then you will be presented with the Simple Mode interface.

- **Simple part**

The Simple Mode interface can be separated into two parts (Figure 58):

A Activation buttons
Clicking on the buttons will bring you to the corresponding tab and can launch events.

B Configuration options and display
Here you can make some configurations or you will get some result displays.

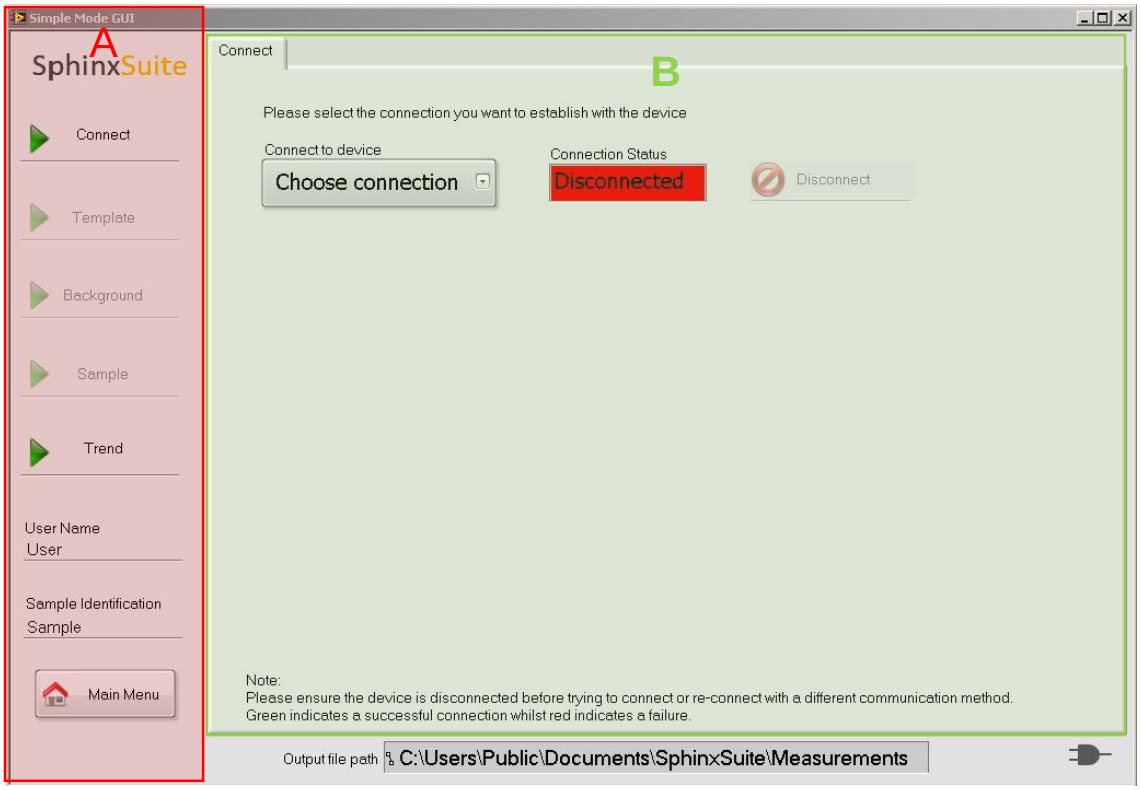


Figure 58: Simple Mode

If the auto connection has been activated the software tries to connect to your device which you selected as standard connection. A window will appear (Figure 12) which is similar to the auto reconnect option. You will see the Serial number and IP address and connection type of the device the software tries to connect and also a counter for the connection attempts. When you click ***Quit to reconnect*** you will come back to the main window.

Click on *Choose connection* then select the method of connection required. The connection status indicator will show you if the connection is established or not. When you click ***Disconnect*** your device will be disconnected and the connection status will indicate the status disconnected.

After this click on ***Template*** and select one of the templates previously saved within the Additional Information tab in the standard interface. (Section 8.1) After selecting a template to be used, all of the field values will be imported. These values can be edited by left clicking on

the relevant cell or by right clicking and selecting from a dropdown box. After the parameter information has been edited, click on ***Save entries***.

To take a background place the background sample onto the sample window and press ***Background***. A progress bar will fill up as the reading is obtained.

Once this has completed, place the sample you wish to measure onto the sample window, ensuring it is completely covered and press ***Sample***. Again, a progress bar will appear on the screen whilst the measurement is taking place.

When the analysis is complete a droplet will appear on the screen: green (good), yellow (warning) or red (bad). These drops provide an image corresponding to the boundaries set when creating the model being used.

Following the test result, pressing ***Trend*** opens the **Trend analysis tool**. This allows you to analyze the results from sample to sample. (Figure 59)

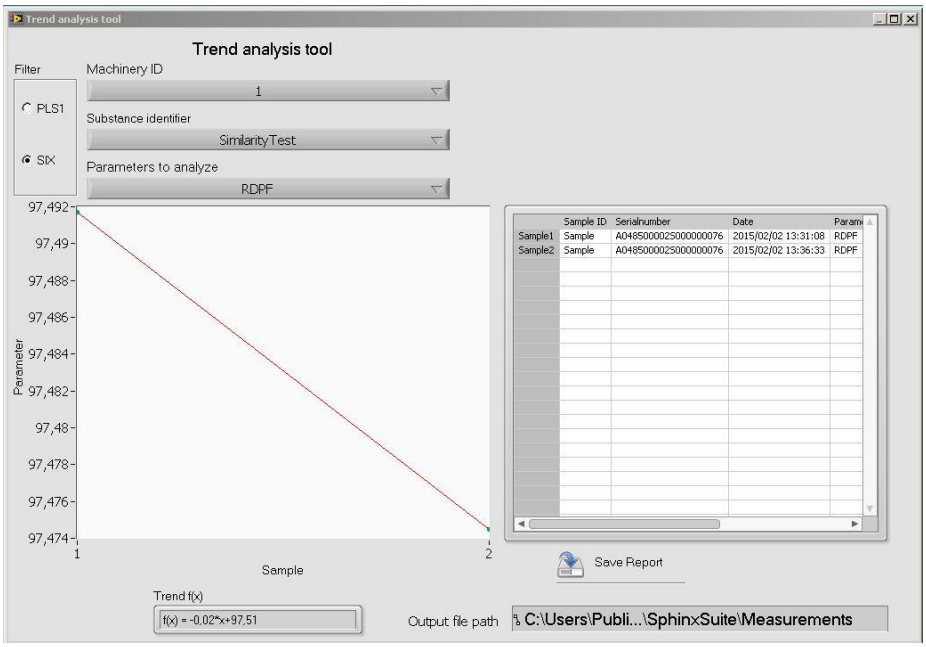


Figure 59: Trend analysis tool

You will have to filter the results as to whether your template contained a PLS1 () or SIX (Similarity) Model. After this, select from the dropdown menu the applicable *Machinery ID, Substance Identifier, Parameters to analyze.* A graph will be placed on the left window, with the result marked and a trend line and function displayed.

There is an option to save this report in a PDF file into your output folder by pressing ***Save Report***.

The **Trend analysis tool** can also be accessed in the Standard Interface by clicking **Prediction > Simple Mode > Simple Trend Analysis**.

To switch back to the main window you have to click on the ***Main Menu*** button and enter the password. If it's correct you will be returned to the main window.

9 STAND-ALONE TOOLKIT

9.1 OVERVIEW

The Stand-Alone Toolkit allows you to configure your device to run independently of a host computer. Automated test sequences, created with the **AUTOMATION TOOLKIT** can be transferred to the device and will be executed after starting stand-alone operation. If you own the **PREDICTION TOOLKIT**, you can also exchange prediction models between a device and a computer and automatically analyze your measurement data on board on IRSphinx. Measurement data will be provided via serial connection. At the present time you can use this toolkit to carry out Timed Measurements, however in the future it will support customized test sequences created with Automation Toolkit and will provide on board prediction. The Stand-Alone Toolkit also allows you to activate or deactivate OCOM. To open the **STAND-ALONE TOOLKIT**, navigate to ***Standalone > Configure Mode…*** to open a configuration dialog. You can click on ***Mode Information*** to find out more about the modes available.

Note
To view data streaming from the IRSphinx device (RS232 and RS485) you will need an IRSphinx Extension Board (only available for industrial Transmission Devices).

9.2 CONFIGURE STAND-ALONE MODE

At first you need to be connected to a device before you can start the configuration. The Configure Stand-Alone window has a dialogue on the upper left of the screen that allows you to select the stand-alone mode. (Figure 60)

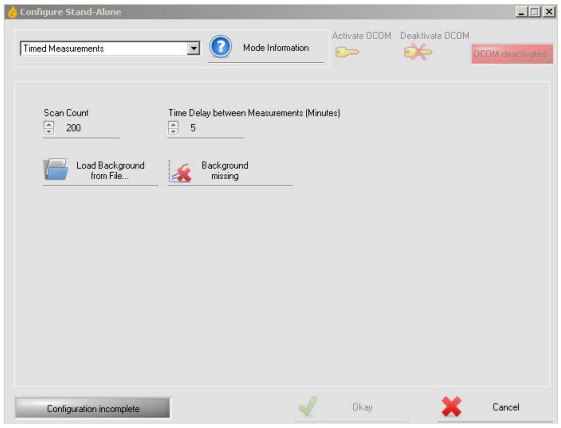


Figure 60: Configure Stand-Alone Mode

In "*Timed Measurements*" mode, your spectrometer will record spectra automatically. You can choose the time interval between measurements and set the scan count per measurement.

To record a transmission spectrum, a background measurement must be loaded by SphinxSuite and transferred to the device. It will be saved on board in order to compute the transmittance spectrum. The transmittance spectrum will be provided by the *IRSphinx Extension Board*. For more information about how data is published refer to the documentation of the *IRSphinx Extension Board.*

If the configuration settings are valid, the indicator switches to "*Configuration complete*" and the ***Okay*** button is enabled. At this point the chosen stand-alone mode will be shown on the main window and the indicator switches from ***Not configured*** to ***Correctly configured.*** You will need to transfer the configuration to your device by clicking on ***Update Device.*** Once you have done this an indicator (✓/✗) will show you if the transfer has been successful.

Figure 61: Stand Alone Options

9.3 START AND STOP STAND-ALONE MODE

Once you have updated your device, you can define the name of the measurement run and the tester under "*Sample Identification*" and "*User Name*". Clicking ***Start*** activates the stand-alone mode and will cause the indicator to switch to ***Stand-Alone active***. You will also be able to view the configuration, starting date, sample identification and tester's name by selecting ***Stand-Alone Details*** when stand-alone mode is active. (Figure 61)

The stand-alone mode can be stopped by clicking on ***Stop.***

If you connect to a device in stand-alone mode, you will automatically be navigated to the *Stand-Alone* tab and can view the stand-alone details or stop the mode.

9.4 OCOM STATUS

If you are connected to a device, you will get the status of your device when you open the configure stand-alone dialog window. With the buttons ***Activate OCOM*** and ***Deactivate OCOM*** you can change the OCOM status of your device. (Figure 60) How OCOM is working you can find on the attachment OCOM Documentation.

10 MODULATION FREQUENCY OPTION

10.1 OVERVIEW

This Option allows you to change modulation frequency of infrared sources of IRSphinx. The default value is 8 Hz. The modulation frequency can only be varied within a device specific range. To find whether or not your device is within the correct range you first need to be connect to the device. If you enter a value outside the valid range, it will automatically be corrected to a valid value. If you are not connected to the device you will not be able to make any changes and all settings will be greyed out.

10.2 CONFIGURATION

The modulation frequency can be set for: "*Live-View"*, "*Background"* and "S*ample"* measurements in the **BASE VERSION** of SphinxSuite, *"Measure Background"* and "*Measure Sample"* under **options** in the test sequence editor of the **AUTOMATION TOOLKIT** and in the *Trigger Mode Configuration* dialog.

Please note:

As we continue to improve SphinxSuite we will upgrade the software and manual. Your comments and suggestions on how to improve the software and manual will be welcomed. For software updates or manual updates, please send an email stating your customer ID to software@irsphinx.com

11 ATTACHMENTS

- OCOM Documentation
- Hardware Manual
- Status LEDs
- Troubleshooting SphinxSuite
- Possible Extension Board Solutions