
GETTING STARTED

BioLight Studio Software FL

Version 1.04.00

Table of Contents

Installation.....	2
Requirements.....	2
Installation Procedure.....	2
Installed Programs.....	2
Setting up a new System.....	3
Entering the License Key.....	3
Changing the Comport.....	4
Manual Control.....	6
Running a Simple Scan-Method.....	7
Loading the Example Scan Method.....	7
Running the Scan.....	9
Saving the Results.....	11
Auto-Save Results.....	11
Experiment Data.....	12
Defining a Complex Scan-Method.....	13
This Scan method contains the following features:.....	13
Creating a new Method.....	13
Defining Sample Information.....	14
Defining the Scan Parameters.....	15
Setting up the Calculations.....	16
Defining the Report Template.....	19
Automatically Exporting Spectra.....	24
Defining a Concentration Method.....	25
This Read method contains the following features:.....	25
Creating a new Method.....	25
Setting up the Well Plate Reader.....	26
Defining the Read Parameters.....	27
The Concentration Calculator.....	28
Defining the Report Template.....	29
Defining a 3D Scan-Method.....	35
This 3D Scan method contains the following features:.....	35
Defining the 3D Scan Parameters.....	35
The 3D Viewer.....	36

Installation

Requirements

The following items are required for operation of BL Studio:

- PerkinElmer LS 45/50/55 Spectrometer with F1 Firmware
- PC Pentium 500 MHz with a free RS232 serial com port or alternatively an USB-Serial converter
- At least 64 MB Ram
- Operating System: Windows 8, Windows 7, Windows Vista, Windows XP

Installation Procedure

Download the install set from the homepage www.biolight.com/Downloads

Start the installation by double clicking on SetupBLStudio.exe. Please note that you need Windows administration rights to run the installation.

If BLStudio has already been installed you will be asked if you want to uninstall the existing version. To update the version firstly uninstall it, then re-start SetupBLStudio.exe.

Now follow the instructions. During the installation you can change the program and data directory. **Please ensure that all analysts will have read/write access to the data directory.**

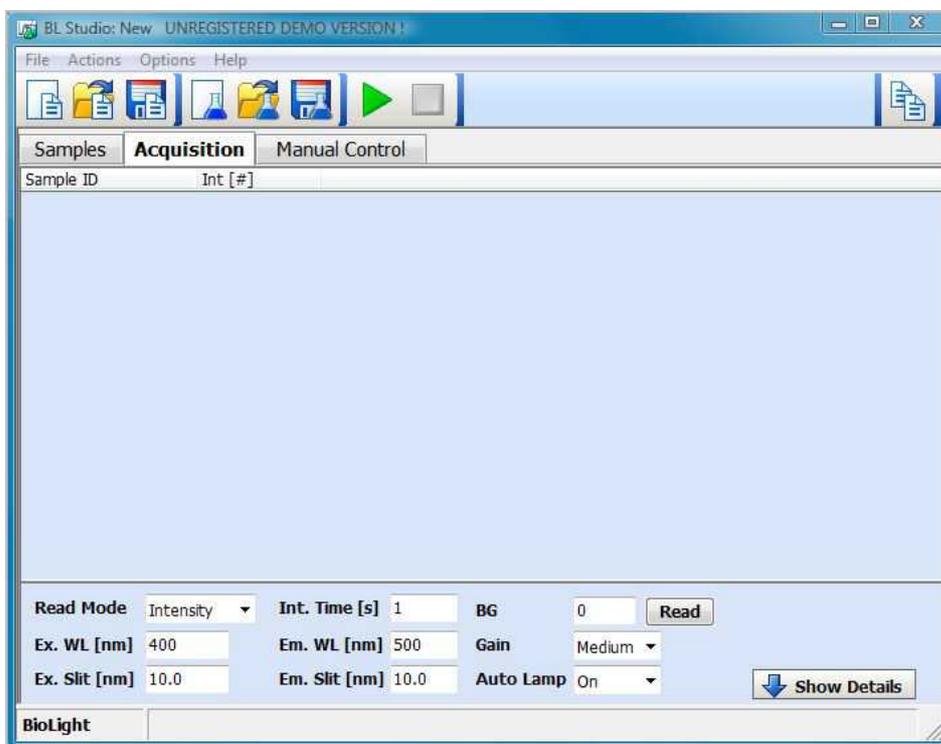
Installed Programs

The BL Studio software package consists of 5 programs:

BLDevelopment:	Flexible tool for measurements and method development
BLValidation:	Instrument performance validation tool
BLAdministration:	Administration tool
BLRoutine:	Tool for routine measurements
BLCalculator:	Free form calculator.

Setting up a new System

After the installation has finished you can start all BLStudio programs either from the icon on the bench-top or via the start menu. Start BLDevelopment:



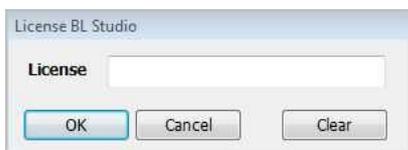
Entering the License Key

Please note that BLStudio is installed in Demo mode by default (Data and Methods cannot be saved). To enable full access to BLDevelopment you need to enter your license key now:

From the menu select Help/About. Click on the first line in the license table, then on the Enter License button:



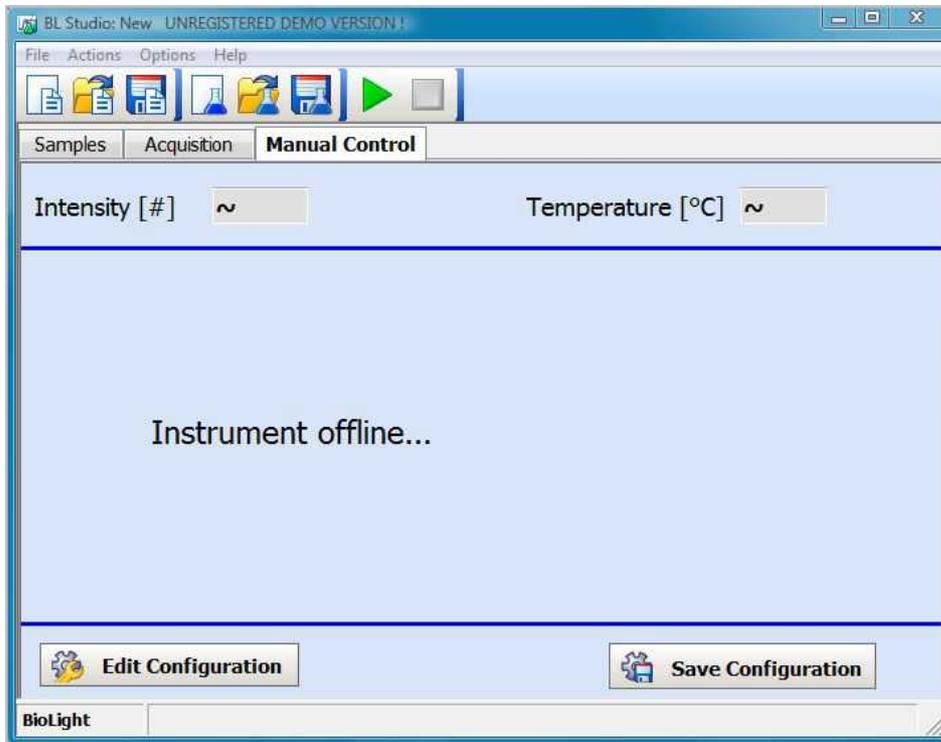
Now enter or paste your license key:



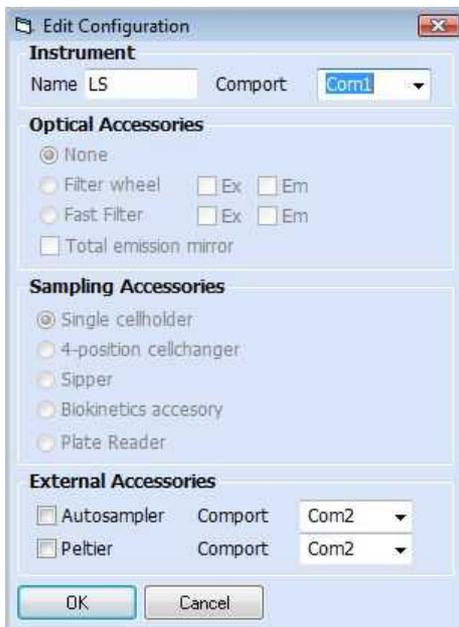
and press the “Ok” button. If the License key is invalid the dialogue will stay open and an error message is displayed.

Changing the Comport

By default BL Studio uses Com1 as interface for the Spectrometer. External accessories are disabled. To change the configuration select the Manual Control tab.



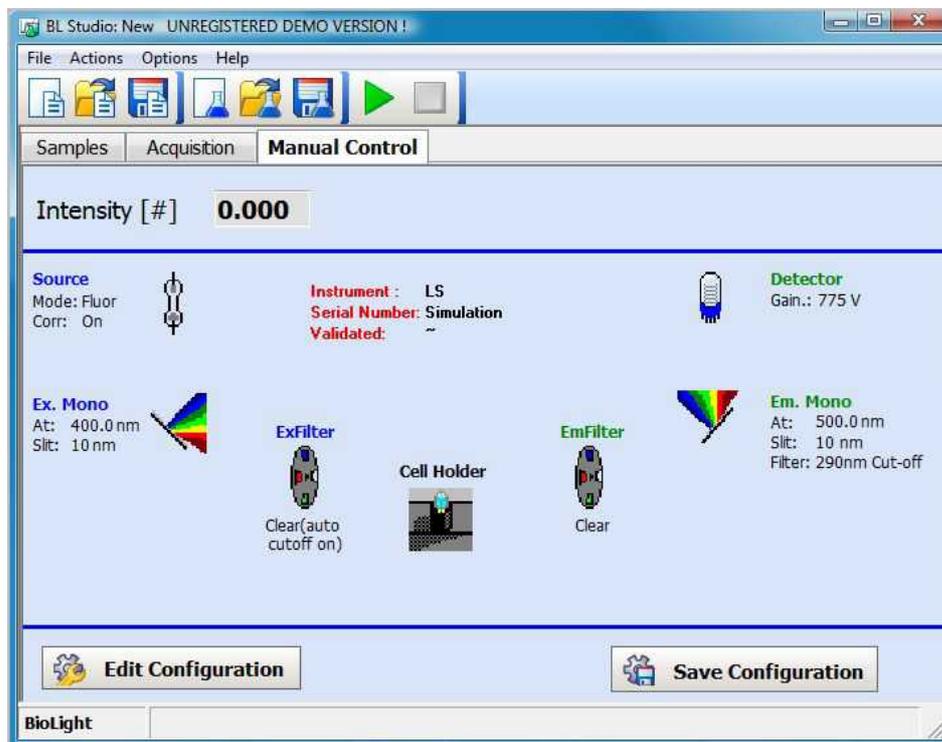
Now click on the “Edit Configuration” button and select the comport in the configuration dialogue:



Please note that if the comport address is higher than com3 it cannot be selected from the list but must be entered manually.

At this point you can also define a user friendly name for the instrument. This name will be used in reports and the validation. Furthermore you can select a S10 auto-sampler or P1 peltier module as external accessory: Select the corresponding option and define the desired comport.

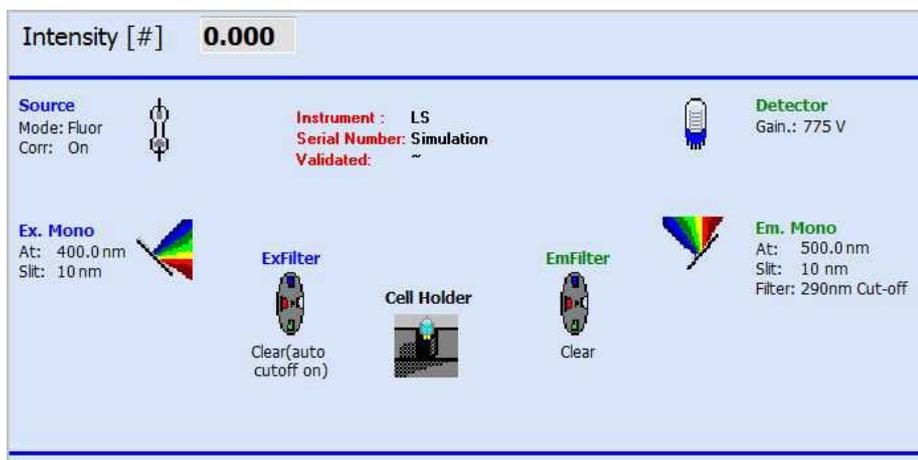
All other accessories can be selected in simulation mode only, otherwise they are determined by the configuration of the currently connected instrument.
After pressing Ok the software re-connects to the instrument.



To save the new configuration the “Save Configuration” button.

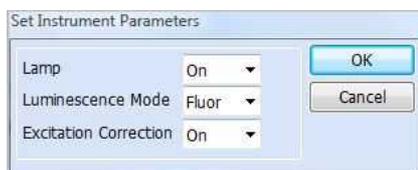
Manual Control

The manual tab displays a schematic of the LS-45/50B/55 optical system with icons representing the individual components. The current status of the instrument is described by the text next to the icons. Furthermore the instrument type, the firmware revision and the serial number of the instrument are displayed.

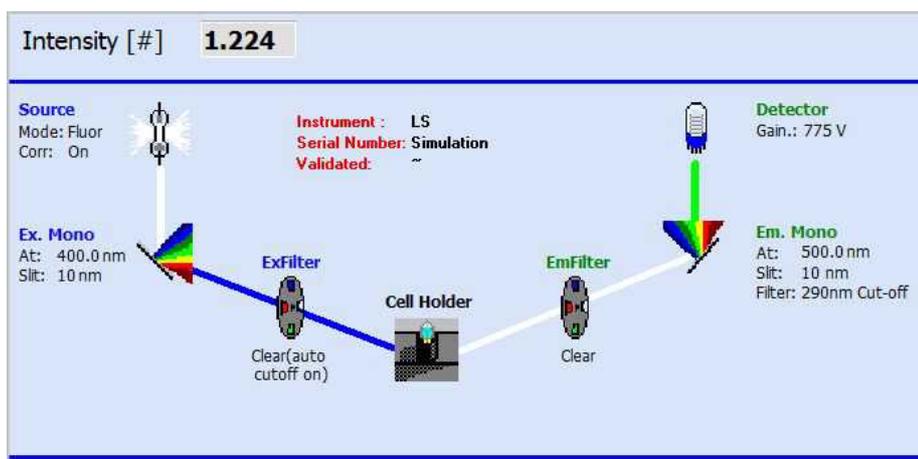


To enhance the lifetime of the lamp and to reduce the noise level of the LS, BLStudio switches the lamp off automatically, when it is not needed.

Left clicking on the Source icon starts the source dialogue:



Selecting lamp on and pressing ok turns the lamp on



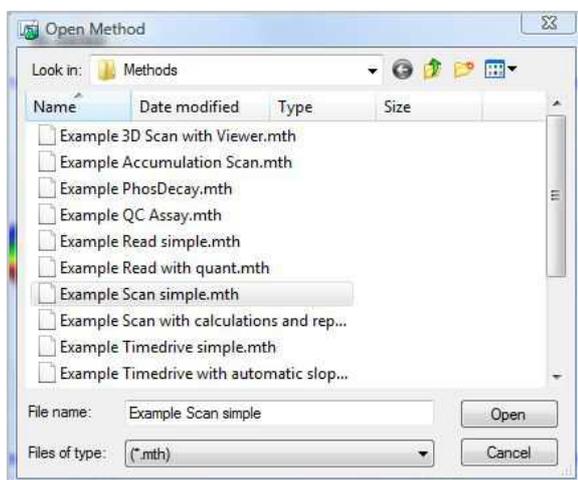
The colours of the beams between each icon in the optical schematic are an approximate indication of the wavelength. On top of the LS-Module the current intensity is displayed.

In this way it is possible to change all instrument parameters. Left click on the icon relating to that part of the optical system and enter the desired value for the parameter.

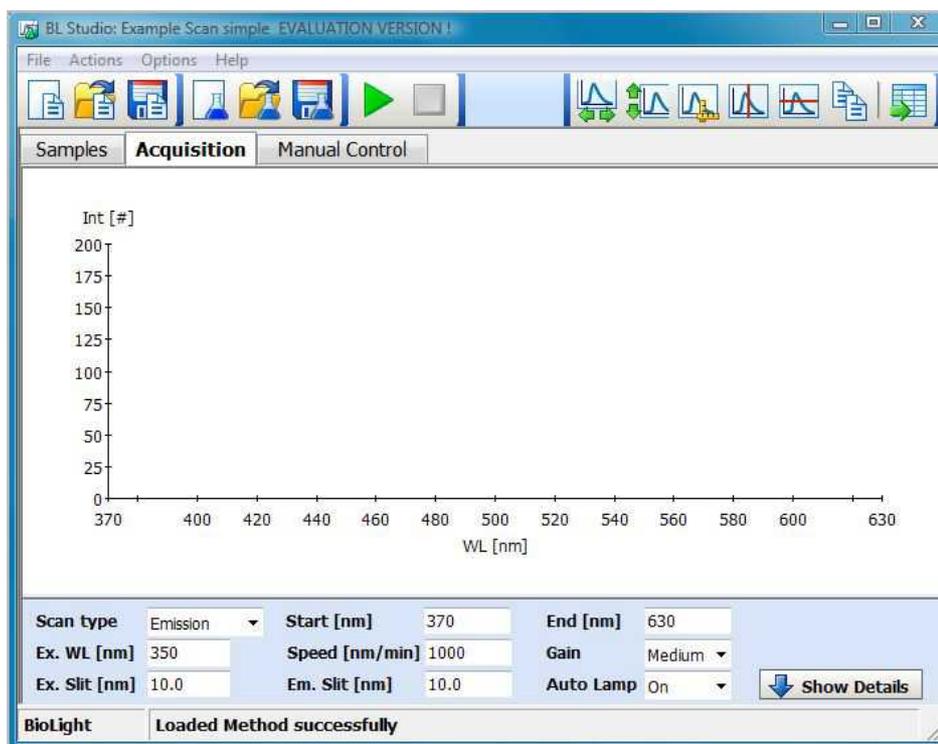
Running a Simple Scan-Method

Loading the Example Scan Method

In BLDevelopment select this function from the file menu or the toolbar. A standard windows file open dialogue opens. Select Example Scan Simple and press Open.

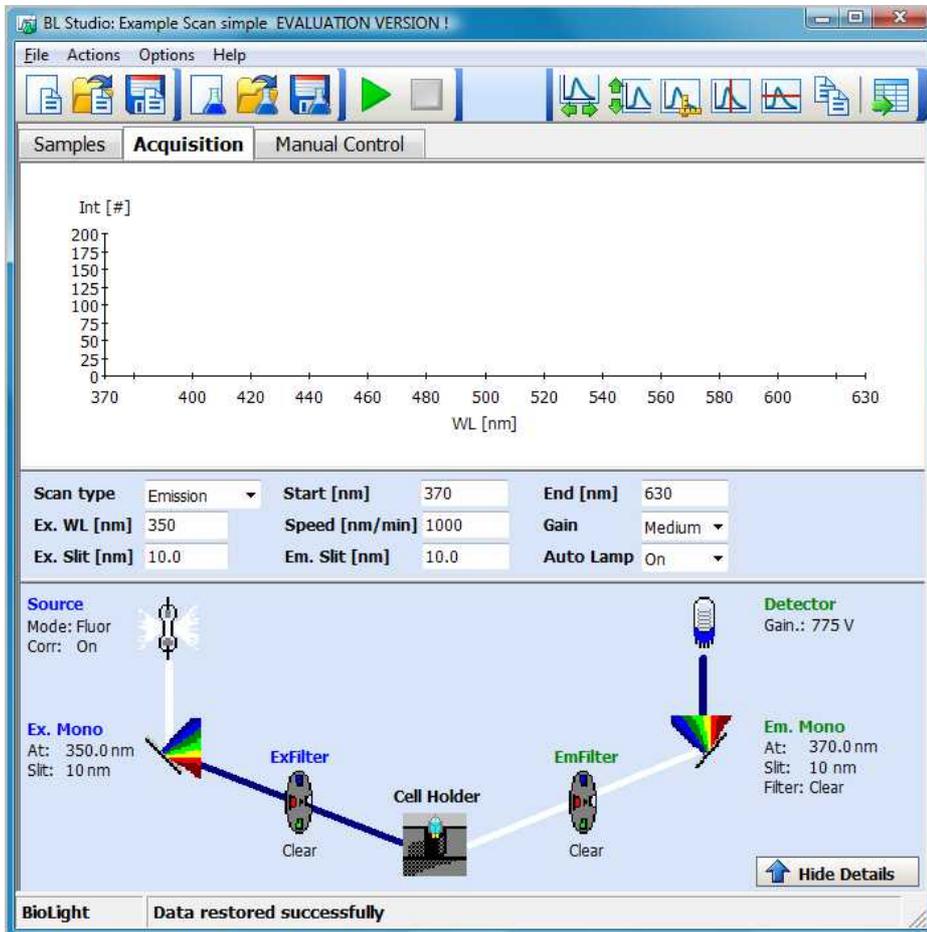


The method is loaded and the method parameters are displayed:



Now the scan parameters can be entered. Using the Autolamp option has three effects: it preserves the lifetime of the lamp, protects sensitive samples from photobleaching and generates less audible noise. The lamp will be switched on only when needed. Since the lamp is pulsed, and is switched on and off 50 times per second, the use of the auto lamp off function has not been seen to cause problems compared to other types of lamps which have warm-up times. However, in order to maintain compatibility with older FLWinLab methods, the Auto Lamp option can be turned off.

To complete the method set-up all instrument parameters can be viewed and modified after pressing the “Show Details” button. A schematic of the instrument is shown, displaying the instrument set-up for the current method:



Please note, that this schematic does NOT show the current status of the instrument. To view the current status or modify it the manual control must be used.

Finally save the method under a new name via the Save Method Dialogue from the toolbar or the file menu. Now the data acquisition can be started:

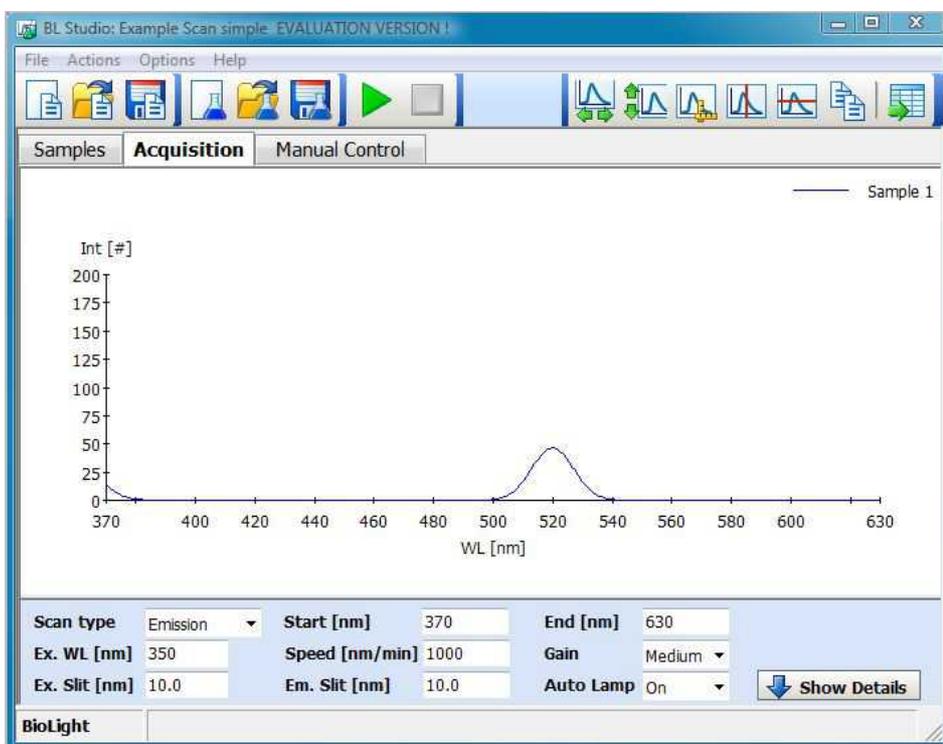
Running the Scan

Press the run button . The instrument is setup with the starting conditions. Then the user is prompted to enter the first sample:

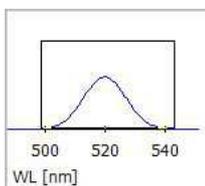


Insert the sample, then press the start button: the data collection for the first sample is started. The collected data is displayed online in the graph.

When the measurement is complete, the user will be prompted again for a new sample. Click on stop to end the measurement.



It is possible to optimize the graphical display e.g. by zooming into the data: Left click with the mouse on the upper left point and move the mouse to the lower right point, keeping the left mouse button pressed. A black rectangle appears:



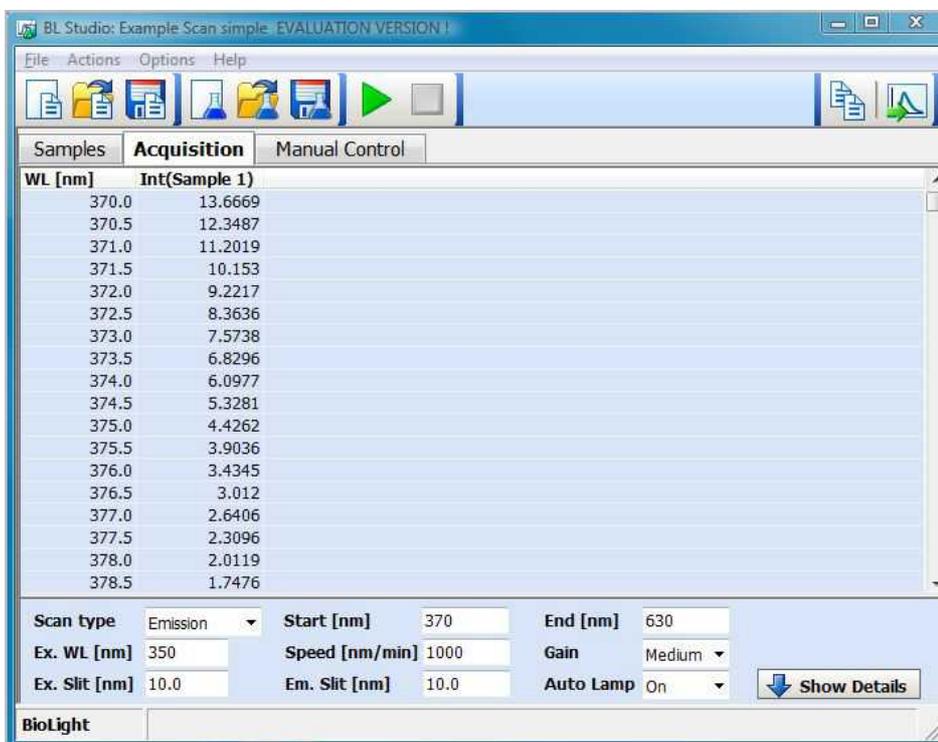
Releasing the mouse button will then actually perform the zoom. To undo a zoom double click into the graph. It is possible to zoom into a graph in several steps. The undo zoom will then undo the zooms step by step

Alternatively the graph toolbar can be used to format the display:

-  expands the x-axis to show the entire abscissa range of all selected data file
-  expands the Y-axis to show the entire ordinate range of all selected data files
-  starts a dialogue to format abscissa and ordinate ranges

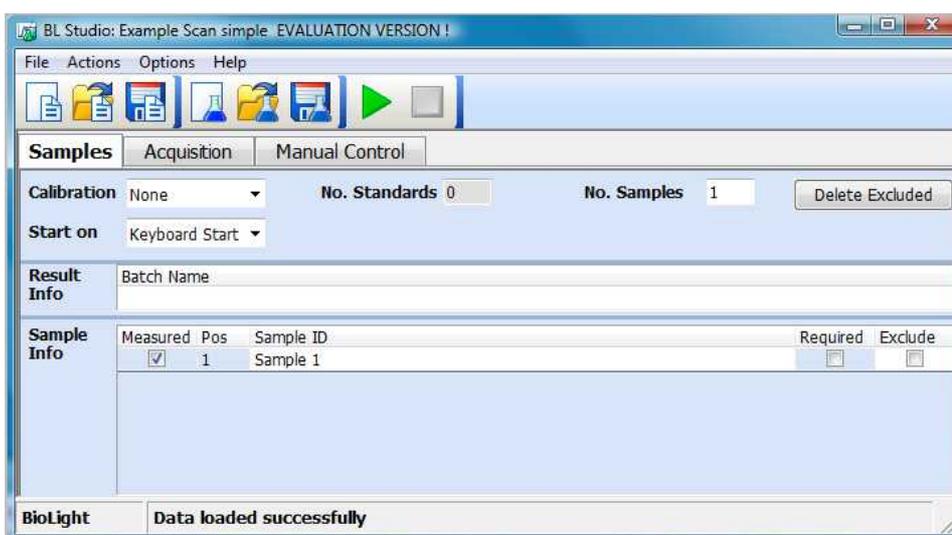
To evaluate the data it is possible to use the following functions from the graph toolbar:

-  activates the vertical cursor. The current position of the cursor is displayed in the title of the x-axis, corresponding y-values of data are displayed in the legend.
-  activates the horizontal cursor. The current position of the cursor is displayed in the title of the Y-axis
-  copies the picture of the graph to the clipboard, from where it can easily be pasted into Word or Excel.
-  alternates the data display as a table or a graph (tabular form shown below)



 copies the tabular data to the clipboard, from where it can easily be pasted into Word or Excel. Alternatively you can right-click on the table and save the data to a file in an Excel compatible ASCII format.

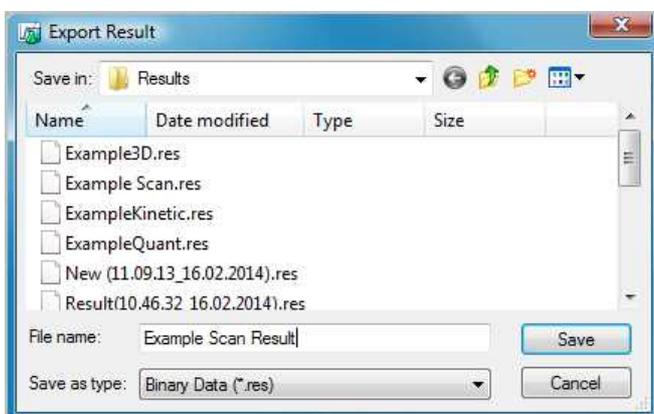
Now select the sample tab. As you can see the sample has been added automatically:



It is possible to rename the sample, exclude the corresponding data from evaluation or even delete the data completely. If desired it is also possible to define a sample list before starting the acquisition.

Saving the Results

Finally you can save the result to a file. Click on the Save Result icon . A standard windows file save dialogue opens

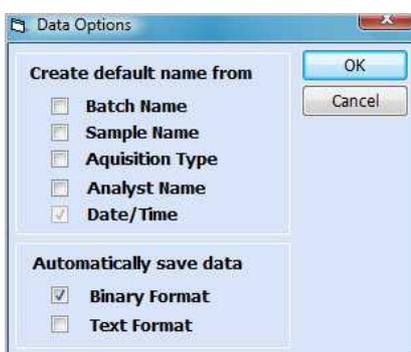


By default the standard data directory is selected in the dialogue, but you can save the result on any location on your PC (or network).

Please note that it is possible to save the results in ASCII, or the PerkinElmer .sp format. However, only the .res format contains all necessary information to re-load the result into the BLDevelopment program.

Auto-Save Results

To ensure no data is lost if e.g. the power fails it is possible to automatically save all results to hard disk. Select the Data Options Dialogue from the Options menu:



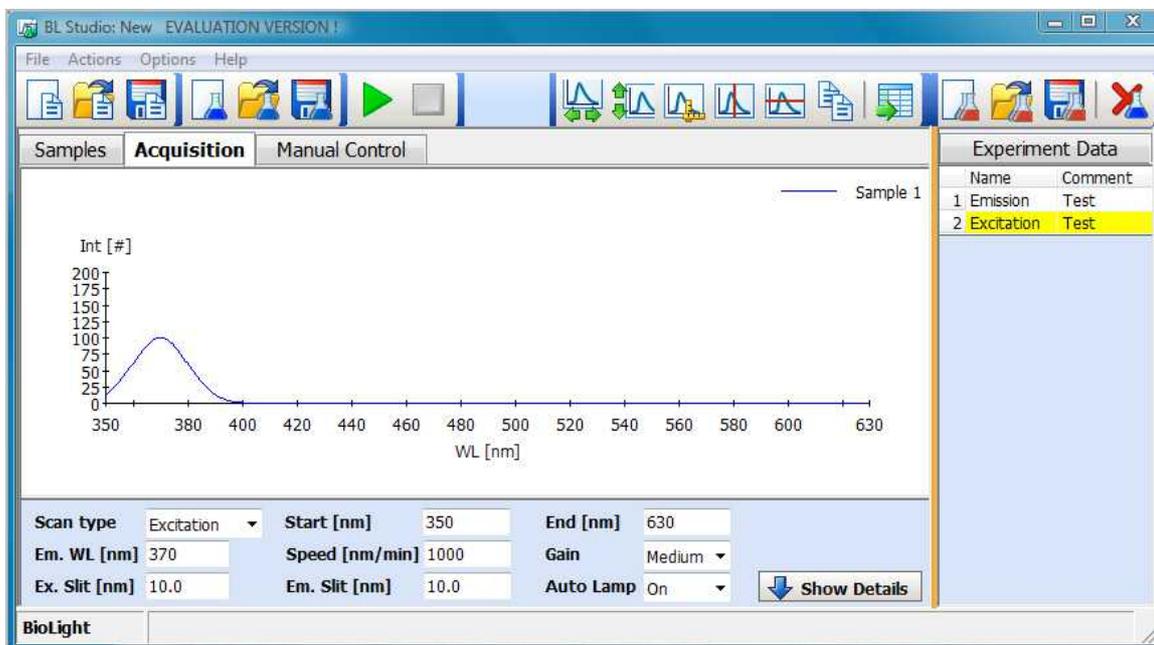
If the binary and/or text format is selected a temporary data record is saved whenever a sample has been measured (every minute in time-drives). When the measurement is completed the result is saved in the default data directory. Using this dialogue it is also possible to define the name of the result file (date/time cannot be deselected, this ensures that the file name is unique).

In binary format a .res file is generated, which can be re-loaded into BLDevelopment, the text format creates a tab delimited ASCII file.

Experiment Data

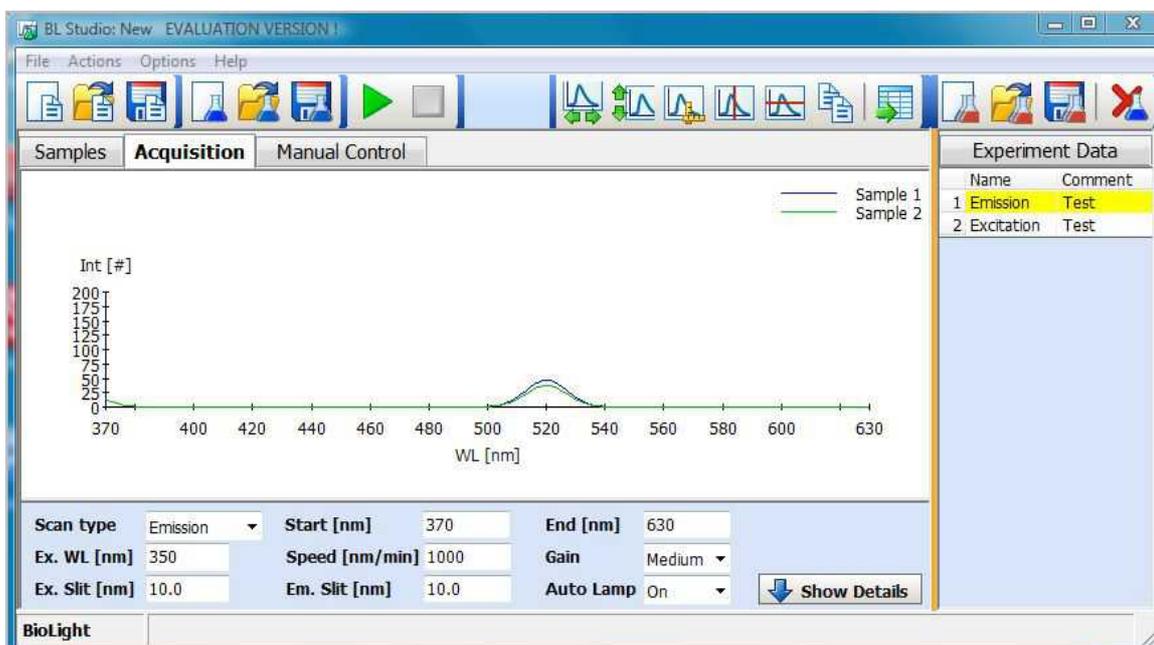
The Experiment Data feature is comparable with FL/UV WinLab's data region: Whenever data is collected or loaded a result record is automatically generated in memory and displayed in the Experiment Data table. To display/hide the Experiment Data Table open the Display Options Dialog from the Options menu and select the Display Experiment Data Window option.

Change the scan type to excitation and re-start scan: A new record is added to the Experiment Data:



Selecting a record by left clicking on it automatically loads the corresponding method and data. Thus it is easy to view all results, generated during one session.

Since the complete method is restored from the result it is also possible to append new data to a result: Select a result from the Experiment Data and press start. The new data is appended to the existing result.



It is possible to modify the name and comment of each result by editing the table. To remove a result, select it and click on the Remove Result  button. Finally it is possible to save/load a complete experiment via the Load Experiment  and Save Experiment  button.

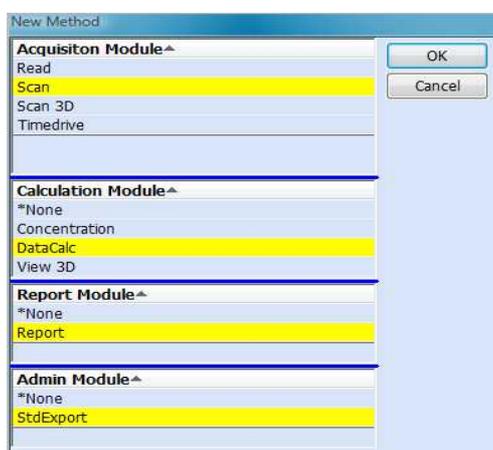
Defining a Complex Scan-Method

This Scan method contains the following features:

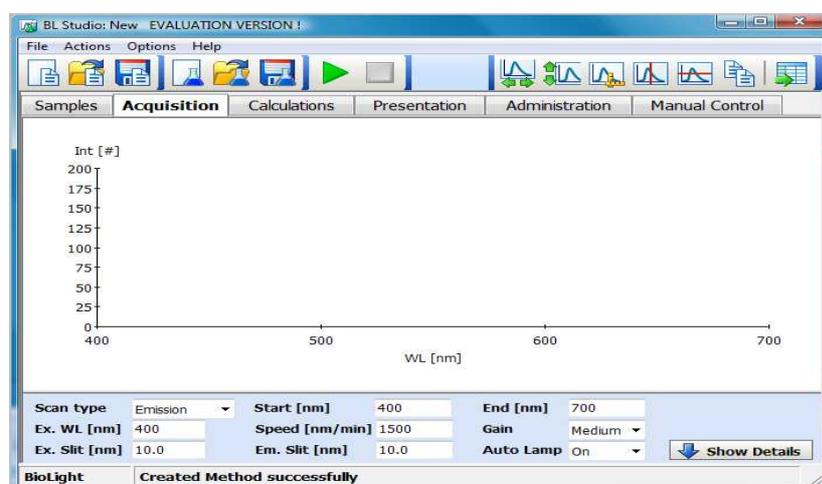
- A data entry column is added to the sample table, allowing the user to enter weight values for each sample
- A background sample is pre-defined
- The background sample spectrum is subtracted from each analytical sample spectrum
- The peak height is calculated for each sample
- The peak height for each individual sample is divided by the sample's weight value
- A report is generated which shows the sample spectra and the calculated results
- Spectral raw data for each sample is automatically exported in ASCII format

Creating a new Method

To create a completely new method press the New Method  button of the toolbox of BLDevelopment. From the New Method Dialogue select the desired modules for each step of the method:



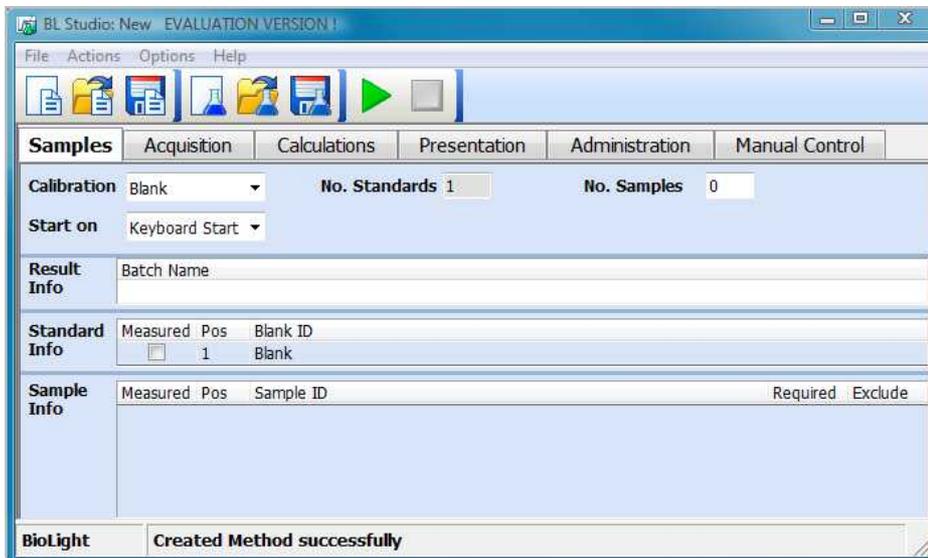
In this case Scan, DataCalc, Report and StdExport are selected. After pressing the OK button the selected modules are added to the BLDevelopment program:



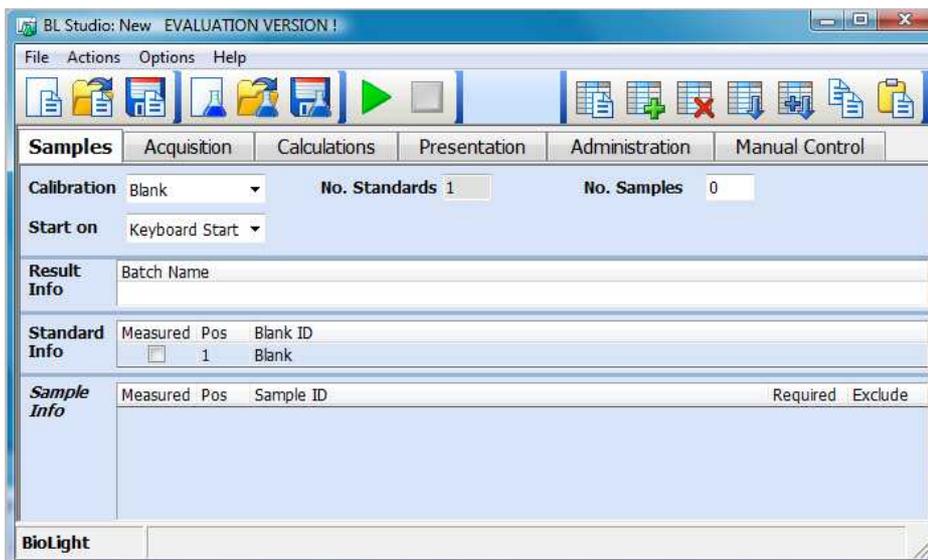
The work flow consists of a logical series of steps. These steps are represented by tabs on the main page, starting from sample handling through data acquisition, calculations and report presentation to administrative functions such as automatic exporting of results. It is recommended to define the method step by step following the work flow.

Defining Sample Information

First add a blank measurement by setting the Calibration Mode to Blank. A new standard info table is shown with the predefined blank sample:



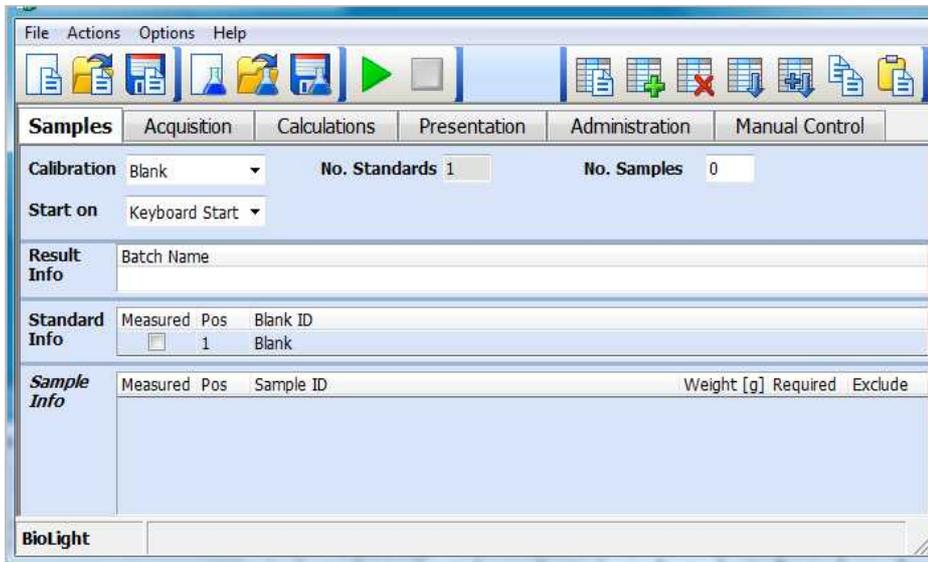
Now left click on the Sample Info table. A table toolbar is shown on the upper left corner.



Click on the add column icon  and enter the information for a new numeric column Weight into the Setup Column Dialogue:

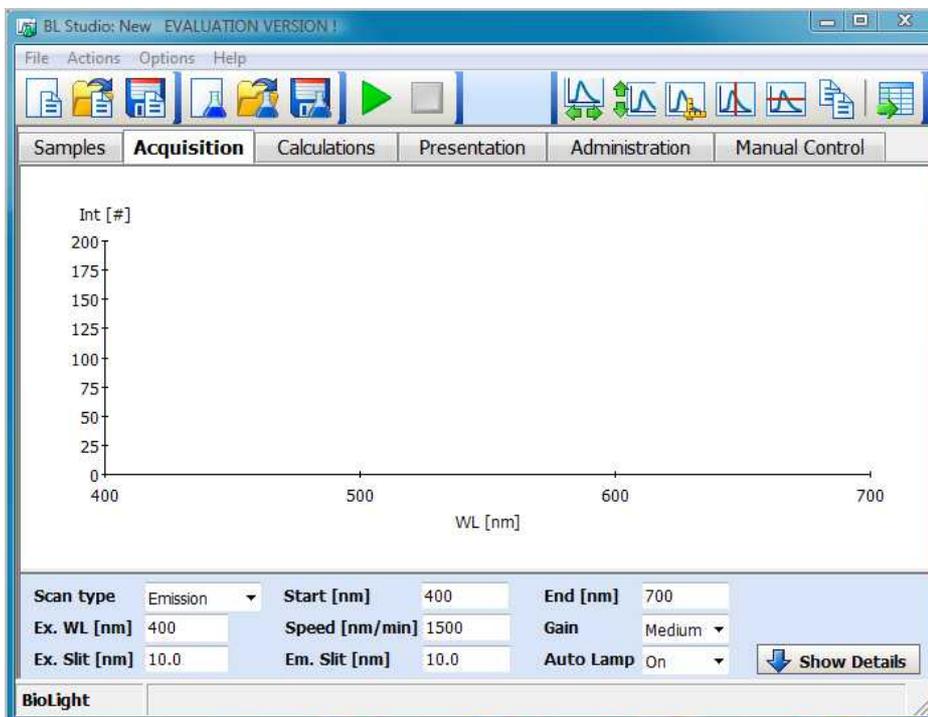


Now press OK to add the column to the sample info table.



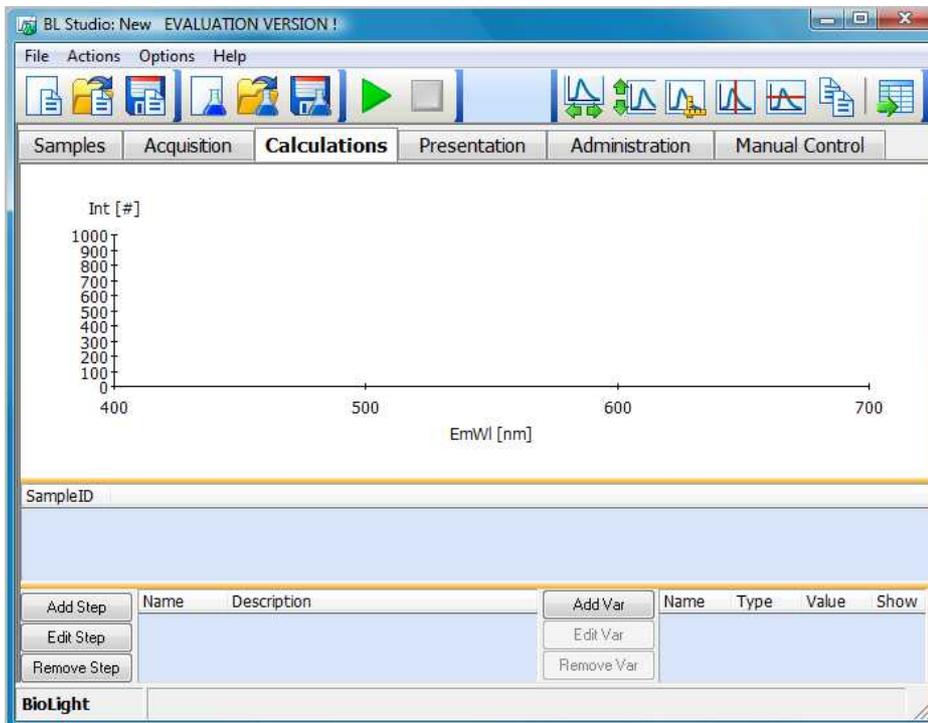
Defining the Scan Parameters

Select the Acquisition tabulator and enter the scan parameters:



Setting up the Calculations

Select the calculation page

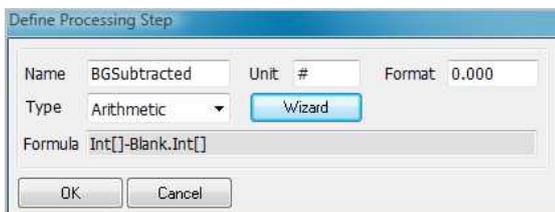


To subtract the blank from all measurements perform the following steps:

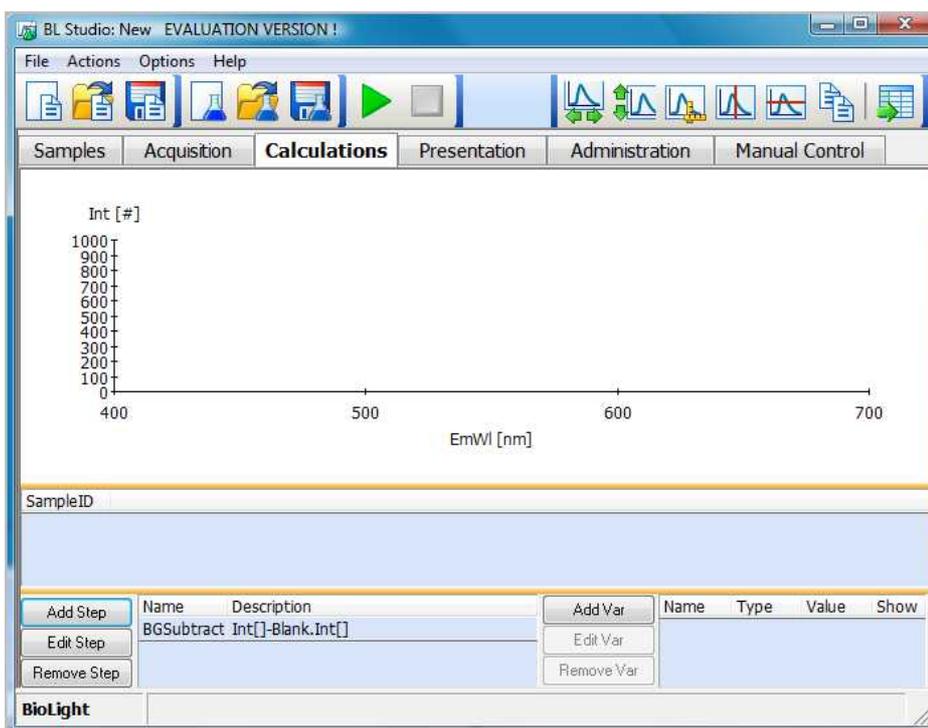
Press the Add Step button, enter the name (please do not use . within names), unit and format for the background subtracted data. Then select Arithmetic as step type.

Now press the Wizard button to start the Formula Editor dialogue and enter the formula $\text{Int}[] - \text{Blank.Int}[]$ by clicking the corresponding buttons.

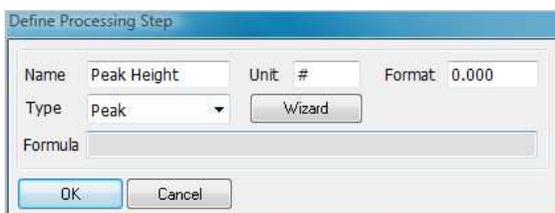
After pressing OK the formula is checked, the dialogue is closed and the formula is added in the “Define Processing Step” dialogue:



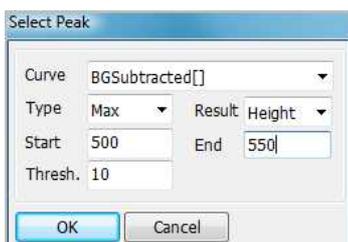
Press OK to add the new Step.



Now define the calculation of the peak height in the area between 500 and 550 nm. Press Add Step

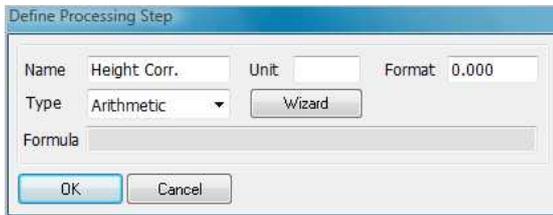


Select type Peak and press the Wizard Button. Enter the parameters in the Peak dialogue

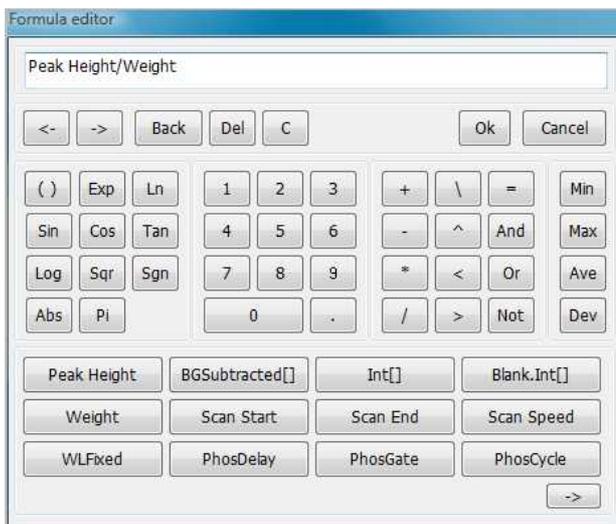


and press OK to add the new Step.

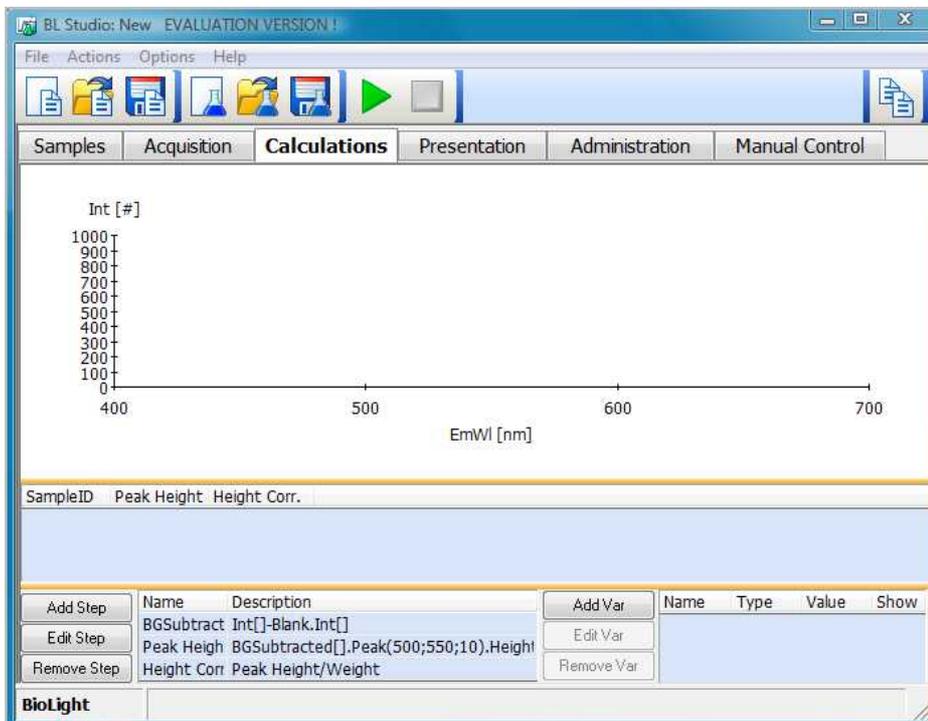
Finally add a corrected height.



via the formula editor



and press OK:

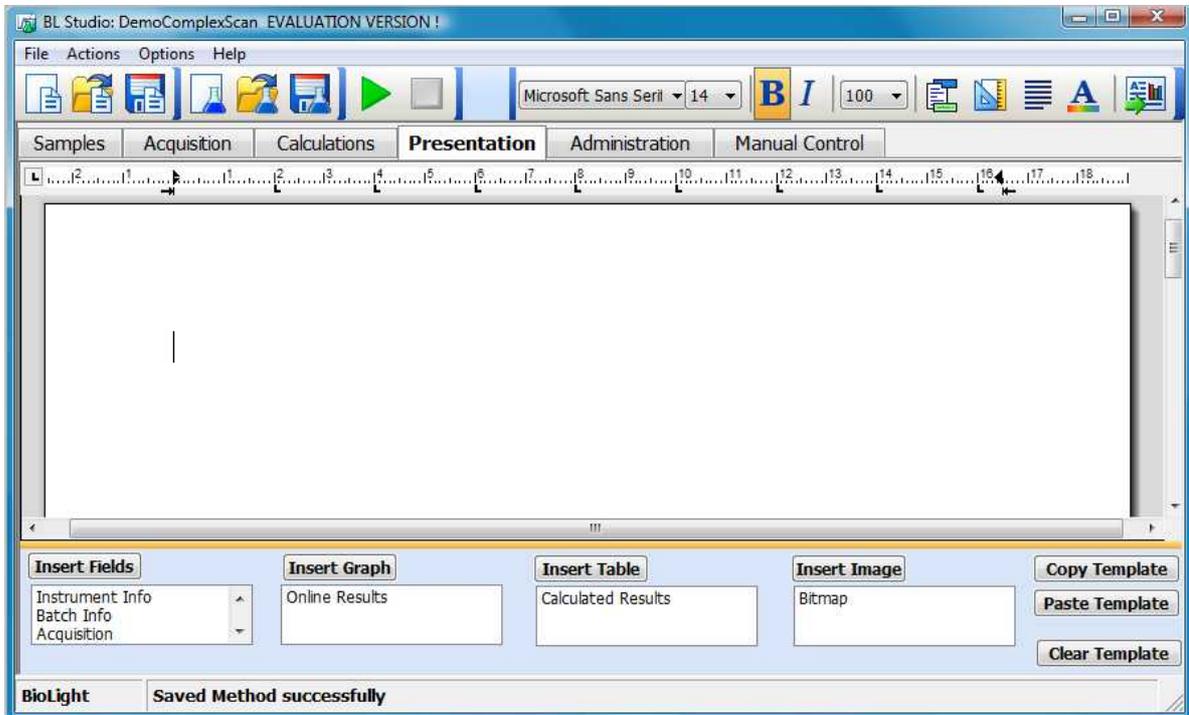


The calculations have been defined successfully.

Defining the Report Template

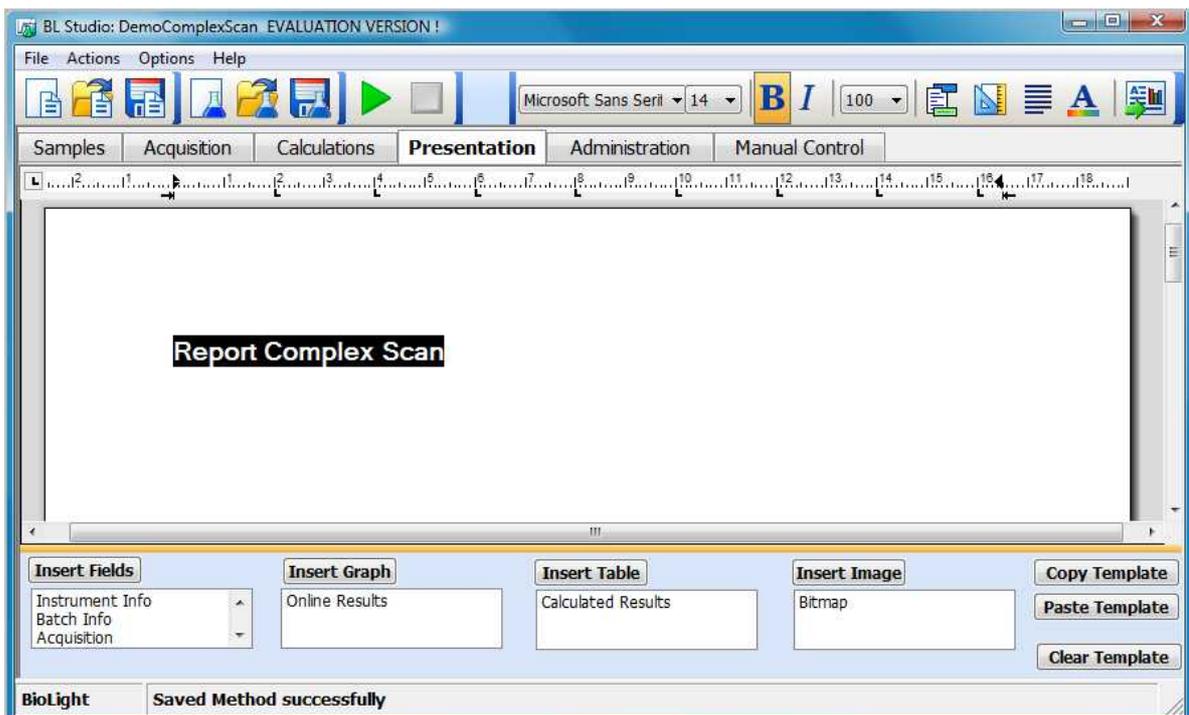
Select the Presentation page. An empty report is shown. A report is generated by filling a report template with result data.

To edit a new template click on the “To Template” icon  at the upper right corner this displays the template edit mode:

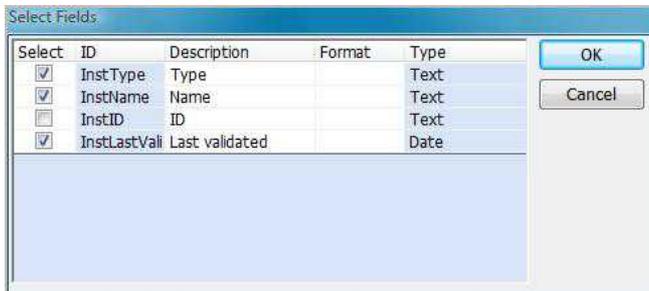


Basically the template can be edited like a MS Word document. It is for example possible to paste an existing Word document into the report (except fields). Please note, that there is no “undo” function yet, therefore it is recommended to save the method frequently.

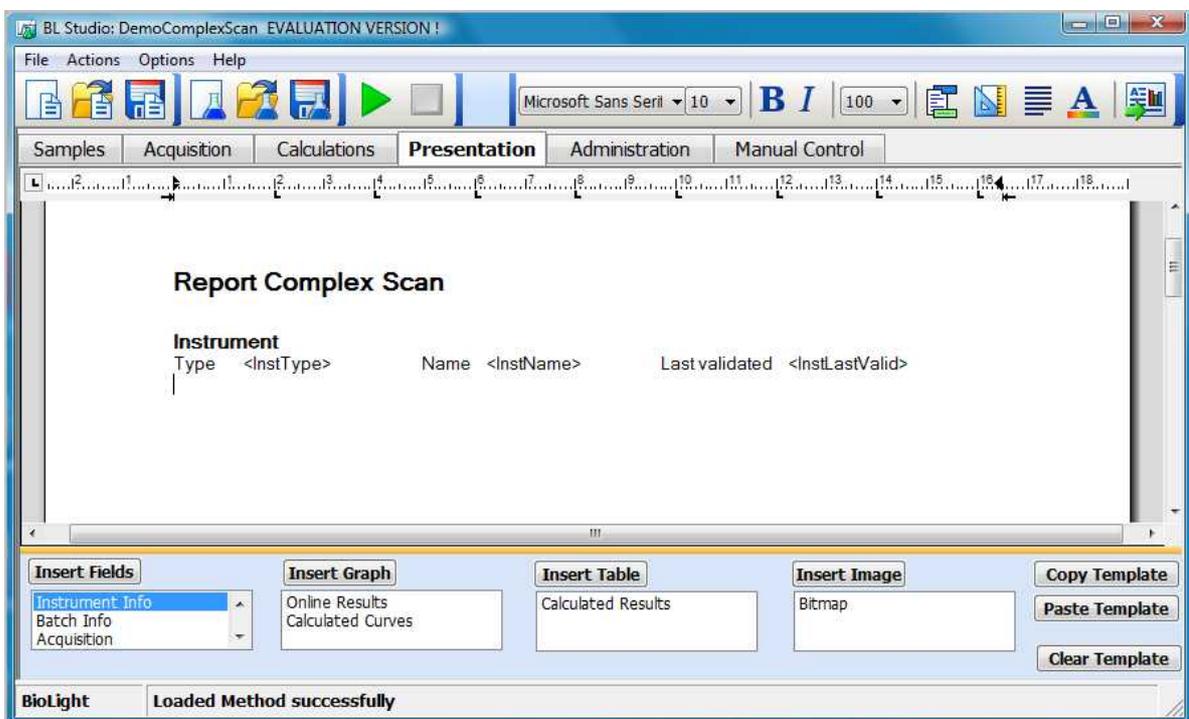
Firstly click into the template and enter a caption for the report. Then select the caption and format it:



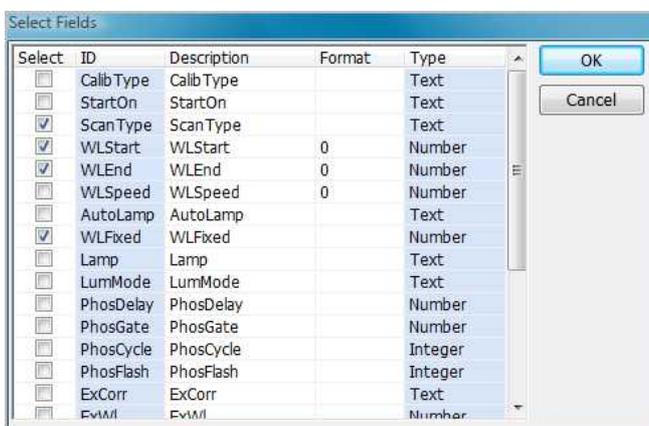
Now enter the first fields. In the template click at the position, where the fields are to be inserted. Then select the Instrument Info entry in the field table and press the Insert Fields button. From the “Select Fields” dialogue select the information you want to add



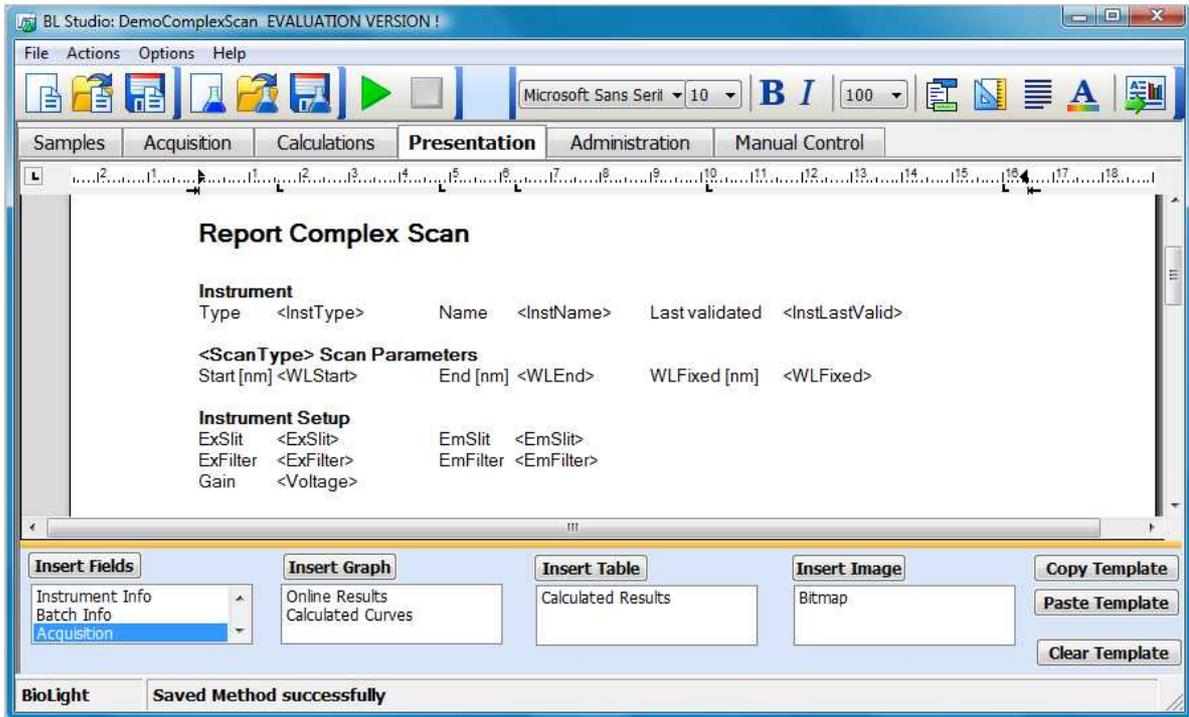
and press OK. The fields are added. All fields are marked with “<>”. Enter a text “Instrument” above the fields, then format the fields. You can use tabs in conjunction with the ruler to define the space between the fields.



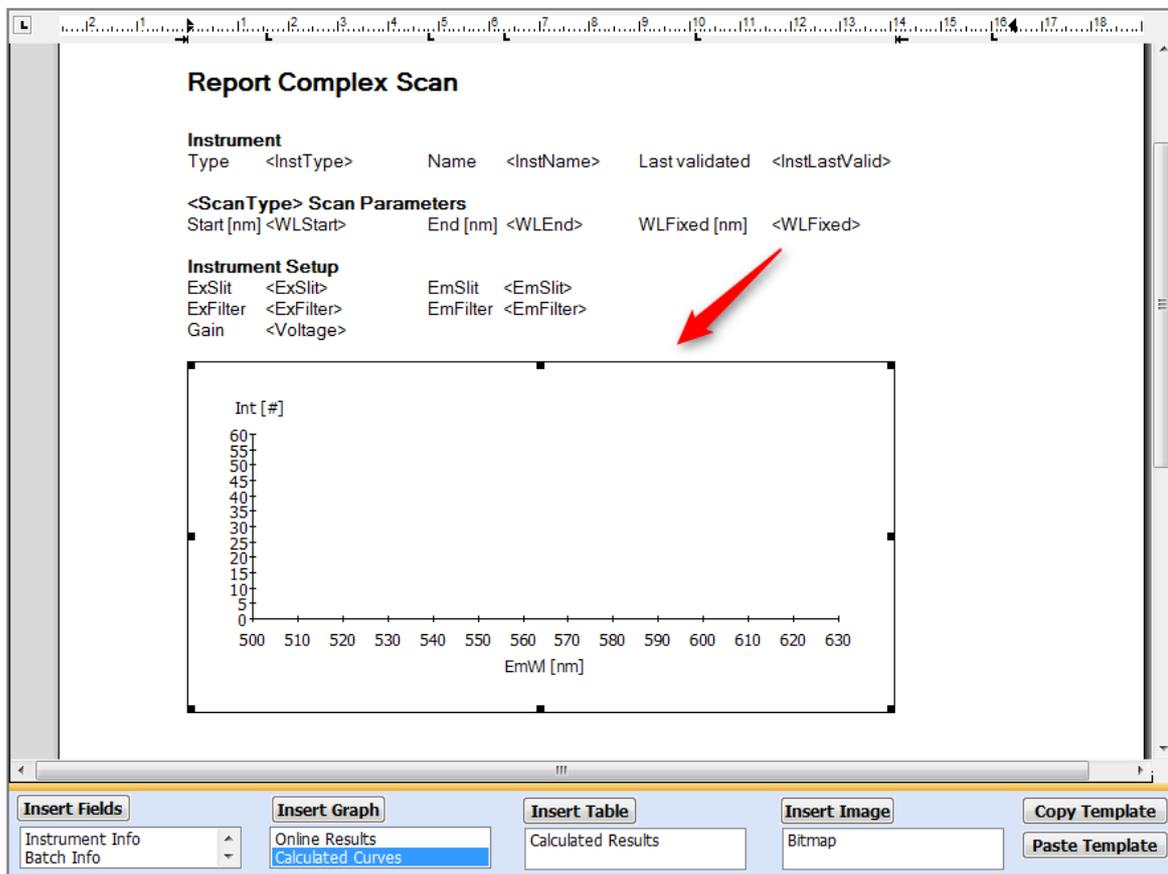
In the same way add the scan parameters from the Acquisition entry:



Format them and add set up parameters (also from the Acquisition entry). To delete a field left click on it and press the delete button on the keyboard (do NOT select the complete field or a complete line)

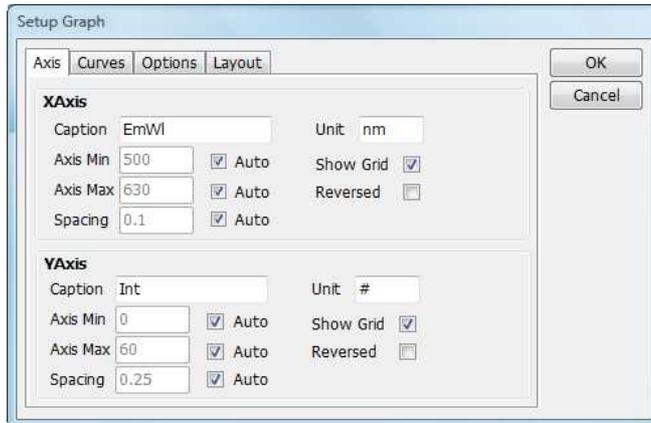


Now add the first graph: click in the template at the position the graph should be inserted, select Calculated Curves from the Graphs table and press the Insert Graph button and left click on the graph to select it. A rectangular frame is drawn around the graph:



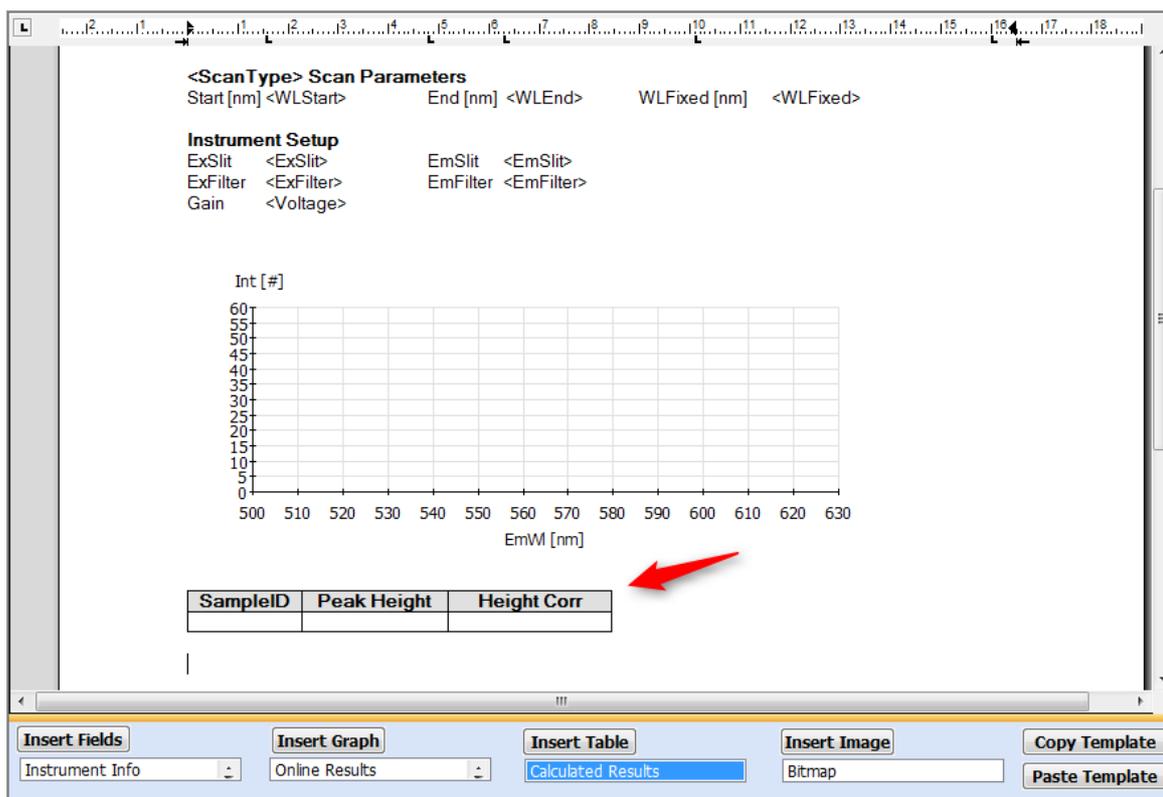
The graph can now be moved and sized using the grip handles round the frame.

Right clicking on the graph opens the set-up graph dialogue. Use this dialogue to scale the axes, change the axes titles, add a grid, define curve colours and symbols, define the legend position etc.



By default the graph will display the last calculated curve, in this case BGSubtracted. Please note that since in the current version it is not possible to select a different curve to be displayed, you must delete and re-insert the graph, if the calculations are modified. To delete a graph left click on it and press the delete button on the keyboard.

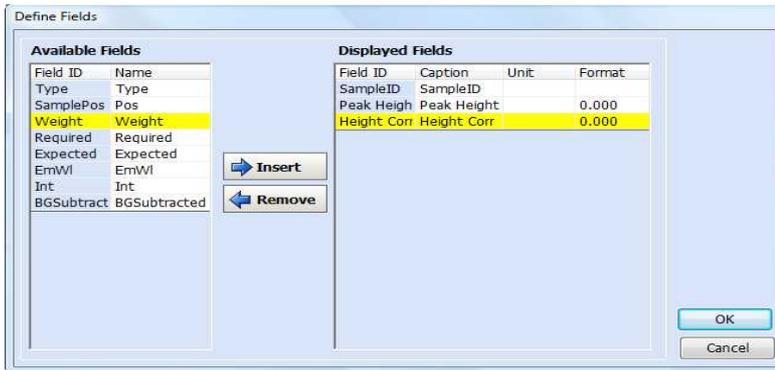
Now add a table to display the tabular data. Click at the position you want to insert the table, select Calculated Results and press insert table:



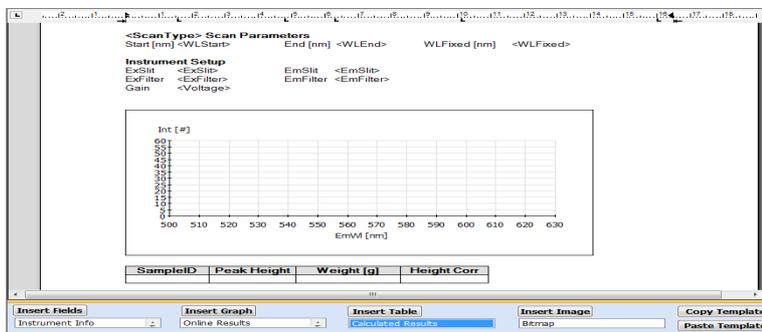
By default the table displays all fields defined in the calculator. Right click on the table to open the “Define Table” dialogue



Then press the fields button. Select “Weight” from the available fields and “Height Corr” from the displayed fields.

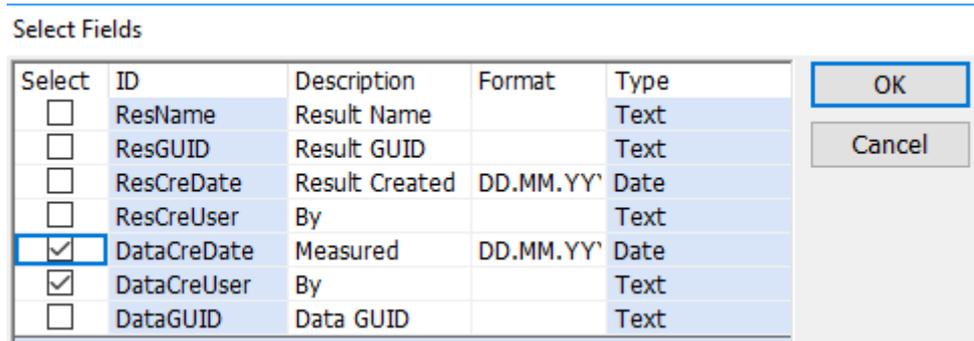


Now press the Insert button to insert the Weight field before the Height Corr field. (Please note that it is possible to also change the caption and format of the columns)

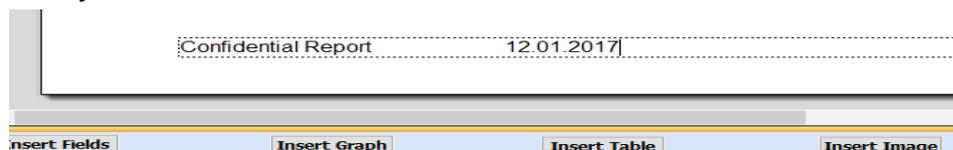


To delete a table left click anywhere inside the table and press the delete button. Do NOT select the complete table.

Add user info to the report: select ‘Data Info’ and click on ‘Insert Fields’. Select ‘DataCreDate’ and ‘DataCreUser’, for example. This will insert the UserID and creation date for the data.



Finally add a header and a footer: Click on the Header/Footer button:  And enter text as required:

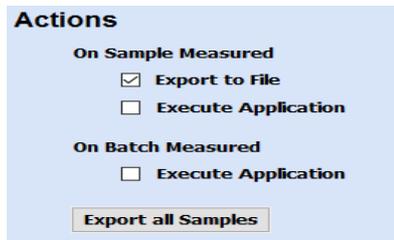


Click on the Header/Footer button to switch between header and footer fields as required.

Automatically Exporting Spectra

The StdExport module allows the user to automatically export individual spectra and/or start an external application. If automatic export is selected, then each sample's spectrum will be exported immediately on completion of the scan. External applications can be executed after each sample or after all samples have been measured.

Select 'Export to File':



Actions

On Sample Measured

Export to File

Execute Application

On Batch Measured

Execute Application

Export all Samples

In the 'Export File Definition' section, select the following:

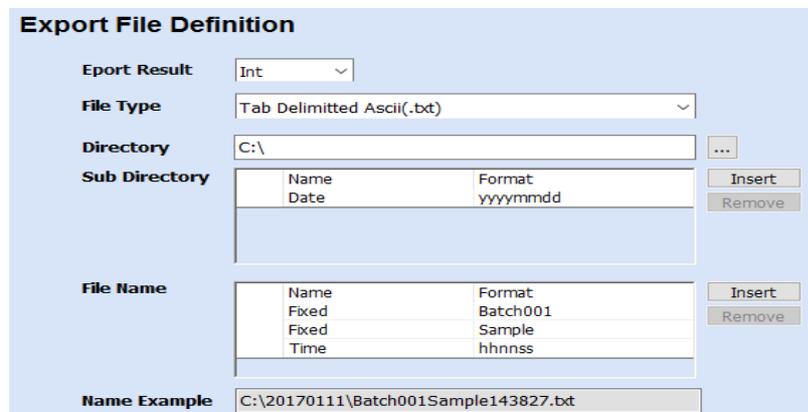
Export Result: Int (to save Intensity, i.e. raw spectral data)

File Type: Tab Delimited ASCII (for compatibility with a spreadsheet program or similar)

Directory: C: (this could be another destination for example on a server)

Sub Directory: Date (a new directory will be created with the current date as name)

File Name - Fixed: 'Batch001'; 'Sample'. The File Name information will generate a unique name for each sample as indicated in the 'Name Example' field at the bottom of the dialog.



Export File Definition

Export Result Int

File Type Tab Delimited Ascii(.txt)

Directory C:\

Sub Directory

Name	Format
Date	yyyyymmdd

File Name

Name	Format
Fixed	Batch001
Fixed	Sample
Time	hhnnss

Name Example C:\20170111\Batch001Sample143827.txt

Press the run button . The instrument is setup with the starting conditions. Then the user is prompted to enter the first sample:



Please insert sample:

Sample 1

 **Start**  **Stop**

Insert the sample, then press the start button: the data collection for the first sample is started. The collected data is displayed online in the graph.

When the measurement is complete, the user will be prompted again for a new sample. Click on stop to end the measurement.

Defining a Concentration Method

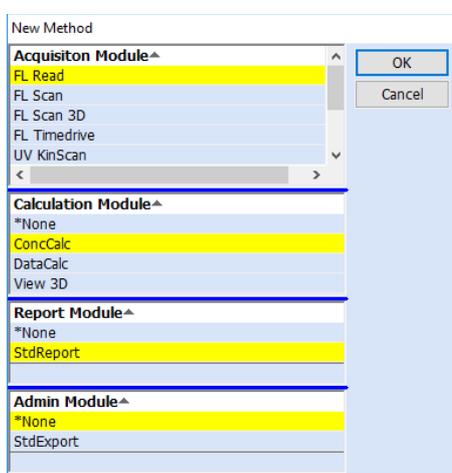
This Read method contains the following features:

- The Well Plate Reader is used for sampling
- Fixed wavelength measurement is carried out
- A calibration curve is calculated
- The concentration is calculated for each 'unknown' sample
- A report is generated which contains the calculated results

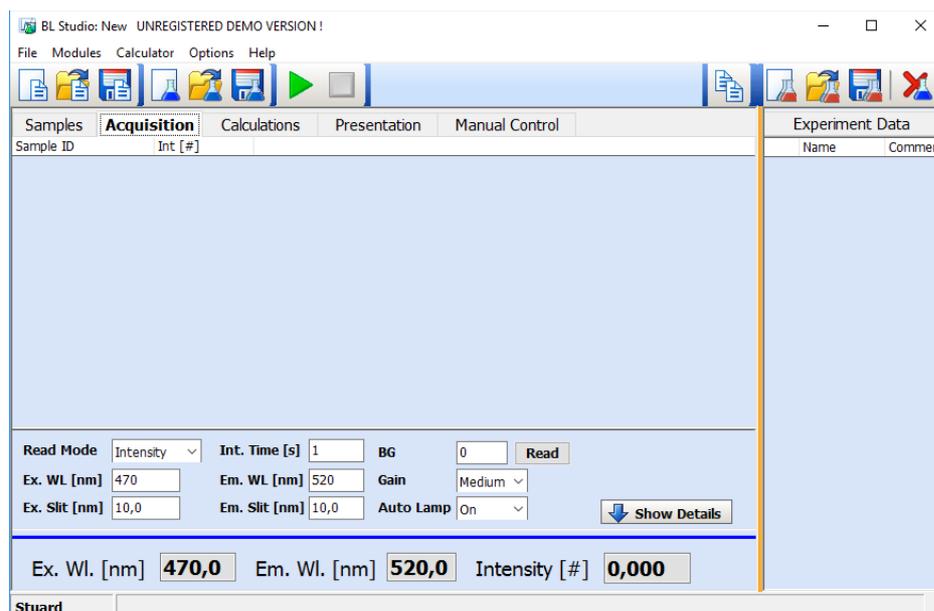
Creating a new Method

To create a completely new method press the New Method  button of the toolbox of BLDevelopment.

From the New Method Dialogue select the desired modules for each step of the method:



In this case Read, ConcCalc, and Report are selected. After pressing the OK button the selected modules are added to the BLDevelopment program:



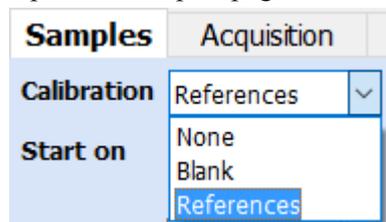
The work flow consists of a logical series of steps. These steps are represented by tabs on the main page, starting from sample handling through data acquisition, calculations and report presentation to

administrative functions such as automatic exporting of results. It is recommended to define the method step by step following the work flow.

Setting up the Well Plate Reader

Defining Sample positions

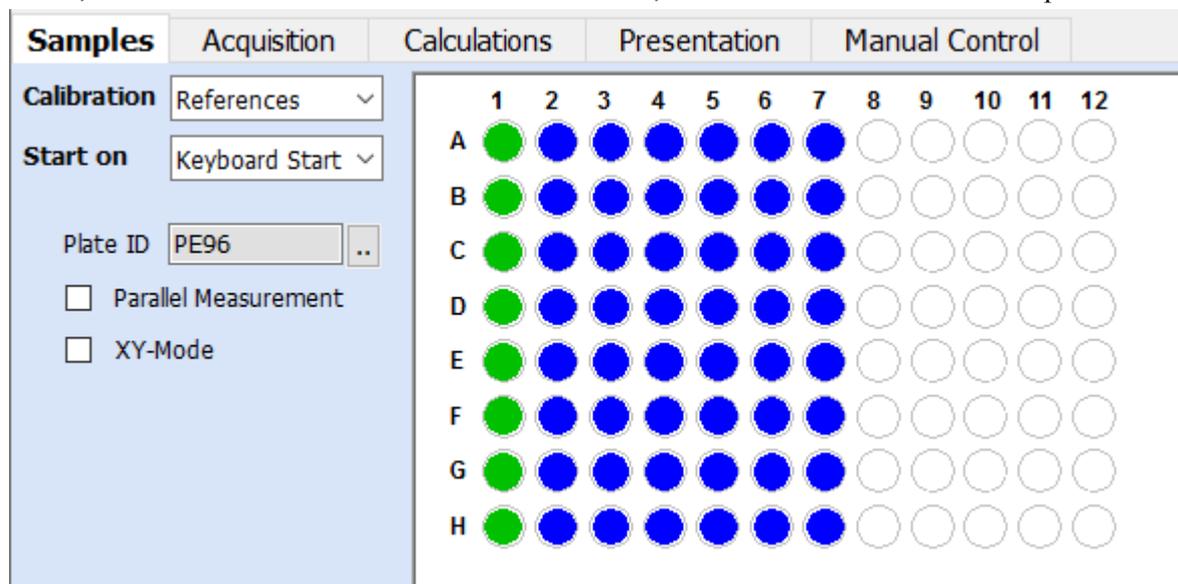
Open the Samples page. Select Calibration: References from the dropdown textbox:



The microplate is represented graphically. To define the positions for the reference samples, hold down the shift key and left click in the relevant well positions. Alternatively, hold down the shift key and draw a zoombox while holding down the left mouse button. Reference positions will be coloured green.

To define unknown samples, repeat this process with the left mouse button without holding down the shift key. Unknown samples will be coloured blue.

To remove any well positions from the measurement sequence, repeat the above steps for reference samples or unknown samples using the right mouse button instead of the left button. In the example below, wells A1 to H1 have been defined as references, wells A2 to H7 as unknown samples:



Enter a batch name for the assay if required in the Result Info textbox.

Defining Reference Sample Concentrations and sample IDs

Enter the concentration values and sample IDs for the reference samples in the Standard Info Table, enter unknown samples IDs in the Sample Info table:

Standard Info	Measured	Pos	Standard ID	Conc	Exclude
	<input type="checkbox"/>	A1	R1	0	<input type="checkbox"/>
	<input type="checkbox"/>	A2	R2	0	<input type="checkbox"/>
	<input type="checkbox"/>	A3	R3	10	<input type="checkbox"/>
	<input type="checkbox"/>	A4	R4	25	<input type="checkbox"/>
	<input type="checkbox"/>	A5	R5	50	<input type="checkbox"/>
	<input type="checkbox"/>	A6	R6	100	<input type="checkbox"/>
	<input type="checkbox"/>	A7	R7	250	<input type="checkbox"/>
	<input type="checkbox"/>	A8	R8	500	<input type="checkbox"/>

Sample Info	Measured	Pos	Sample ID	Required	Exclude
	<input type="checkbox"/>	A9	S1	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	A10	S2	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	A11	S3	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	A12	S4	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B1	S5	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B2	S6	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B3	S7	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B4	S8	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B5	S9	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B6	S10	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B7	S11	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B8	S12	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	B9	S13	<input type="checkbox"/>	<input type="checkbox"/>

Defining the Read Parameters

Open the Acquisition page. Set the parameters as shown below (this example would be suitable for fluorescein, GFP, picogreen and similar fluorophores):

Samples	Acquisition	Calculations	Presentation	Manual Control
Sample ID	Int [#]			
Read Mode	<input type="text" value="Intensity"/>	Int. Time [s]	<input type="text" value="1"/>	BG <input type="text" value="0"/> <input type="button" value="Read"/>
Ex. WL [nm]	<input type="text" value="470"/>	Em. WL [nm]	<input type="text" value="520"/>	Gain <input type="text" value="Medium"/>
Ex. Slit [nm]	<input type="text" value="10,0"/>	Em. Slit [nm]	<input type="text" value="10,0"/>	Auto Lamp <input type="text" value="On"/>
				<input type="button" value="Show Details"/>

The Concentration Calculator

Select 'Linear' fit type from the dropdown textbox:

Reference ID

r1
r2
r3
r4

Sample ID

Type **Linear** ▾

Name **Linear**

Post Ca

Exponential

Logarithmic

Enter Min=0 and Max=250:

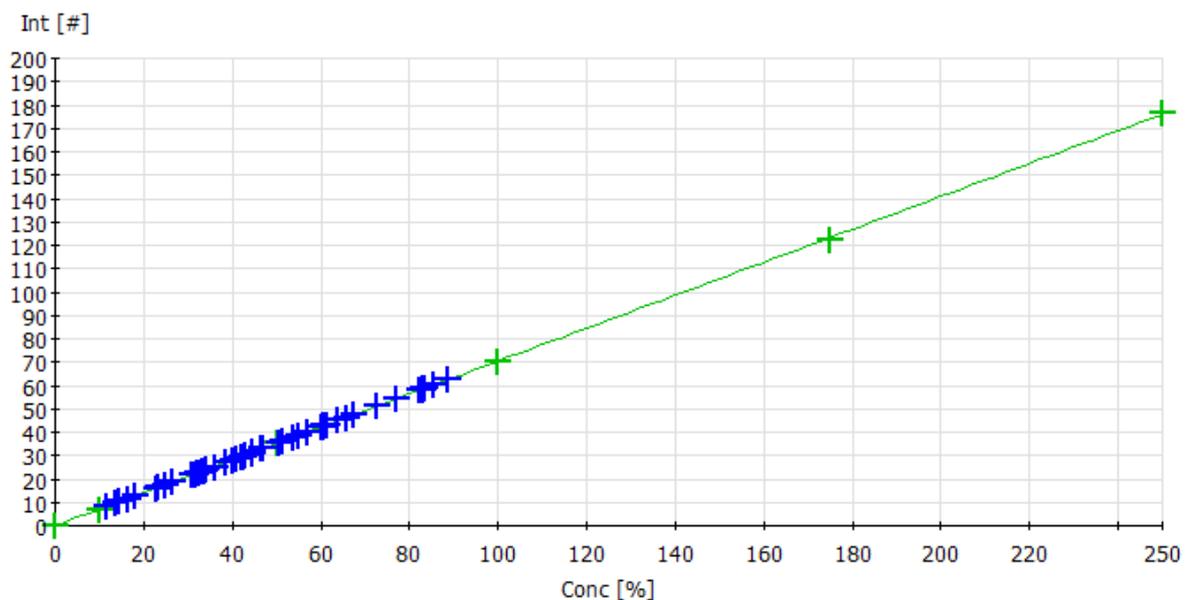
Type **Linear** ▾ through 0 Function **0.704*x - 0.031** .. Corr. Coeff. **0,99999**

Name **Conc** Unit **%** Format **0.00** Min **0** Max **250**

Insert the microplate with reference and unknown samples as defined in the samples page. Click on the start button



The sequence of reference samples will be measured, the calibration curve will be calculated and displayed, then the unknown samples will be measured. During measurement, the graphic is updated with the fitted calibration curve and measured sample values. Reference results are displayed in green, unknown samples in blue:



For each unknown sample, the 'In Calibration' column in the results table shows whether the calculated concentration lies within the predefined 0 to 250 limits.

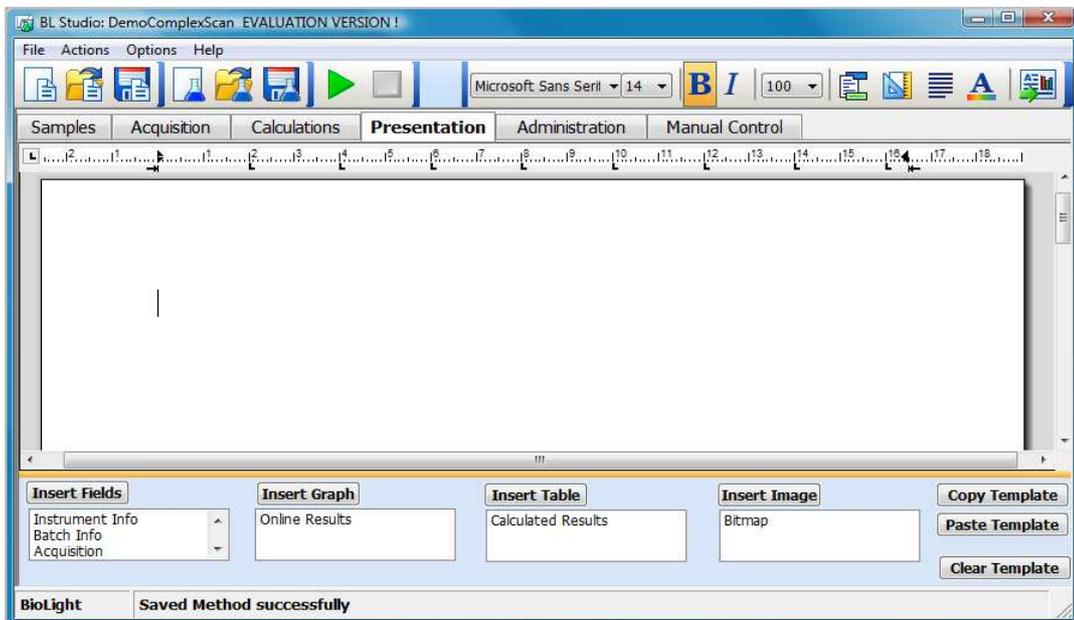
Defining the Report Template

Select the Presentation page. An empty report is shown. A report is generated by filling a report template with result data.

To edit a new template click on the “To Template” icon

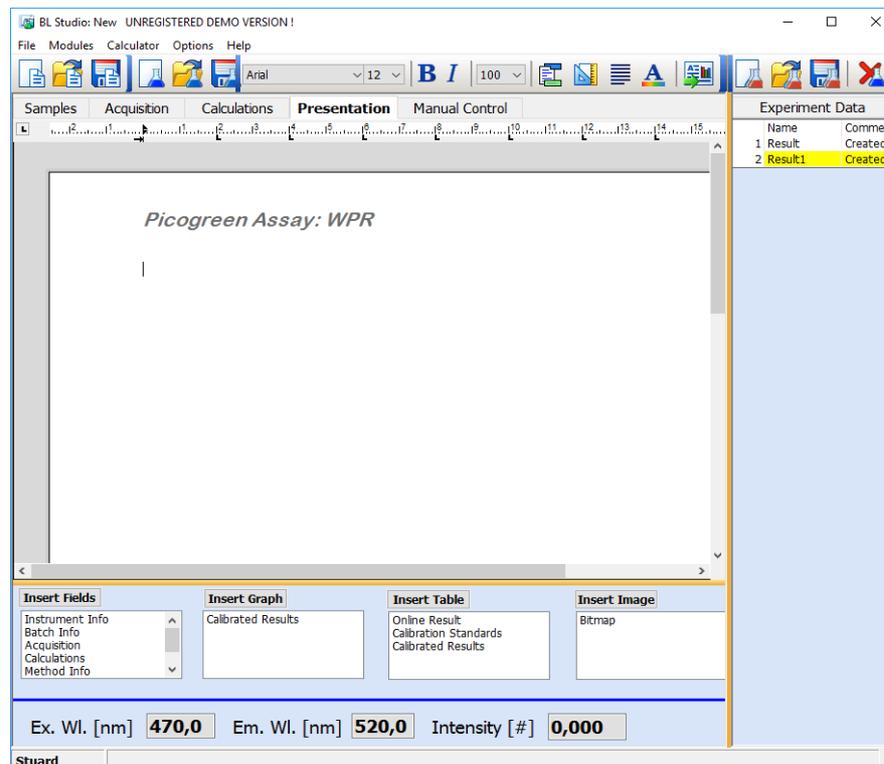


template edit mode:

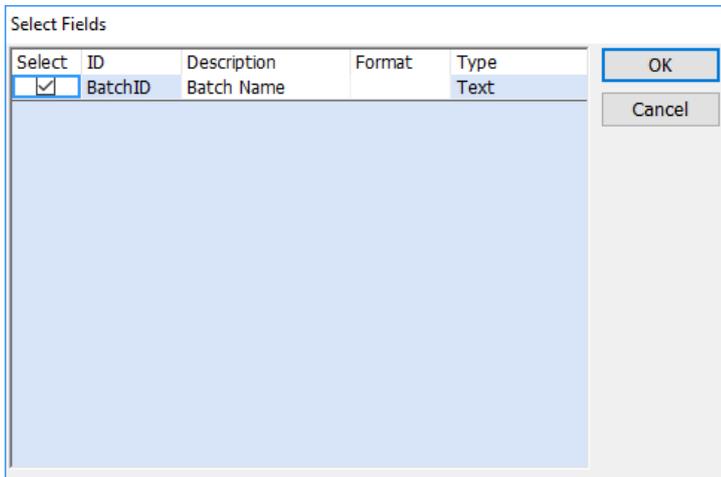


Basically the template can be edited like a MS Word document. It is for example possible to paste an existing Word document into the report (except fields). Please note, that there is no “undo” function yet, therefore it is recommended to save the method frequently.

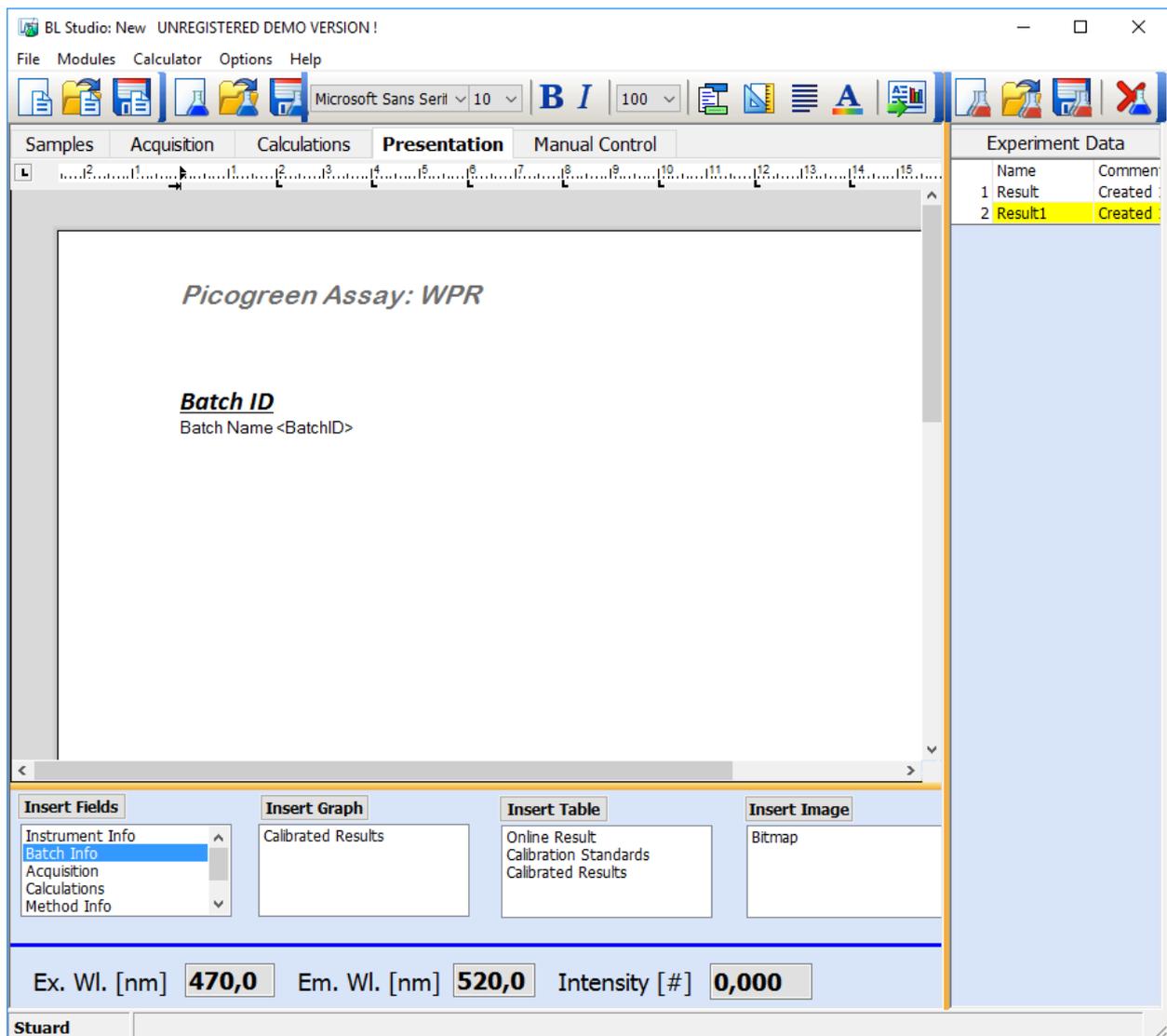
Firstly click into the template and enter a caption for the report. Then select the caption and format it:



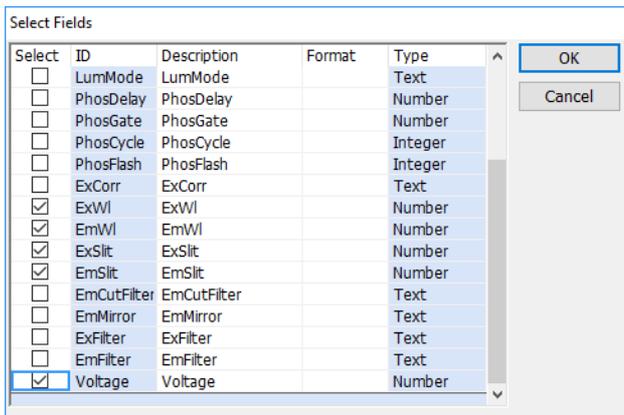
Now enter the first fields. In the template click at the position where the fields are to be inserted. Then select the Batch Info entry in the field table and press the Insert Fields button. From the “Select Fields” dialogue select the information you want to add



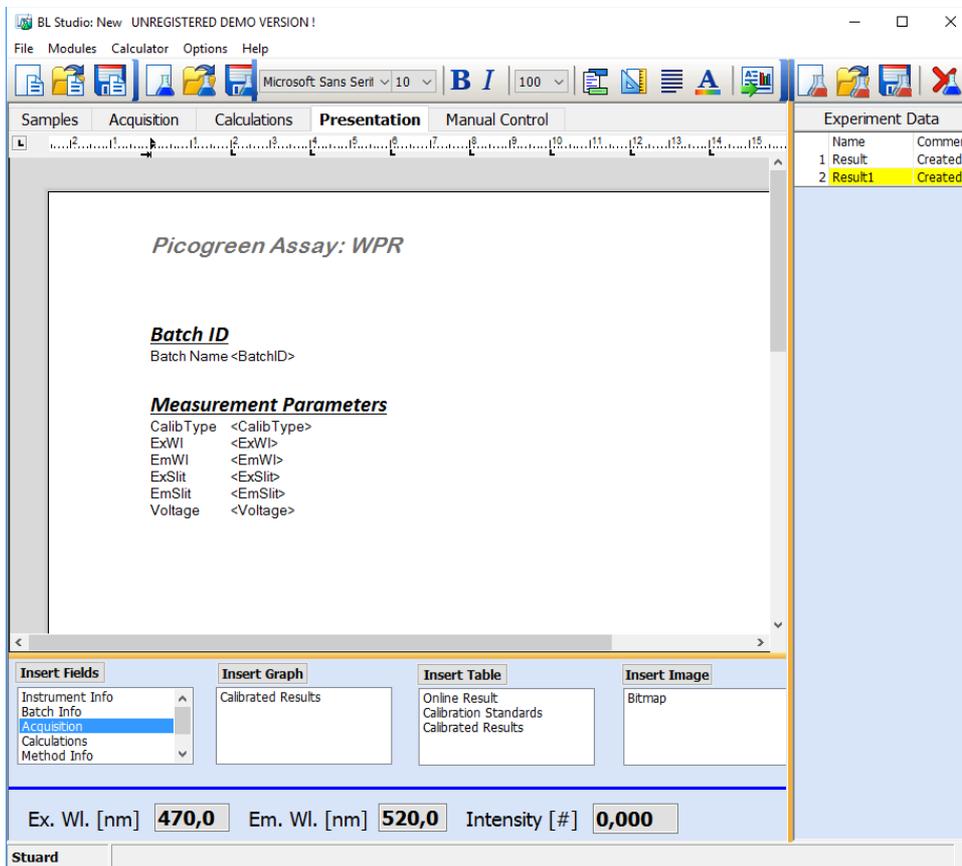
and press OK. The fields are added. All fields are marked with “<>”. Enter a text “Batch ID” above the fields, then format the fields. You can use tabs in conjunction with the ruler to define the space between the fields.



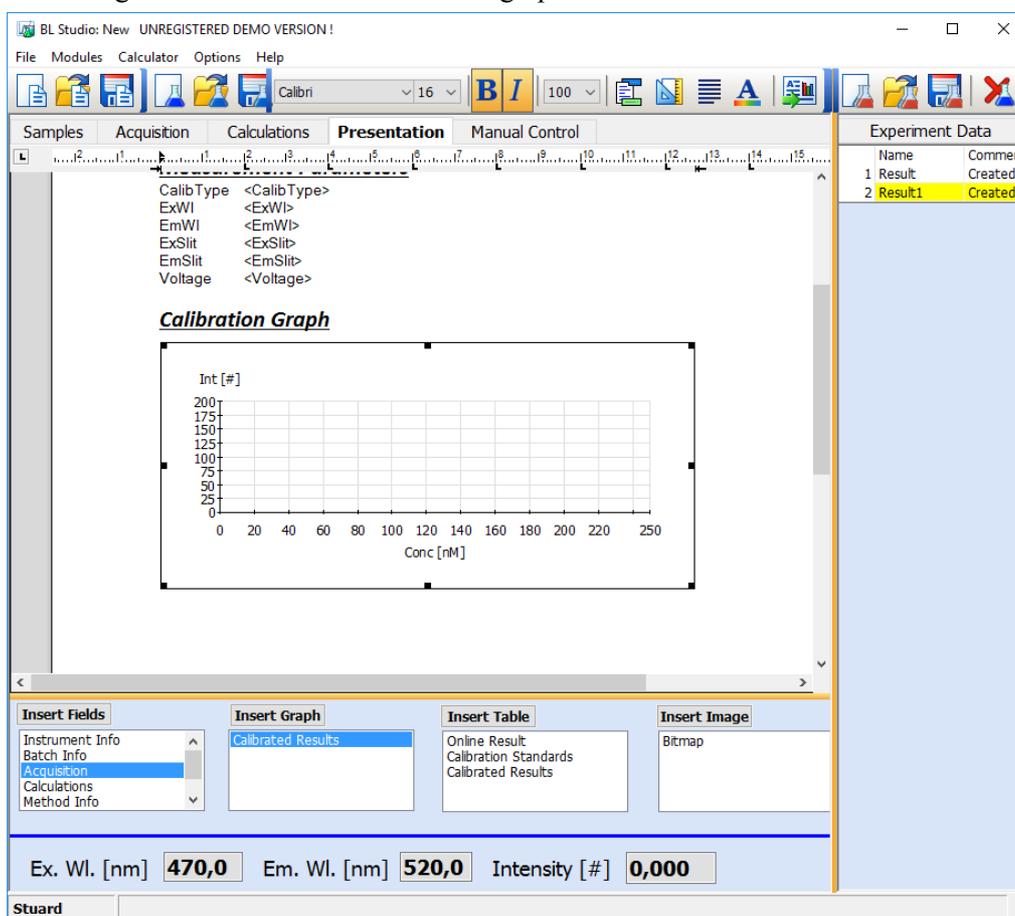
In the same way add the measurement parameters from the Acquisition entry:



Format them and add set up parameters (also from the Acquisition entry). To delete a field left click on it and press the delete button on the keyboard (do NOT select the complete field or a complete line)

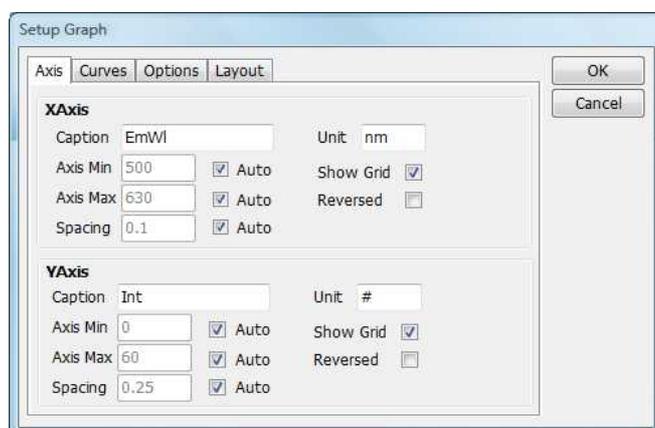


Now add the calibration graph: click in the template at the position the graph should be inserted, select Calibrated Results from the Graphs table and press the Insert Graph button and left click on the graph to select it. A rectangular frame is drawn around the graph:



The graph can now be moved and sized using the grip handles round the frame.

Right clicking on the graph opens the set-up graph dialogue. Use this dialogue to scale the axes, change the axes titles, add a grid, define curve colours and symbols, define the legend position etc.



To delete a graph left click on it and press the delete button on the keyboard.

Now add the first table to display the calibration results. Click at the position you want to insert the table, select Calibration Standards and press insert table:

The screenshot shows the BL Studio software interface. The main window displays a graph with 'Int [#]' on the y-axis (0 to 200) and 'Conc [nM]' on the x-axis (0 to 250). Below the graph is a table titled 'Calibration Results' with the following structure:

<i>Reference ID</i>	<i>Int [#]</i>	<i>Conc [nM]</i>

At the bottom of the interface, there are four panels for inserting elements:

- Insert Fields:** Instrument Info, Batch Info, Acquisition (selected), Calculations, Method Info.
- Insert Graph:** Calibrated Results.
- Insert Table:** Online Result, Calibration Standards (selected), Calibrated Results.
- Insert Image:** Bitmap.

Below these panels, there are input fields for 'Ex. Wl. [nm]' (470,0), 'Em. Wl. [nm]' (520,0), and 'Intensity [#]' (0,000). The status bar at the bottom left shows 'Stuard'.

To delete a table left click anywhere inside the table and press the delete button. Do NOT select the complete table.

Now add the second table to display the calibrated results. Click at the position you want to insert the table, select Calibrated Results and press insert table:

The screenshot shows the BL Studio software interface. At the top, there is a menu bar with 'File', 'Modules', 'Calculator', 'Options', and 'Help'. Below the menu bar is a toolbar with various icons. The main window is divided into several sections:

- Graph:** A plot with 'Conc [nM]' on the x-axis (0 to 250) and a y-axis. The plot area contains two tables.
- Calibration Results Table:**

Reference ID	Int [#]	Conc [nM]
- Calibrated Results Table:**

Sample ID	Int [#]	Conc [nM]
- Experiment Data Panel (Right):**

Name	Comment
1 Result	Created
2 Result1	Created
- Bottom Panel:** Contains 'Insert Fields', 'Insert Graph', 'Insert Table', and 'Insert Image' buttons. The 'Insert Table' dropdown is open, showing 'Online Result', 'Calibration Standards', and 'Calibrated Results' (selected). Below these are input fields for 'Ex. Wl. [nm]' (470,0), 'Em. Wl. [nm]' (520,0), and 'Intensity [#]' (0,000).

Finally add a header and a footer. Click on the Header/Footer button:



And enter text as required:

The screenshot shows the software interface with a header and footer area. The header area contains the text 'Confidential Report' and '12.01.2017'. Below the header area is a footer area with the same text. The 'Insert Fields', 'Insert Graph', 'Insert Table', and 'Insert Image' buttons are visible at the bottom of the interface.

Click on the Header/Footer button to switch between header and footer fields as required.

Defining a 3D Scan-Method

This 3D Scan method contains the following features:

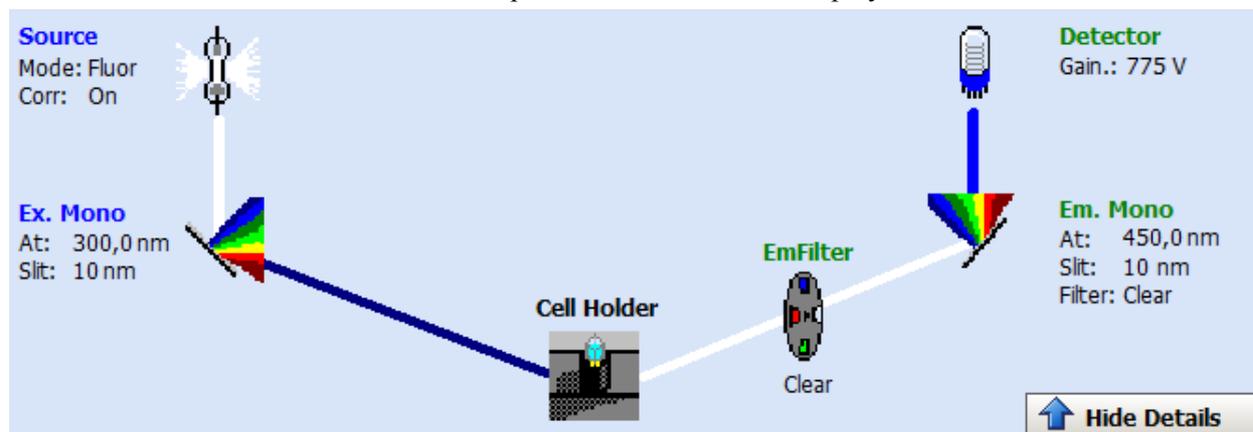
- 3D EEM scan will be collected and displayed in contour mode
- Emission cut-off filter is selected to reduce second-order stray light artefacts
- ‘Cuts’ are defined to show Excitation and Emission spectral data from the 3D matrix
- Data from these cuts is copied to clipboard for further use in a spreadsheet program
- The entire 3D EEM matrix is copied to clipboard for further use in a spreadsheet program

Defining the 3D Scan Parameters

Open the Acquisition page. Enter parameters as shown below:

Scan X-Axis	Emission	Start [nm]	450	End [nm]	650	Speed [nm/min]	500
Scan Y-Axis	Excitation	Start [nm]	300	End [nm]	500	Step [nm]	5
Ex. Slit [nm]	10,0	Gain	Medium				
Em. Slit [nm]	10,0	Auto Lamp	On	Show Details			

Click on ‘Show Details’. The instrument optical schematic will be displayed:



Click on the Em. Mono icon:



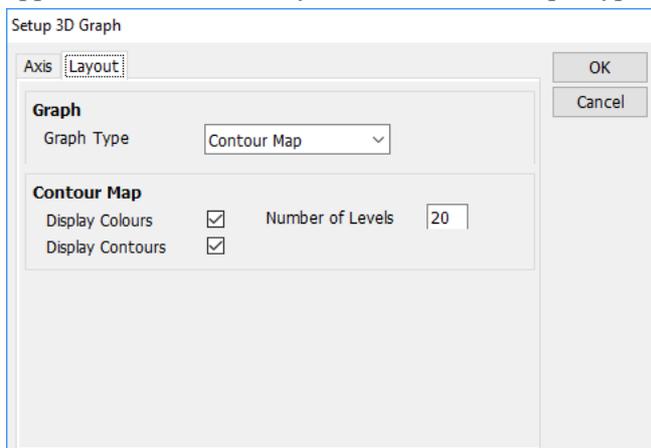
Select ‘390nm Cut-off’ from the Emission Cut-Off Filter dropdown textbox:

Set Instrument Parameters		
Emission Wavelength [nm]	450	OK
Emission Slit [nm]	10	Cancel
Emission Cut Off Filter	390nm Cut-of	

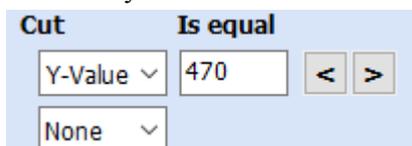
Click on ‘OK’. Click on ‘Hide Details’.

The 3D Viewer

Open the Calculations page. Right click in the graphics area, select 'Setup Graph'. In the dialog which appears, click on the 'Layout' tab. Select 'Graph Type' 'Contour Map' as shown below:

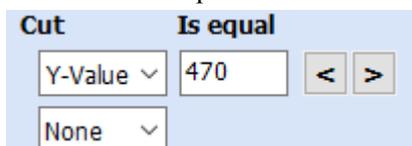


Click on 'OK'. In the bottom right part of the window, click on the top 'Cut' dropdown textbox (this is selected by default as 'none'). Select Y-Value. Enter 470 in the 'Is Equal' textbox:



This will automatically extract and display the emission spectrum for excitation at 470nm.

Click on the lower 'Cut' dropdown textbox (this is selected by default as 'none'). Select X-Value. Enter 520 in the 'Is Equal' textbox:



This will automatically extract and display the excitation spectrum for emission at 520nm.

Press the run button . The instrument is setup with the starting conditions. Then the user is prompted to enter the first sample:



Insert the sample, then press the start button: the data collection for the first sample is started. The collected data is displayed online in the graph. After the second individual spectra has been collected, the 3D Viewer in the Calculations page will display the 3D EEM graph, updating the X- and Y-Value cuts as data is collected.

When the measurement is complete, the user will be prompted again for a new sample. Click on stop to end the measurement.