OLYMPUS

INSTRUCTIONS

scan^R

AUTOMATED IMAGE AND DATA ANALYSIS SOFTWARE

This instruction manual describes the Olympus scan^R Automated Image and Data Analysis Software for Life Science. To ensure safety, obtain optimum performance and familiarize yourself fully with the use of these products, we recommend that you study this manual thoroughly before operation. Together with this manual, please also read the scan^R Automated Image Acquisition Software manual, the Hardware and Software user manual as well as the manuals of all other devices controlled by the software in order to understand general operation methods. Retain this manual in an easily accessible place near a system for future reference.

OLYMPUS SOFT IMAGING SOLUTIONS GMBH
Johann-Krane-Weg 39
D-48149 Münster
Tel: +49 251 - 798 00 0
Fax +49 251 - 798 00 6060
Email: info@olympus-sis.com

www.olympus-sis.com

OLYMPUS

Imaging Excellence

Manual version: October 2021

END USER LICENSE AGREEMENT

IMPORTANT-READ CAREFULLY: Below you will find the contractual agreements governing the use of this imaging Software. These conditions apply to you, the user, and to OLYMPUS SOFT IMAGING SOLUTIONS GmbH. With any of the following actions you explicitly agree to be bound by the conditions of this contract: purchasing the Software, opening the package, breaking of one of the seals or using the Software.

In case you do not agree with any of the conditions of this contract, please return all parts of the Software including manuals, the license certificate and/or the Software protection key before using and without delay. Remove all Software installations of the Software from any computer you might have installed it on. Return all electronic media of the Software or completely destroy all electronic media of the Software and send proof that this has been accomplished. For a refund, please return everything to where you purchased the Software.

1. Scope

(1) This License Agreement explicitly covers only the Software diskettes or other media you received with the purchase and the Software stored on these media, the manuals, the license certificate and/or the Software protection key, together referred to as “Software”, as far as they were developed and produced by OLYMPUS SOFT IMAGING SOLUTIONS GmbH.

(2) The Software may be modified or enhanced by OLYMPUS SOFT IMAGING SOLUTIONS GmbH or its licensors for debugging, upgrading or any other purposes. The Software shall contain such modified or enhanced version. You can only use either the previous version or the modified or enhanced version and cannot use both of the versions. You have, however, no right to receive modifications or enhancements or upgrades, unless those are provided for warranty purposes and you have a legitimate claim towards warranty.

(3) The Software may include (i) some open source software and (ii) other third party software (collectively, "Third Party Software"). Regarding the use of the Third Party Software, you shall follow the license terms and conditions applied to such Third Party Software. Any part of this Agreement does not restrict, change or alter any of your rights and obligations imposed by the license terms and conditions of the Third Party Software.

2. User rights

(1) OLYMPUS SOFT IMAGING SOLUTIONS GmbH permits the User, for the duration of this contract, to use the Software on a single computer and a single terminal on that computer within the Territory of installation. This license is explicitly non-exclusive, i.e., the User does not have an exclusive right to use the Software. As a licensed user you can copy the Software from one computer to another by using a computer network or other storage devices, as long as it is assured, that the Software can only be used on a single computer and terminal at any time and that the conditions set forth under 4 are observed. For the purpose of this License, the term Territory shall mean (i) for installations in a member state of the European Union: all member states of the European Union and (ii) for installations in a country outside of the European Union: the country in which the computer is located on which the Software is installed.

(2) The User has the right to produce a copy of the Software only for backup purposes.

(3) The Software is provided with a license certificate or a dongle. It is the responsibility of the owner to take care of the license certificate and/or of the dongle. In case of loss, neither the license certificate nor the dongle will be replaced by OLYMPUS SOFT IMAGING SOLUTIONS GmbH.

3. Transfer

You may permanently transfer all of your rights under this Agreement only if the recipient agrees to the terms and conditions of this Agreement and if you destroy and will not keep any copies of the Software in your possession.

4. Copyright

(1) OLYMPUS SOFT IMAGING SOLUTIONS GmbH or its subsidiaries remain owners of the copyright to the Software and its documentation. With the purchase, the User obtains ownership of the diskettes or other physical storage devices (excluding the Software and other data contained thereon), the manuals, and the license certificate and/or the Software protection key. Furthermore, a right to use the Software as defined in Sec. 2 will be granted.

(2) OLYMPUS SOFT IMAGING SOLUTIONS GmbH reserves the right to all publications, duplication, editing, and marketing of the Software and the Software documentation.

Without prior written permission the User may not:

- change, translate, de-compile or de-assemble the Software,
- copy any of the written or printed documentation of the Software,
- rent, lease, or license the Software to a third party,
- use the Software protection different than described in this contract.

(3) OLYMPUS SOFT IMAGING SOLUTIONS GmbH is the legal owner of all copyrights and trademarks of the Software and documentation. Copyrights and trademarks are protected by national and international law. OLYMPUS SOFT IMAGING SOLUTIONS GmbH reserves all rights, which are not explicitly expressed in written form.

5. Warranty

(1) OLYMPUS SOFT IMAGING SOLUTIONS GmbH warrants that the Software delivered under this Agreement for the period of 12 months of the date of purchase works in all major aspects according to the descriptions in the specification. OLYMPUS SOFT IMAGING SOLUTIONS GmbH shall not be liable for the Software being suitable for a particular purpose unless otherwise agreed upon to which you intend to put it.

(2) This warranty does not cover defects in the Software which are due to improper installation or changes to the program which you have undertaken.

(3) OLYMPUS SOFT IMAGING SOLUTIONS GmbH does not warrant, that the Software is defect free, that the Software fulfills the specific requirements of the User, or that the Software works with other Software provided by the User.

(4) Where any claim in respect to the Software which is based on any defect in the quality or condition of the Software to meet the specification is notified to OLYMPUS SOFT IMAGING SOLUTIONS GmbH in accordance with these conditions, OLYMPUS SOFT IMAGING SOLUTIONS GmbH shall be entitled at its own discretion to either replace the Software free of charge or repair the Software. Should OLYMPUS SOFT IMAGING SOLUTIONS GmbH fail to repair the Software within a reasonable time frame more than three times, the customer is entitled to resign from this contract.

(5) The warranty is provided by OLYMPUS SOFT IMAGING SOLUTIONS GmbH under the condition that the total price for the Software has been paid by the due date.

(6) You are obligated to establish sufficient precautions for the case of Software malfunctions (for example by regular data storage, frequent backups, verification of results, emergency planning). It is your responsibility to provide a suitable system environment.

6. Liability

(1) Whatever the legal grounds, liability for damage claims based only on ordinary negligence against OLYMPUS SOFT IMAGING SOLUTIONS GmbH (including its vicarious agents) shall exist only if OLYMPUS SOFT IMAGING SOLUTIONS GmbH breaches a basic/cardinal obligation under this Agreement. In this event the amount of the claims are limited to typical and foreseeable damages. In cases of gross negligence the liability shall be limited to the amount the User paid for the Software.

(2) Nothing in this clause shall affect personal injury and property damage claims based on reasons of product liability.

7. Contract duration, legal consequences of violating the license

(1) The contract is deemed to be in force for an unspecified period. The User rights are automatically terminated if one of the conditions of the contracts has been violated.

(2) In case of a contract violation the User has to return the original storage devices and all copies thereof including all modified copies, all printed and written documentation, as well as the license certificate and/or the Software protection key to OLYMPUS SOFT IMAGING SOLUTIONS GmbH, or the User has to destroy these items.

(3) In addition OLYMPUS SOFT IMAGING SOLUTIONS GmbH reserves the right to file a lawsuit to claim reparations for damages, non-compliance, or removal of the Software in case of license violations. The following laws and/or conditions are in effect: the conditions of this contract, copyright laws, and the laws of the German civil code (BGB).

8. Applicable Law, Place of Performance and Jurisdiction

This Agreement shall be governed by German law. The Convention on Contracts for the International Sale of Goods (CISG) shall not apply. Place of Performance and Jurisdiction shall be at Münster, Germany.

9. Severability

In the event a provision of this Agreement is or becomes invalid or should this Agreement be incomplete, the validity of the remaining provisions shall not be affected thereby. The invalid or incomplete provision shall be substituted or supplemented, respectively, by an appropriate one reflecting - to the extent legally permissible - as closely as possible what the parties hereto had originally intended or would have intended according to the sense and purpose of this Agreement, if the specific issue had been taken into consideration by them

Contents

1 Introduction

Thank you very much for purchasing Olympus' Screening Station for Life Science and for your confidence in our products and services.
The scan^{R} Analysis Software is designed for the automated analysis of images that were acquired by the Olympus scan^{R} Screening Station and with the scan^{R} Acquisition Software. The software is intended for the use in biomedical research.

> ⚠ The scan^{R} Analysis Software, the scan^{R} Acquisition Software as well as the hardware components of the Olympus scan^{R} Screening Station for Life Sciences are for research use only.

1.1 Abstract

This user manual will guide you through the usage of the Analysis software of the Olympus Screening Station scan^{R}. It will assist you in setting up efficient and reliable assays from scratch. This part of the scan^{R} software is intended to be used for the analysis, quantification and navigation through your data and results. The Analysis software of scan^{R} allows you to run the analysis during and in parallel to the-acquisition or in "offline mode" afterwards.

Special care has been taken to guarantee correct and accurate information within this documentation, although this is subject to changes due to further development of the screening system. Thus, the manufacturer cannot assume liability for any possible errors. We would appreciate reports of any mistakes as well as suggestions or criticism.

1.2 Technical support

If you find any information missing in this manual or you need additional support, please contact Olympus directly.

2 Main User Interface

This chapter explains the features of the image displays and briefly introduces the different menu points and buttons accessible from the main user interface.

2.1 General

The scan^{R} Analysis main graphical user interface contains four histograms and an image viewer window. The functions of these histograms are explained in Chapter 2.4, *Managing histograms and scatter plots*.

The image viewer window shows the image with the object corresponding to a data point selected in a histogram. A description of the image viewer functions is given in Chapter 2.5, *Using the Image Viewer*.

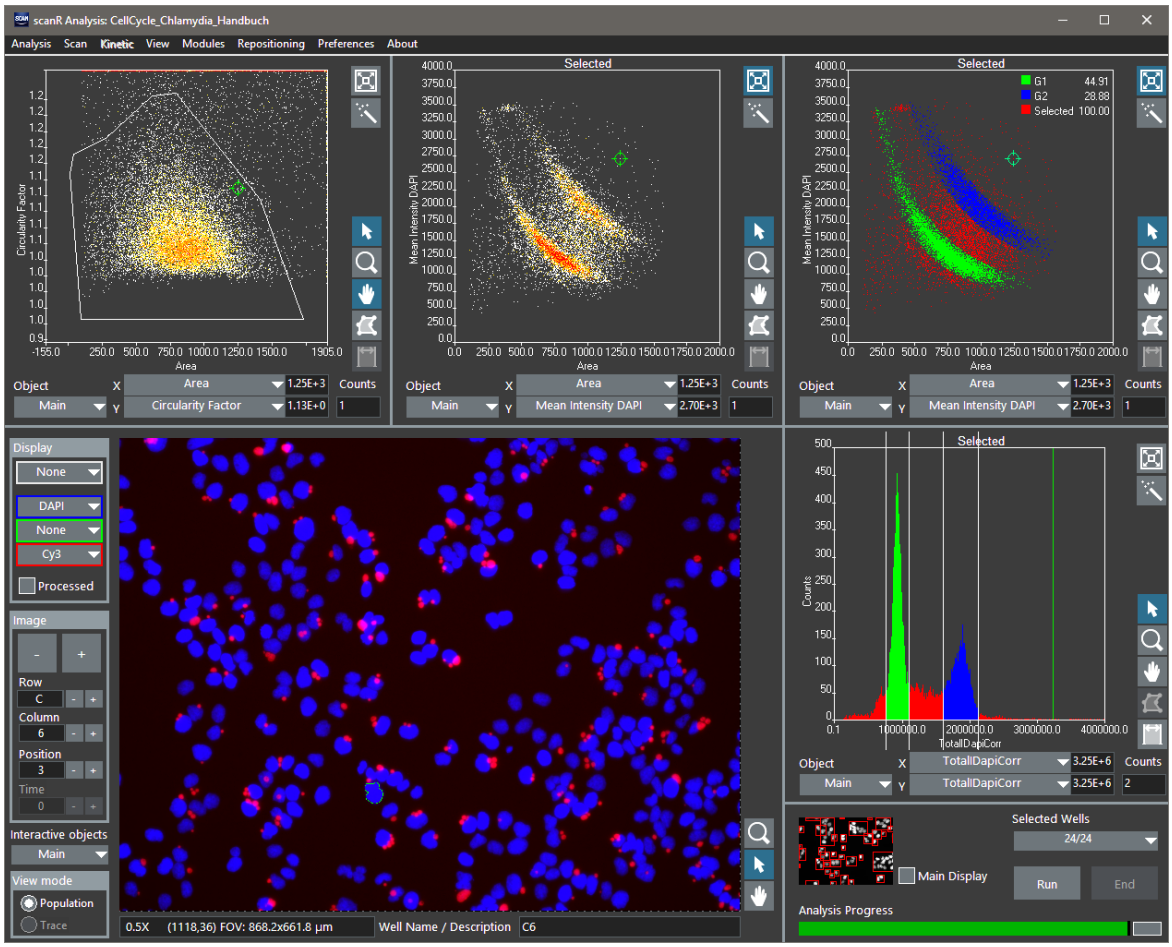
Navigation through the images of a scan is possible with the tools in the **Image** box. Color channel selection is done with respective drop-down lists in the **Display** box.

An analysis can be started and followed online with the tools and displays in the field at the lower right of the main window.

The menu bar at the top contains several drop-down menus and commands. They are listed in the following overview:

Analysis ▸ Run: starts the analysis with the current settings

Analysis ▸ Batch Run: starts multiple analyses with the current or individual assay settings per scan.



Analysis ▸ Open: opens a previous analysis (*.sca-file)

Analysis ▸ Save/Save as: saves the current analysis (*.sca-file)

Analysis ▸ Export Table: exports the values determined for each detected object to a spread sheet. The values exported depend on the active view mode. When also sub-objects are detected not only one file is exported but for every sub-object a separate list is exported. The values that are exported depend on the active view: population view / trace view (see Chapter 4.4.4)

Analysis ▸ Assay Settings: opens the “Assay settings” dialog to define and review the analysis pipeline

Analysis ▸ New Assay: Resets the assay setting to default values as starting point for new assay definitions

Analysis ▸ Load Assay: loads an existing assay (*.say-file). In contrast to the *sca-file the *.say-file contains only the analysis definitions, i.e. the operations to perform on a data set but not the results of a specific analysis.

Analysis ▸ Save Assay: saves the current assay (*.say-file)

Analysis ▸ Assay Gating: opens the Gate Manager (see Chapter 4.2.2)

Analysis ▸ Exit: exits the analysis

Scan ▸ Open: opens a scan and assigns it to the current assay. The file types that can be opened are the scan^R experiment descriptor files (*.xml-format)

Scan ▸ Open Last Acquired: opens the last acquired scan and assigns it to the current assay.

Scan ▸ Relink images: re-links acquired images to an analysis. To do so, navigate to the folder where the images are stored and select **Current Folder**.

Scan ▸ Custom conversion: converts a data set acquired with another software into the scan^R format (see Chapter 7.1.1)

Scan ▸ Plate…: selects a set of wells for analysis and data navigation; allows also to display a well overview, i.e. an overview of the images that were acquired in a well. (see Chapter 2.7)

Scan ▸ Reassign wells: enables you to reassign wells (see Section 2.7 *Interactive plate result view*).

Scan ▸ Scan Info: displays the path to the image data

Scan ▸ Settings: displays the settings to a scan

Kinetic ▸ Trace View: toggles between the **Population** and **Trace View** modes.

Kinetic ▸ Configure Tracer: opens the **Trace Configuration** window to select how the object tracking is to be performed.

Kinetic ▸ Define Parameters: opens the **Trace Parameters** window to select trace analysis operations.

Kinetic ▸ Show traces: opens the **Trace Viewer** that visualizes the trace graphs.

View ▸ Adjust Display…: display properties for the RGB display (affects only the displays).

View ▸ Well Results…: opens the Well Results window (see Section 4.4, *Well results*).

View ▸ Parameter View: lists all parameters for a selected object, that were determined during analysis

View ▸ Log File: opens a log file viewer which provides technical information about the analysis process and possible issues.

Modules ▸ Object Finders: list and configure available Object Finder Modules (Chapter 7.4, *Libraries*)

Modules ▸ Object Analyzers: list and configure available Object Analyzer Modules (Chapter 7.4, *Libraries*)

Modules ▸ Image Processors: list and configure available Image Processing Modules (Chapter 7.4, *Libraries*)

Modules ▸ VC Processors: list and configure available Virtual Channel Modules (Chapter 7.4, *Libraries*)

Repositioning ▸ Interactive: the scan^{R} screening system moves to the position of the selected object

Preferences: Gives access to some preferences including galleries, directories, data export and settings for repositioning and reclassification.

About: Provides information about the software version and licenses, and offers to create a “Support Package”. This is a .zip file containing log files and technical information which can be used for error diagnosis by Olympus support.

2.2 Preferences

The general preferences can be defined in the **Preferences** dialog.

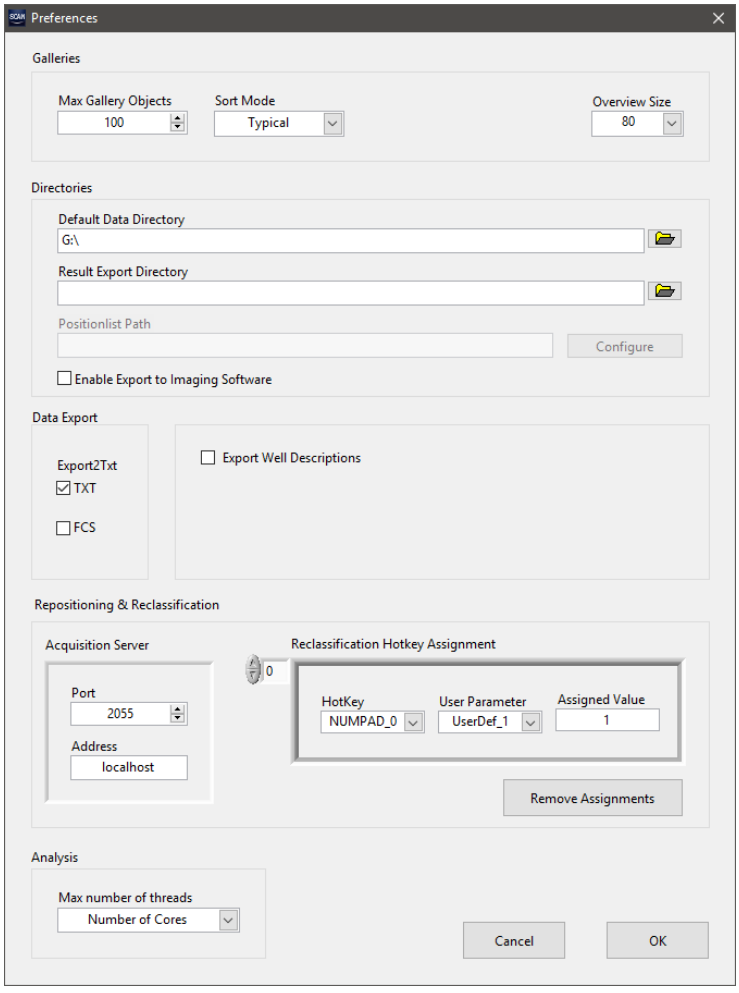
Max Gallery Objects. Sets the maximum number of objects that are displayed in a gallery.

Sort mode. The options are **Typical**, **Random**, **yx** and **–yx**. When **Typical** is set, the galleries display the objects which are closest to the center of gravity of the selected region or histogram in the order of distance to that center. **Random** displays randomly selected objects within the selected region. The options **yx** and **–yx** allow to create a gallery that is ordered by one parameter.

Overview Size. The pixel size of a single position for the well overview can be set to **80**, **160**, **240** pixels (see Chapter 2.7).

Default Data Directory. Enter the directory where the data are read by default (used for **Scan ▸ Open…**)

Result Export Directory. Enter the directory where the results are to be stored. When this field is empty the results will be stored in the ***<scan directory>\Population Results***. The results of a tracking analysis will be stored in the ***<scan directory>\Trace Results***.



Enable Export to Imaging Software. Configuration of the export of position lists of gated object to an imaging software (cellSens or xcellence). For further information regarding this functionality, see Chapter 7.6 *Export to imaging software*.

Export2Txt. The results can be exported as .txt files or as .fcs files. (For export to .fcs format, see Chapter 7.3). In case of exporting to .txt files, it can be chosen if the well name/description (see Chapter 2.7) is included in the results file or not.

Acquisition Server. Set the Port and the Address for communication with the scan^R screening system for experiments with repositioning.

Reclassification Hotkey Assignment. Define hotkeys available in galleries and histograms to assign a value to the user-defined parameters **UserDef_1** and **UserDef_2**. These user-defined parameters can be used to gate objects based on user input.

Max number of threads. Defines a maximum number of threads to be used for the analysis. Restricting the number of threads can be useful when running memory intense analysis because the memory usage depends on the number of parallel running threads.

2.3 The scan^R data structure

scan^R analysis data can be separated into the acquired images and the analysis assay being applied to them. The acquired images as a whole are called a ***scan***; it includes the individual images and their acquisition settings like color channels, exposure time, plate information etc. An assay describes the processing and analysis steps applied to extract data out of the images.

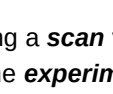
This separation between assay settings and acquired images allows the reuse of once adapted assay settings for different scans.

The images acquired during a scan^R scan are stored as 16-bit *.tif files in a ***data*** subfolder in the experiment scan storage folder.

Additionally, the scan settings are stored in an ***experiment_descriptor.xml*** and the stage positions in the ***Acquisitionlog.dat*** file.

The scan^R Analysis software serves for the analysis of the scans. The instructions (assays) for these analyses are stored in the **scan^R Analysis/Assays** folder as ***.say** files. These files can be loaded via **Analysis ▸ Load Assay…** to then apply the assay on a scan data set.

Once an analysis has been performed, i.e. the analysis steps defined in an assay have been applied to scan, a ***.sca** file is generated and can be stored in the experiment storage folder. These files contain all analysis data including histograms and scatter plots etc. ***.sca** files can be loaded via **Analysis ▸ Open…** to revisit the analysis results.



When opening a ***scan*** via ***Scan ▸ Open…*** the experimental settings of a scan will be loaded by reading the ***experiment_descriptor.xml*** file and the images of the scan will be available for browsing and analysis. In contrast to this when opening an ***analysis*** (*.sca) file via **Analysis ▸ Open…** all results of the saved analysis will be opened together with the images, allowing review of the results.

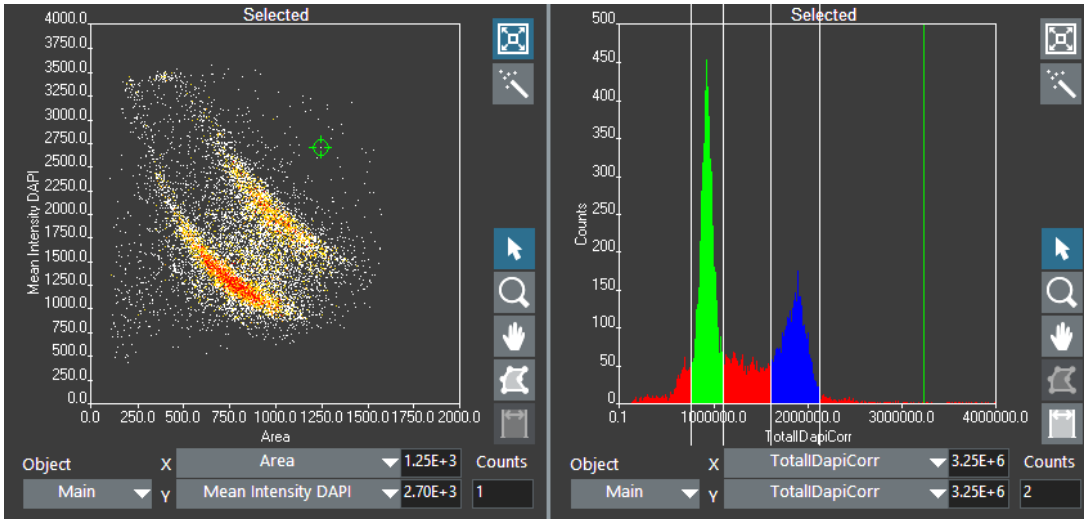
2.4 Managing histograms and scatter plots

Histograms are used in scanR for data representation, classification and navigation. scanR uses 2-D and 1-D histograms following the common representation in cytometry.

A ***1-D histogram*** shows the frequency distribution of only one parameter. A 1-D histogram is created when the same parameter for X and Y is selected. On the X-axis of the histogram the selected parameter and on the Y-axis the number of counts is plotted.

A ***2-D histogram*** (scatter plot) is a plot of one parameter of a number of objects against a second parameter. Color-coding is used as "third dimension" to represent the frequency of occurrences.

The data displayed in a histogram are assigned to an object type (main-/ sub-object). The detection of the objects is defined by the assay. The object type can be chosen from the **Object** drop-down list underneath the histogram. The **X** and **Y** drop-down lists are then automatically updated to correspond to the measured parameters of the chosen object type (see assay definition). The **X** and **Y** drop-down lists are used to change the parameters displayed in a histogram. The axes of abscissa and ordinate are labeled with the chosen parameter.



A selection of ***buttons*** are located in the lower right corner of each histogram. They allow toggling between the ***Navigation*** mode, the ***Zoom*** mode, the ***Pan*** mode, and the ***Region Drawing*** mode. In the upper right corner buttons are located for ***auto scaling*** and ***automatic region*** drawing actions.

Navigation 

The **Navigation** button with the **pointer** symbol allows navigating within the data. Each data point within a histogram is directly linked to the object from which it is derived. A selected data point is highlighted by a red circle in all histograms in navigation mode as long as the data point is within the displayed area. The corresponding object is displayed in the **Image Viewer**. The X and Y values of the data point are displayed next to the **X** and **Y** drop-down lists. Holding and dragging the mouse using this tool allows to virtually following the objects changes within the parameter set. The navigation tool also allows dragging and modifying existing regions.

Zoom 

The **Zoom** button with the **magnifying glass** symbol allows zooming into and out of the data. When ***Zoom mode*** is enabled, clicking into the histogram zooms in and clicking while the [SHIFT] key is pressed zooms out.

Pan 

The **Pan** button with the **hand** symbol allows shifting the current view port of the histogram.

Region 

The **Region** tools are used to draw polygons into a 2-D histogram and to set a range in 1-D histograms. Regions define bi-dimensional intervals within the parameter range and thus subpopulations of data points. Double-click in order to close a region in a 2-D histogram.

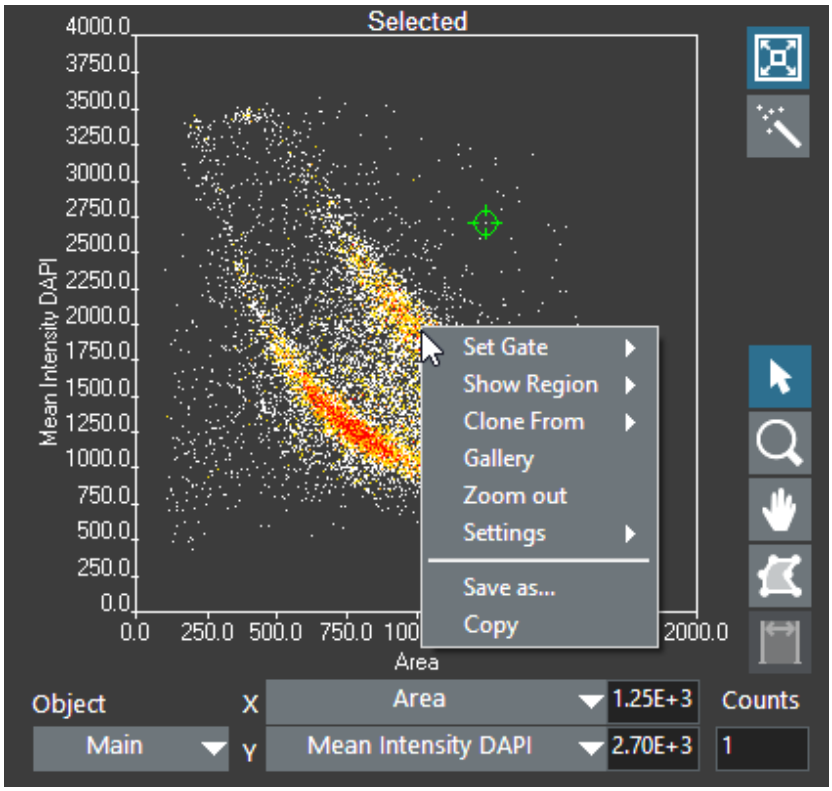
Auto Scaling 

The **Auto Scaling** button re-scales the histogram for optimal visibility. As long as it is enabled, the histogram axes will adapt to the data range and re-scale the axes if necessary when new data points are added.

Auto Regions 

The **Auto Region** tool with the **magic wand** symbol is used to find and draw regions automatically. The regions are detected depending on the current zooming level to identify subpopulations of interest.

2.4.1 The histogram context menu

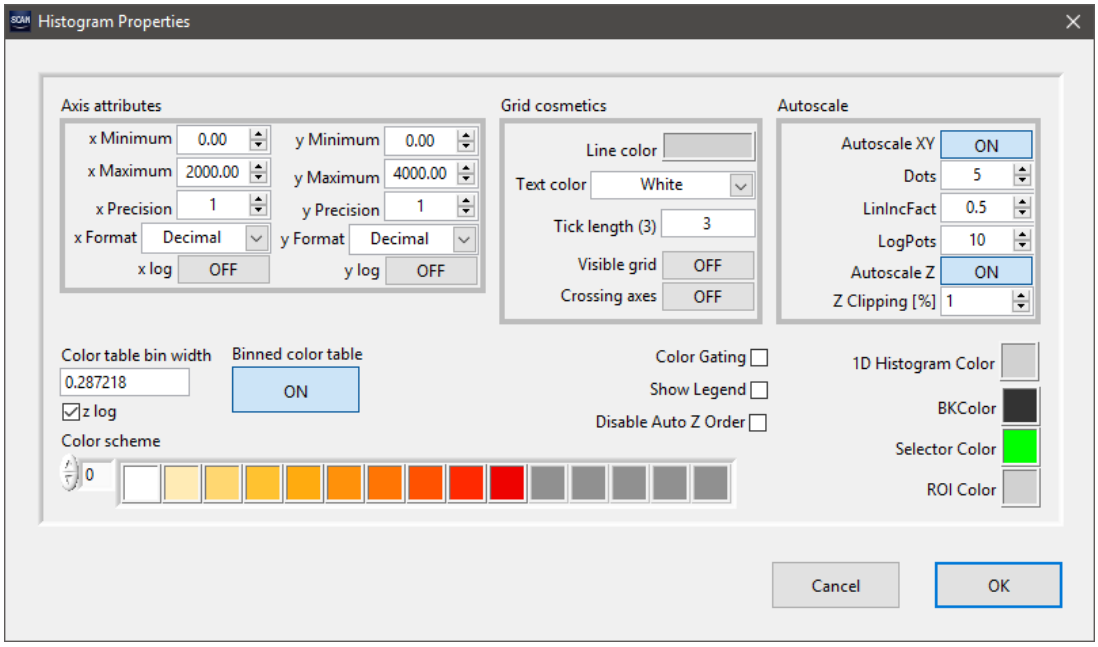


The histogram can be managed through the context menu accessible via right-click into the histogram. (Note that you will get the **region** context menu, if you right-click on a region border as described below). The **histogram** context menu contains the following commands:

- **Set Gate**. Apply a gate on the selected histogram by choosing a region from the list that appears.
- **Show region**. Select a region from the list that appears to generate a zoomed-in view of it with the X/Y parameters that were used to define this region.
- **Clone from**. Select a histogram from the list that appears to duplicate the histogram.
- **Gallery**. This command generates an image gallery of objects in the current histogram. The number of images and the selection criteria are set according to the gallery preferences given in the **Preferences** dialog.
- **Zoom out**. This command zooms out of the current view by a factor of 2 (in case of linear axis scaling).
- **Settings**. This menu item opens a new dialog with options to customize the histogram appearance.
- **Settings ▸ Autoscale XY**. This command activates automatic scaling of the X and Y axes. Auto scaling will remain active until it is deactivated again or the histogram is zoomed manually.
- **Settings ▸ Autoscale Z**. This command activates automatic scaling of the Z axis, i.e. the “height” of the histogram bins. Auto scaling will remain active until it is deactivated again.
- **Settings ▸ Color Gating**. This command causes the population of each gate to be displayed in a different color in the histogram.
- **Settings ▸ Show Legend**. This command toggles display of the color gating legend in the histogram.

- **Settings ▸ Theme** Select a theme for visualization of the histogram data and general style of the histogram.
- **Settings ▸ Advanced…** This command opens the **Histogram Properties** dialog with expert settings described below.
- **Save as…** This command enables saving the current view of the histogram into an image file
- **Copy...** This command copies the current view of the histogram to the clipboard.

2.4.2 The histogram properties dialog

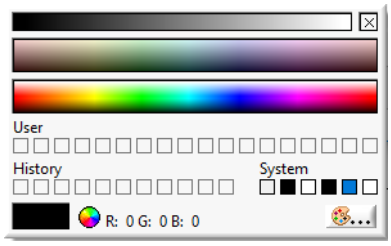


Axis attributes. This field offers different choices to set the axis scaling.

Grid cosmetics. This field offers different choices to set the grid display.

Autoscale. This field offers different choices to set the auto scale of the axes.

BKColor. A click on the colored field opens a window that allows selecting the background color.



Selector Color. A click on the colored field opens a window that allows selecting the cross-hair color.

Region Color: A click on the colored field opens a window that allows selecting the color of the region outline.

Color scheme. This color palette defines the coloring of data points (pixels) in the histogram in dependence of the number of counts (events) they represent. For example, if the **Color table bin width** is set to three and a data point represents 7, 8 or 9 counts, it will be displayed in the third color from the left.

Color table bin width. It defines the range of counts to be given the same color from the **Color scheme**.

Binned color table. Click this button to activate the color binning as set in **Color table bin width**.

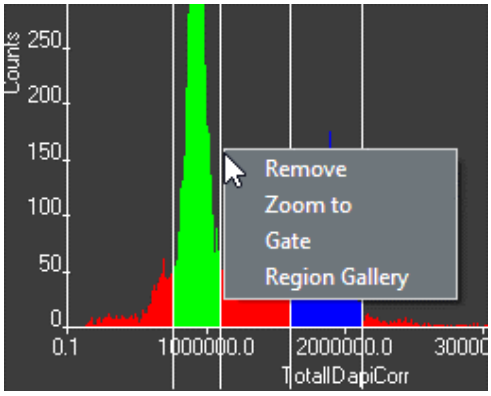
Color Gating. This command causes the population of each gate to be displayed in a different color in the histogram.

Show Legend. This command causes a legend of the colors in the **Color Gating** mode to be displayed in the histogram.

Disable Auto Z Order. Disables the automatic determination of z order for 1D histograms.

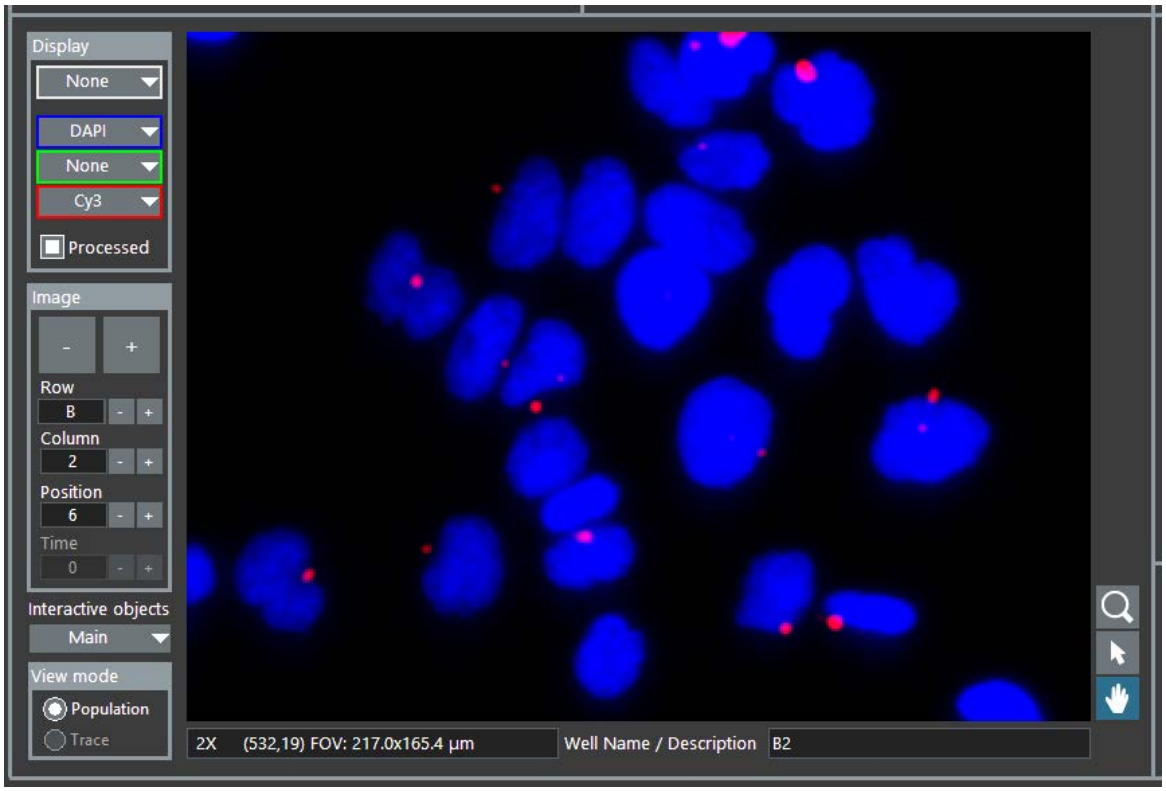
2.4.3 The region context menu

The region context menu will open, if you right-click on a region border



- **Remove.** Deletes the selected region or gate.
- **Zoom to**. Gives a zoomed view of the selected region.
- **Gate**. Creates a new ***AND Gate*** defined by ***the region*** and the ***Gate currently applied*** to the histogram. (See also Chapter 4.2.2, *The gate manager.*) When a gate is applied to a histogram, only the data points within this gate are displayed. To display all data points open the histogram context menu and go to **Set gate ▸ none**.
- **Region Gallery**. This command generates an image gallery of objects in the selected region. The number of images and the selection criteria is set according to the gallery preferences given in the **Preferences** dialog. (For more information see Chapter 2.2 *Preferences*).

2.5 Using the Image Viewer



Each analysis data point is directly linked to the image object it is generated from. These objects can be displayed in a close-up view in the **Image Viewer**. Browsing the data set by clicking on data points with the histograms **Navigation** tool leads to a corresponding update of the image in the viewer. Additionally, the **Row**/**Column**, **Position, Time** fields in the **Image** selection box and the **Well/Name Description** fields are updated and give information about the image origin and other context. The **+/-** buttons next to the **Row**/**Column**, **Position** and **Time** fields can likewise be used to navigate. Values can be typed in to select images of a particular well, position or time step. The larger **+/-** buttons in the **Image** selection box can be used to switch to the next available image or browse through all available images by holding the button pressed.

Display. Multi-color images consist of individual color channels that were recorded with different optical acquisition settings (e.g. different excitation filters). The **red**, **green** and **blue** drop-down lists serve to select the input for the three color channels that are displayed in the respective colors. In order to display a channel (e.g., a transmission channel) in grayscale, select it from the gray drop-down list. When having both grayscale and any color selection active, a combined multi-color image with transmission overlay will be displayed.

The clipping of the RGB display, i.e., the scaling of the image display brightness, can be changed via the menu item **View ▸ Adjust Display…** that opens the **Adjust Display** window.

Display: Processed. Click this button to toggle between the original image and the processed image as defined in **Analysis ▸ Assay Settings** in the **Image Processing** tab. Image processing might improve the quality of the displayed image but slows down the systems image display. Therefore it is especially recommended to switch it off for performance reasons when creating galleries. The image processing is described in Chapter 3.8, *Image processing.*

Image: Row/Column/Position/Time. Use these entries to select a specific image to be displayed. The **+/-** buttons allow fast navigation through your image data set by incrementing or decrementing the **Time**, **Position** and/or **Well** number.

Interactive objects. Select the object type (the **main object** or any sub-object) to be outlined and selected in the display when clicking on an object or data point.

The **Image Viewer** is equipped with a tool bar to select different mouse tools:

Zoom 

The ***Zoom*** mode is used to zoom into the displayed image. A click into the image causes a zoomed-in view with the cursor position as the center. To zoom out, the **Shift** key must be pressed simultaneously.

Selection 

The ***Selection*** mode allows selecting individual objects within the image via mouse click. The object type to be displayed can be selected in the drop-down list in the bottom left corner of the image viewer. Main objects are highlighted by a green outline. The data point corresponding to the selected object is highlighted with a red circle 2-D histograms and a vertical red line in 1-D histograms.

Move 

Depending on the zoom factor, only a part of the image will fit into the display. The ***Move*** mode allows moving the visible area via mouse-drag.

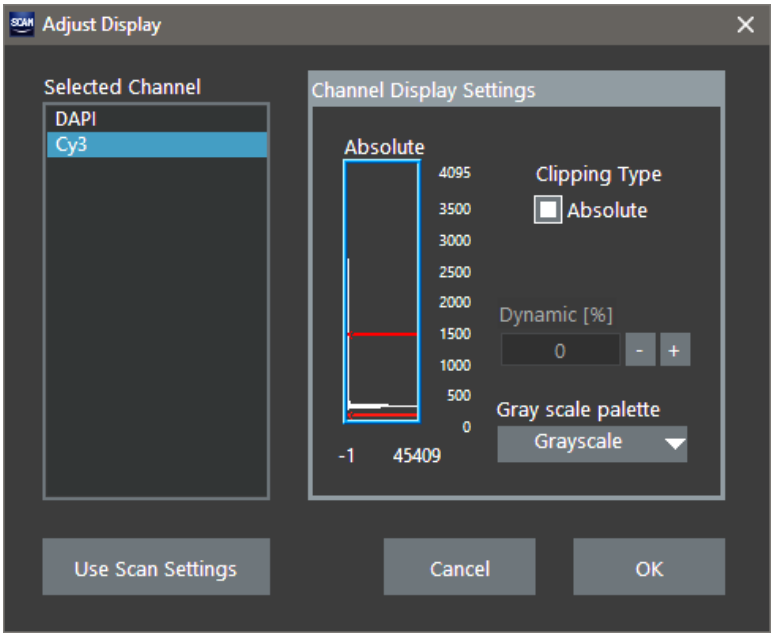
The **status bar** in the lower left corner shows information about the magnification of the displayed image, the current x/y position of the cursor and the pixel value(s) at this position. Furthermore the status bar shows the dimensions of the current field of view (i.e. the currently displayed part of the image) in micrometers as calculated from the scan settings.

2.6 Adjusting the image displays

View ▸ Adjust Display opens the **Adjust Display** window that allows adjusting the display brightness. Note that these image settings affect the front panel display as well as well overviews and galleries.

A raw image will always have a certain background intensity. Also, one will avoid to over saturate images and – especially in fluorescence applications – rather use only a fraction of the dynamic range of the camera. The consequence is that a raw image appears usually low in contrast and may even appear entirely black. ***Clipping*** is applied to change the image brightness by defining a fraction of low pixel intensities to be displayed black as well as a fraction of high pixel intensities to be displayed with maximum brightness.

Clipping Type. This checkbox toggles between **Dynamic [%]** and **Absolute** clipping.



Dynamic [%]. Define here how many pixels (as a percentage of the total number of pixels) will be displayed with maximum and minimum brightness. The intensity of the remaining range of pixels will then be scaled linearly in between. The higher the numbers the stronger the resulting contrasts.

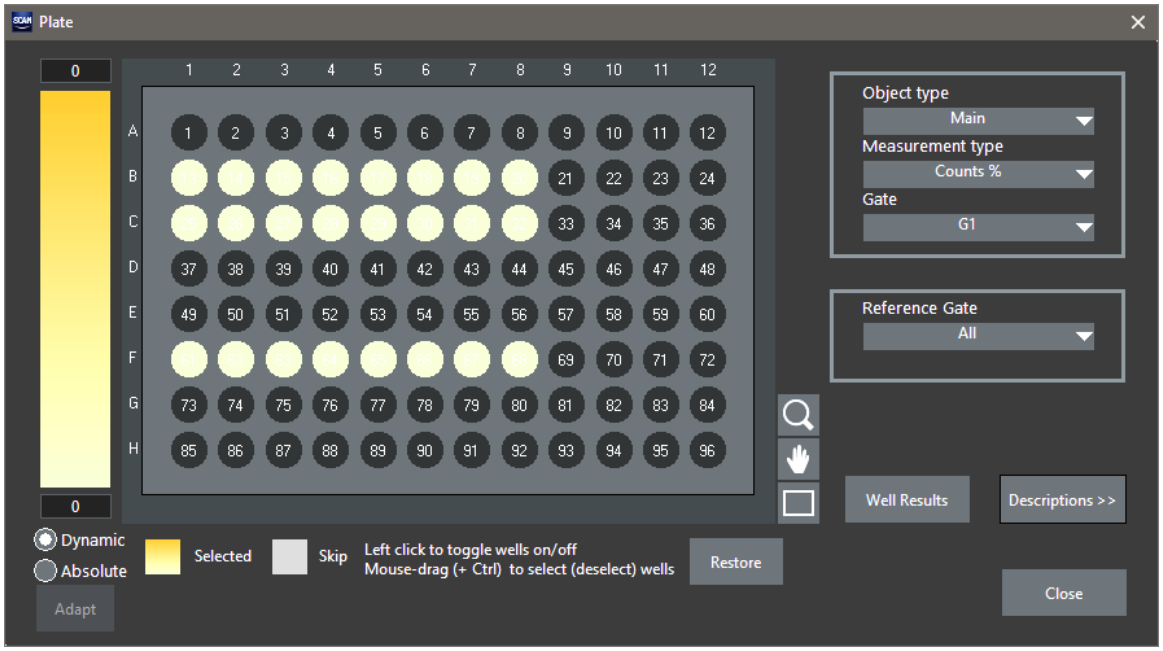
Absolute. Define here the range of pixel counts to be displayed with maximum and minimum brightness by dragging the red horizontal lines – that represent the maximum and minimum thresholds – with the mouse. The intensity of the remaining range of pixels will then be scaled linearly in between.

Gray scale palette. The channel selected in the gray drop-down list in the main GUI can be displayed in different false-color palettes that can be selected here.

Use scan settings. Apply the display settings used for the scan.

2.7 Interactive plate result view

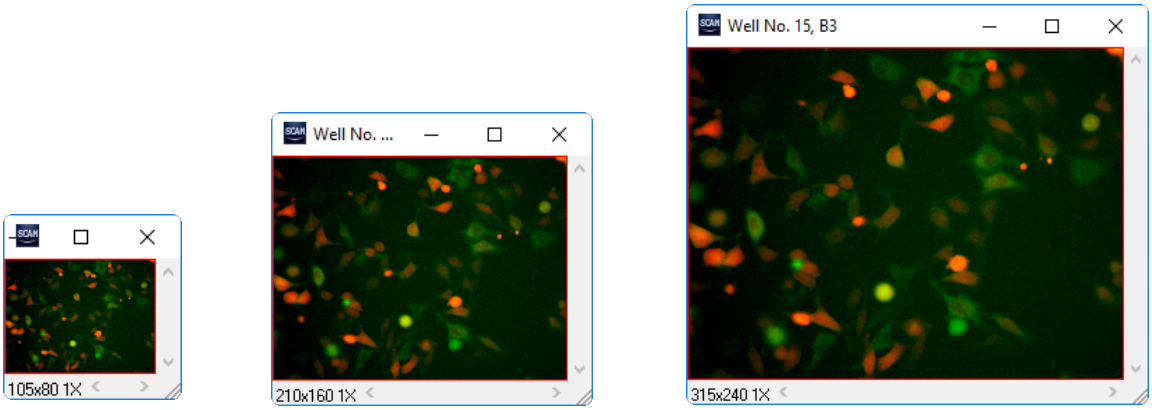
The results of all measurements can be displayed interactively in the **Plate** window. To open this window, go to **Scan ▸ Plate…** In the center of the window you will find a graphical representation of the screened plate. When the window is opened for the first time, the wells that were skipped during acquisition are shown in gray; the scanned wells show a different color.



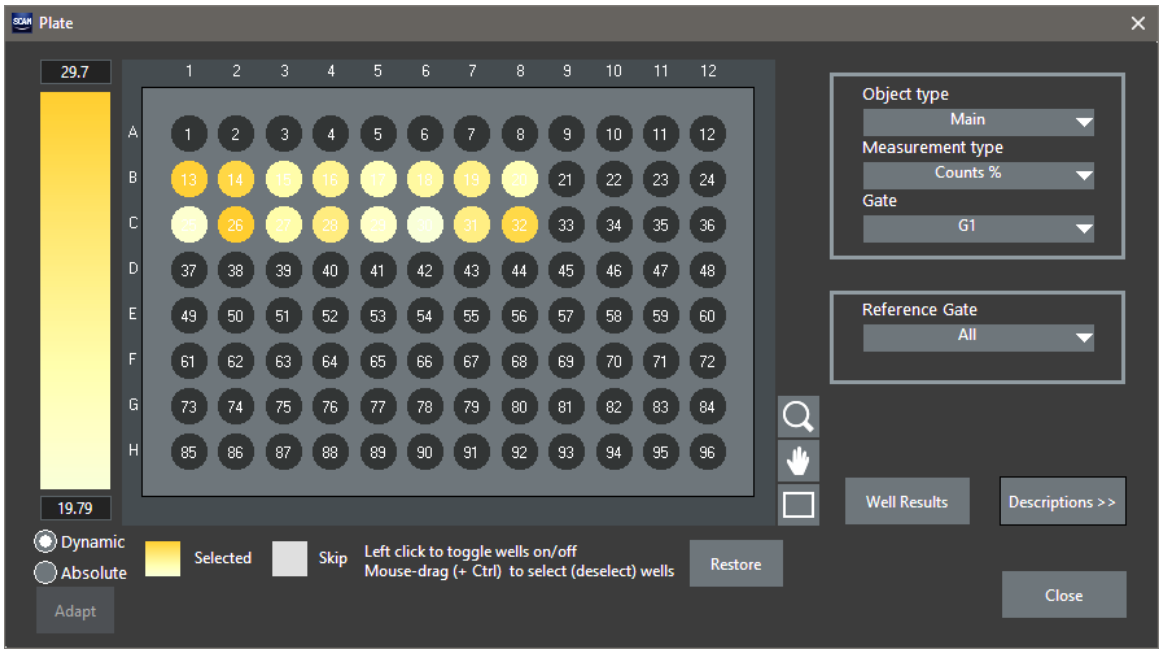
By default all screened wells will be taken into account for analysis. Adjusting the well selection is done similar to the well selection in the ***scanR Acquisition*** software, i.e. to change the selection you can click on the wells. To deselect multiple wells you can press Ctrl on the keyboard and draw a rectangle with the mouse around the wells to be excluded. When a well is deselected, the corresponding data points will be removed from all histograms. To include a deselected well again, click on the well. It will again be displayed in a color other than gray and the corresponding data points will be shown in the histograms.

The **Restore** button will restore the initial well pattern.

A **well overview** of each well can be shown by right-clicking on one of the wells and selecting **Well Overview**. To change the overview size of an image, go to **Preferences**. The size can be set in the **Overview Size** drop-down list. It can be useful to decrease the resolution of the well overview for large overviews in order to increase the speed of the display.



To display a heat map of the measurement results you have to run the analysis first and then proceed as follows: Select the object type (main object or sub-object) from the first drop-down list (**Object type**) on the right. The drop-down list **Measurement type** allows you to switch between **Counts**, **Counts%**, **Mean**, **Error**, **Error%** and **CV%.**



When **Counts** is selected as **Measurement type**, you can select the population you are interested in in the **Gate** drop-down list. Here all defined gates can be selected. You can select the population, you want to use as reference in the **Reference Gate** drop-down list. Here also all gates that you have created in the histograms can be selected. The wells will now be displayed in a color that indicates how many cells in every well are found in the selected gate. When a well is displayed in orange, many cells of this well can be found in the selected gate, whereas a green color indicates, that only few cells can be found in the selected gate.

The same applies when **Counts%** is selected as **Measurement type**, in this case however, the color will indicate the percentage of cells that are in the selected gate.

When **Mean**, **Error**, **Error%** or **CV%** are selected as **Measurement type**, again the **Gate** drop-down list contains all gates that you have defined. However, the next drop-down list now allows you to select the **Measurement** to be performed. Here all parameters that you have previously defined in **Edit Assay ▸ Measurement Parameters** (see Chapter 3.6 *Measurement parameters*) and in **Derived Parameters** (see Chapter 0

Derived parameters) can be selected from the drop-down list. For example, if the parameter **Area** is selected as **Measurement** and **Mean** is selected as **Measurement type**, the ***mean area*** of all cells per well will be color encoded. This means that wells with very large cells will be displayed orange, whereas wells with smaller cells will be displayed green.

> The **Plate** view is fully interactive and updated automatically. When a gate is modified in the front panel, the results will automatically update in the **Plate** view.

Next to the color gradient bar indicating the display range, you can set how the range of the color display will be adapted when the same assay is run on a new data set. The display range can then be either adapted in **Dynamic** mode, which means the **min** and **max** values of the display range will be adapted according to the new data set. Alternatively, the range can be kept in **Absolute** mode, which means the range that you have set in the first place will be used for all other analyses.

In order to change the display range manually, you can directly enter **min** and **max** values at the bottom and the top of the color bar, respectively. When you change the range manually, the display range mode

will automatically change from **Dynamic** to **Absolute**. The **Adapt** button can then be used to adapt the **min** and **max** ranges of the color display to the current data set and when a new experiment is analyzed, also this display range will be used. Switching back to **Dynamic** will also restore the min and max values of the display, but when a new experiment is analyzed, the min and max ranges will be adapted to the results of the new dataset.

A click on the **Descriptions** button opens the **Name/Description** list. This list displays all selected wells. By default, the wells will be named A1, A2 … etc. The names of the wells can already be changed to a customized and more descriptive name when setting up the acquisition (see scanR Automated Image Acquisition Software, Chapter 4.1.1 *Well pattern*). Alternatively, the names can be changed now in the **Plate** window. Click in the **Name/Description** field you want to change and enter a new name.

Note: when the same name is used for multiple wells, these wells can be ***grouped***. In the **Measurement Results** and **Populations** tabs of the **Well Results** window (see Chapter 4.4 *Well results*) you can then switch between **Wells** and **Groups** to have each well listed individually or to show the results of the created groups.

A second click on **Descriptions** hides the **Name/Description** list.

3 Assays

Assays are the protocols that define how to extract the data of interest from the images of a scan. They define which objects are to be recognized and how and which measurements are to be performed on the found objects. This chapter explains in detail how assays are to be set up or modified.

3.1 General

An assay defines all steps necessary to extract quantitative data from the acquired images. It usually starts with some sort of image processing like background correction. Secondly, objects have to be detected in the images. The analysis, for example different kinds of measurements like area, intensity, shape…, is finally performed on these objects and results for the samples (e.g. wells) are generated.

The **Analysis ▸ Assay Settings…** command opens the **Assay Settings** window and allows applying or adapting the assay to the loaded scan. The **Assay Settings** window contains seven tabs: **Main Object**, **Sub-objects**, **Parameters**, **Derived Parameters**, **Image Processing**, **Virtual Channels** and **Image Export**. The tabs are arranged from left to right, but you can always jump back and adjust the settings of former steps. However, background correction (in the **Image Processing** tab) should be performed prior to object detection as it changes the intensity values.

scan^R distinguishes between two kinds of object types: any assay always defines one **Main Object** type and up to four **Sub-object** types connected to it. To give an example, main objects may be individual cells while their sub-objects are individual structures within them.

To represent this hierarchical structure, the **Assay Settings** tabs **Main Object** and **Sub-objects** are used to set the search algorithms for main and sub-objects in order to extract the structures of interest from the images. This is done by different ***Object Finder*** **Modules** that implement different rules for object detection.

The **Parameters** and **Derived Parameters** tabs contain the definitions of the kind of information to be extracted from the objects (e.g. area, shape,…) .

The **Image Processing** tab allows defining image processing steps that are to be executed before the object detection and parameter extraction.

In the **Virtual Channels** tab new channels can be created as a result of post-acquisition image processing (e.g. spectral unmixing).

The **Image Export** tab allows selecting one or more color channels. The processed images of the selected color channels will be exported once available when the analysis is running. Color channels with time-consuming processing steps can be added to the list automatically by enabling the **Processed Image Buffering** option. The exported, processed images will be used as a buffer to avoid re-running image processing.

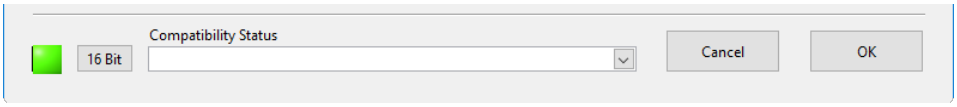
3.2 Assay bit depth

All image processing operations, measurements and threshold definitions can be done with 12 bit or 16 bit precision. While there are still scientific cameras available which only have a dynamic range of 12 bit,

16-bit cameras are the standard for some time. All new assays in scan^{R} will automatically be 16-bit assays and make use of the full precision of modern camera raw data.

For backward compatibility reasons, 12-bit assays can still be used without functional limitations. If they are used with 16 bit image raw data, only the most significant 12 bits of the image data are utilized. This means the 4 least significant bits which mostly contain noise and not signal are ignored but it is possible that precision is not sufficient for analysis of very low signal data.

Existing 12-bit assays can be up-converted to 16-bit assays by clicking the **16 bit** button in the **Assay Settings** dialog which is only available for 12-bit assays.



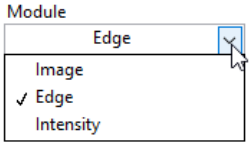
If a 16-bit assay is used with 12 bit raw data, all intensity values are converted on-the-fly to 16 bit for analysis without loss in precision. Original data (image data in a scan) is never modified.

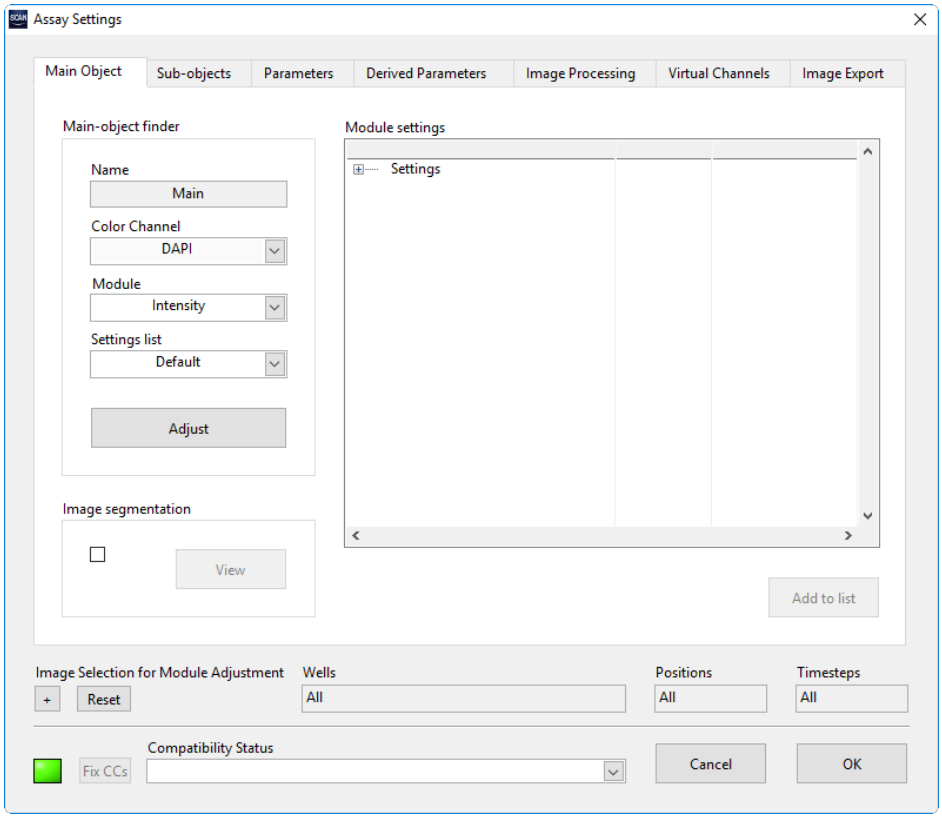
3.3 Object finder: detecting main objects

The **Main Object** tab of the **Assay Settings** window provides the commands to define the **Main Object** detection.

Color Channel. Select the color channel on which the main object detection is to be performed.

Module. Select the method to detect individual **Main Objects** from the drop-down list.





Settings list. Each ***Object Finder* Module** has a list of preset parameters. The lists can be modified and stored at will. Individual modifications of these settings are marked as **Modified**. Select the settings of choice from the drop down list.

Adjust. This command opens the configuration dialog of the selected Object Finder module. (For changing the list of Object Finder modules see Chapter 3.5, *Object finder modules)*

Module settings. This field lists the current settings of the selected Object Finder module.

Add to list. This adds the modified settings to the **Settings list**. Click the button to open the **Add Settings to OFL** window (**O**bject **F**inders **L**ibrary, see Chapter 7.4.2, *Object finders library (OFL)*) where you can provide a **New Settings Name** for the modified settings list.

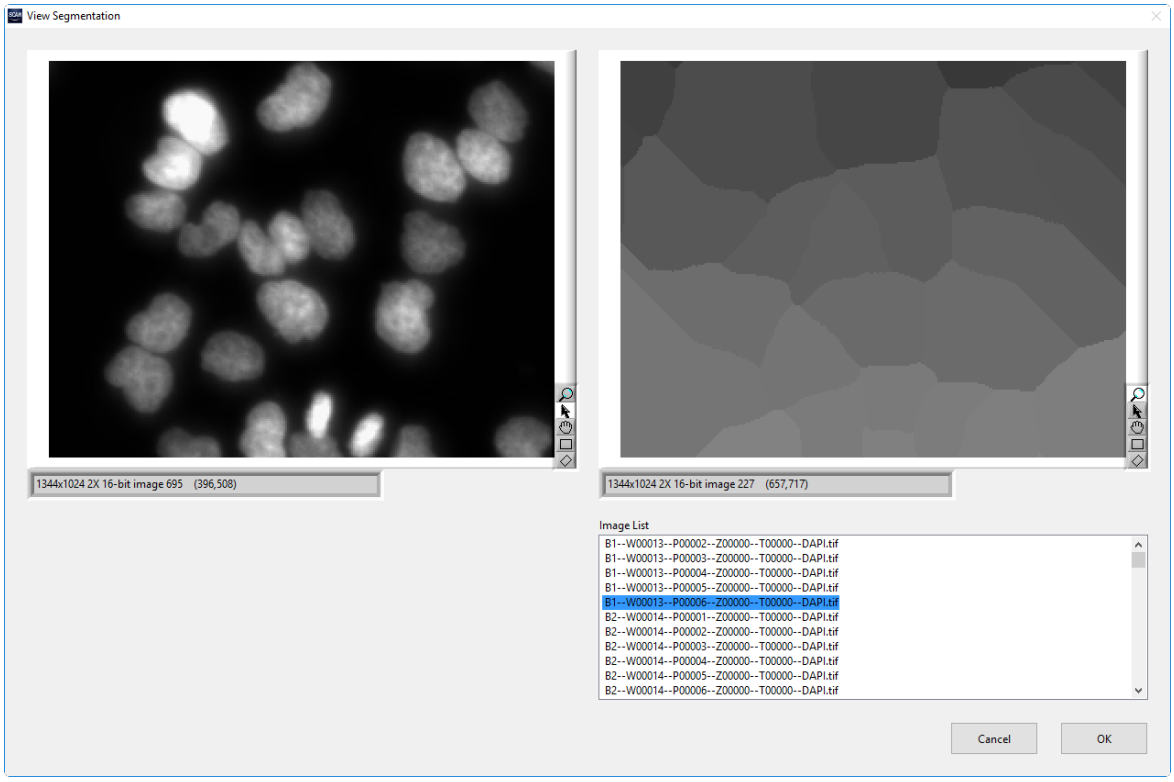
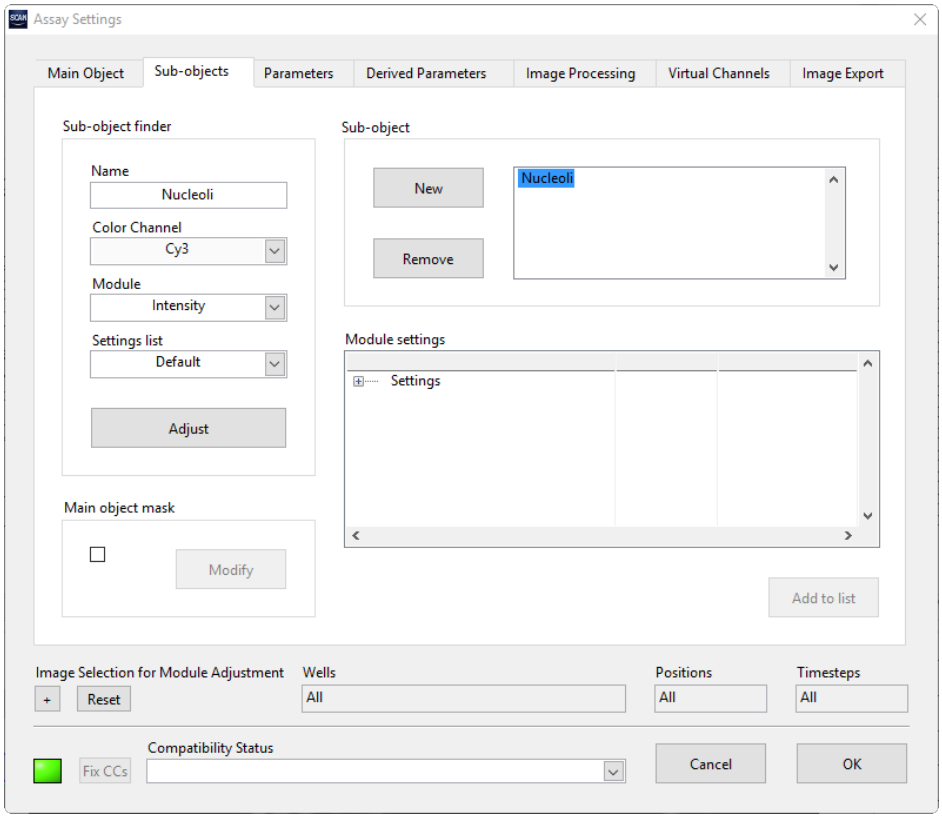


Image segmentation. This function divides – if activated via the check box – the entire image into segments: as many segments as there are **Main Objects** where each segment is assigned to the **Main Object** in its center. In other words, each image pixel is assigned to the **Main Object** it is closest to. All pixels that are assigned to the same **Main Object** form one segment of irregular shape and size. The **View** button opens the **View Segmentation** window that contains on the left a display of the image containing the objects and on the right a display of the segments.

3.4 Sub-object finder: detecting sub-objects

Sub-objects are structures that are directly linked to individual **Main Objects**. The search for **Sub-objects** takes place on an image mask derived from the corresponding **Main Object**. This **Main object mask** can be adapted for each **Sub-object** type separately.



Sub-object finder: Color Channel, **Module**, **Settings list**, **Adjust**, **Add to list**. These functions are analogous to the ones described in the previous Chapter 3.3, *Object finder: detecting main objects*.

Name. Give a name to each new **Sub object**. The default name is ***Subobject 1***.

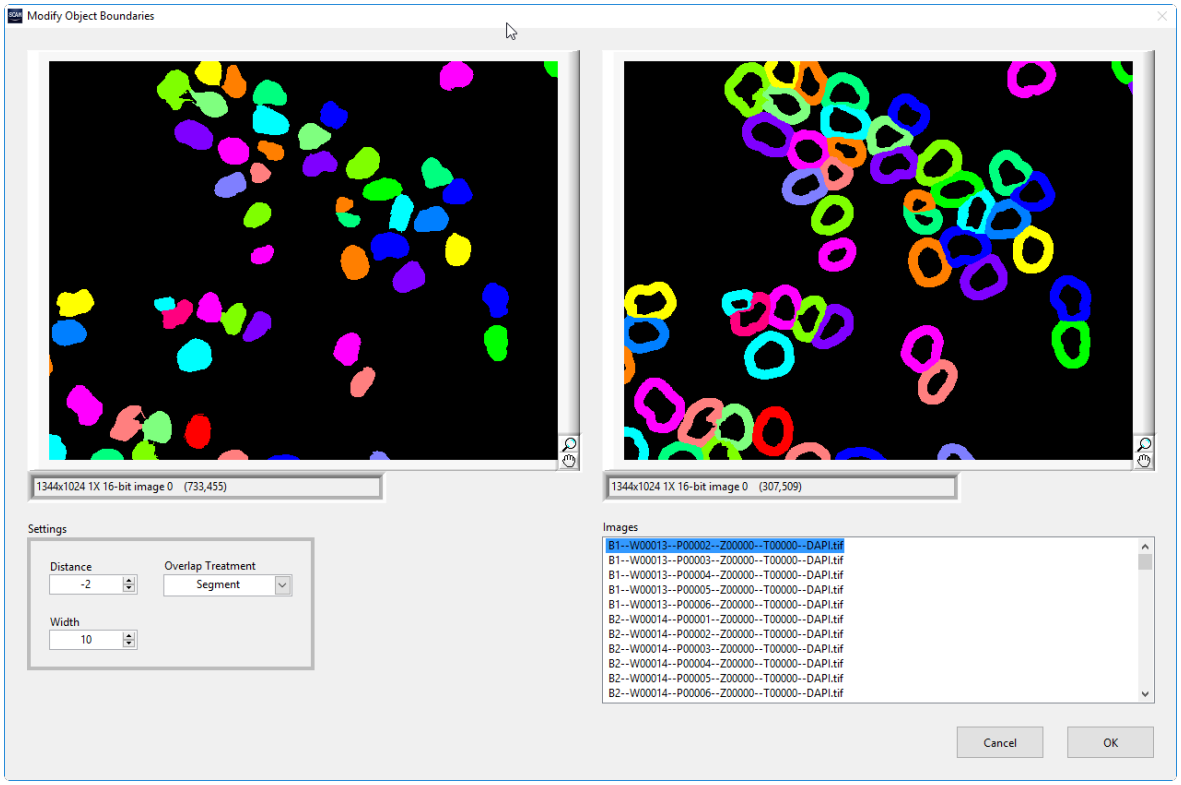
Sub-object list. It gives an overview of the defined **Sub-object** types. The **New** and **Remove** buttons allow the insertion and deletion of **Sub-object** types.

Main object mask. Each individual main object found in an image creates a mask. The individual sub-objects are associated with this mask rather than with the main object itself. Imagine, a main object is the cell nucleus and the sub-objects are structures outside of it. In order to be detected, the original main object mask – which only covers the area on the nucleus – needs to be modified in order to enable the detection of the sub-objects.

Click the checkbox to enable the image mask modification of the main object.

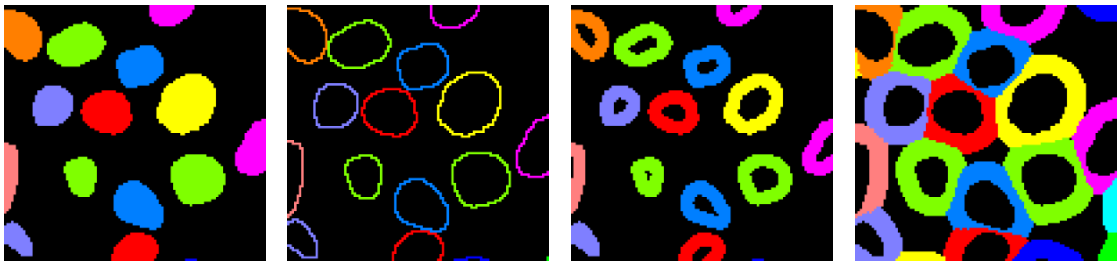
Modify button. Click here to open the **Modify Object Boundaries** dialog to adapt the main object mask to the needs of the sub-objects detection.

Distance. The distance is measured in pixels from the outer rim of the main object mask (positive and negative values are valid)



Width. Extension of the sub-object mask in pixels.

The examples below illustrate the effects of these parameters.



Main object mask | Distance 0, Width 1 | Distance -5, Width 5 | Distance 8, Width 8

Overlap treatment. Options are **Segment**, **Segment (slow)** and **remove.**

> When sub-objects are used for analysis, a further parameter becomes available in the **Parameter** tab (see Chapter 3.6, *Measurement parameters*): **Subobject 1 counts** (if the default name for sub-objects is used, otherwise it would be ***subobjectsname* counts**). This parameter gives the number of sub-objects detected for each main object and is a parameter of the main object.

3.5 Object finder modules

3.5.1 Entire image

This is a very simple object definition: the entire **Image** is used as object. You may use this for measurements of integral intensities of your sample, i.e., for each image and each parameter (see. Chapter 3.6, *Measurement parameters*) a single value is calculated, independent of the objects within the image.

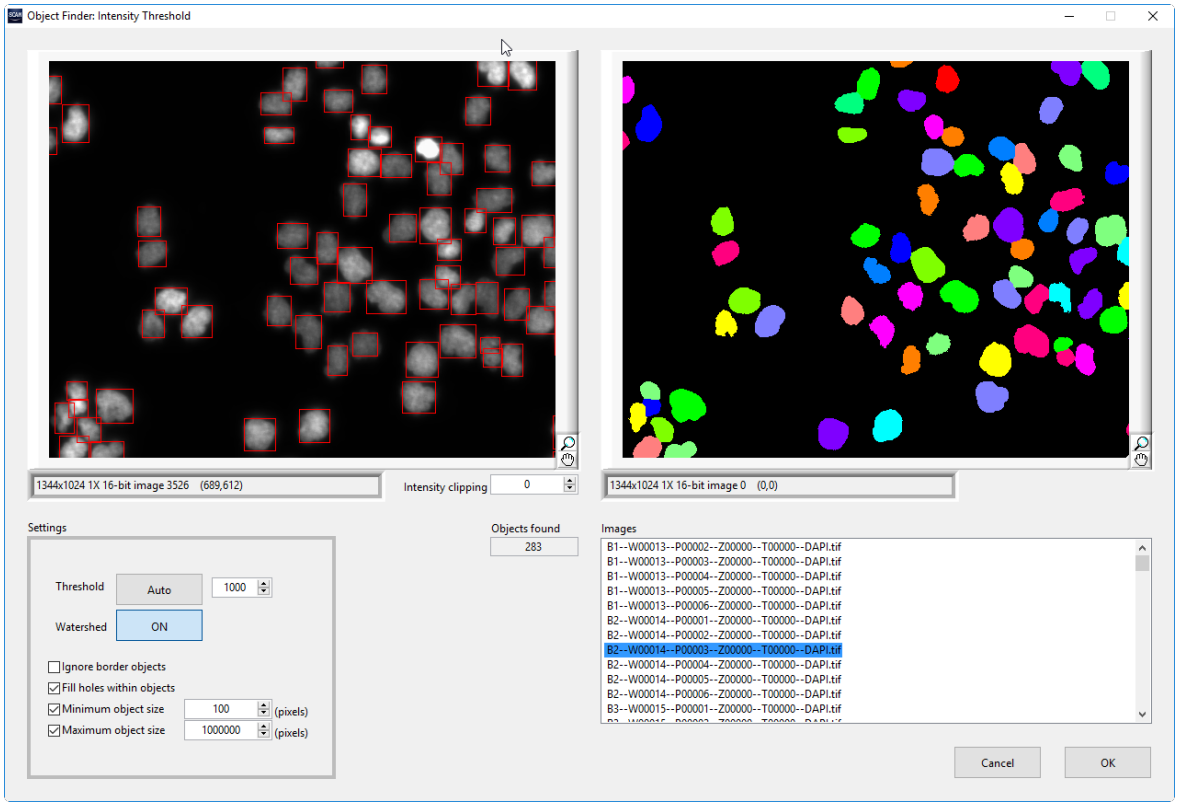
Ignore frame. This is the only parameter to adjust: the size of a bordering frame to be ignored. The default value is 0 (no bordering frame).

When used as object finder module for sub-objects, the **Image** object finder will deliver the entire mask of the main object or the modified mask.

3.5.2 Intensity threshold

As the name says, the **Intensity** threshold method is based on intensity values: pixels with intensities above a predefined threshold will be united to one individual object.

The **Object Finder: Intensity Threshold** dialog has two image viewer displays. The left one shows the gray value image including all detected objects marked with a red bounding box. The right one shows the mask of each object, visualized by different colors for separate objects and black for background.



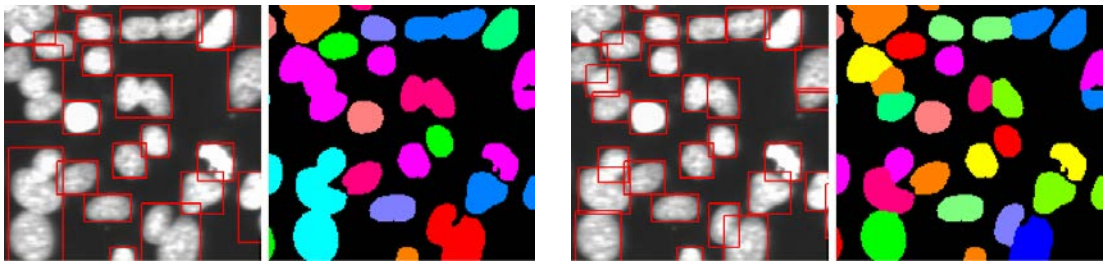
Settings

The settings used when opening the window depend on the **Settings list** selected in the **Main Object** or **Sub-object** tab (see Chapters 3.3 , *Object finder: detecting main objects* and 3.4, *Sub-object finder: detecting sub-objects*) and are loaded from the **Object Finders Library** (see Chapter 7.4.2, *Object finders library (OFL)*).

Threshold. This is the intensity cut-off for objects. Type in a value or use the arrows to adjust it.

Threshold: Auto. Click this button to automatically evaluate the image background and set a meaningful cut-off value.

Watershed. If neighboring objects are so close together that ***thresholding*** does not lead to a clear separation, they will be detected as one object. (See left image pair below.) The watershed algorithm separates these objects along the contractions of the detected masks. (See right image pair below.) Set the toggle button to **ON** to use this option.



Ignore border object. Check this box to ignore all objects that are cut-off by the image border.

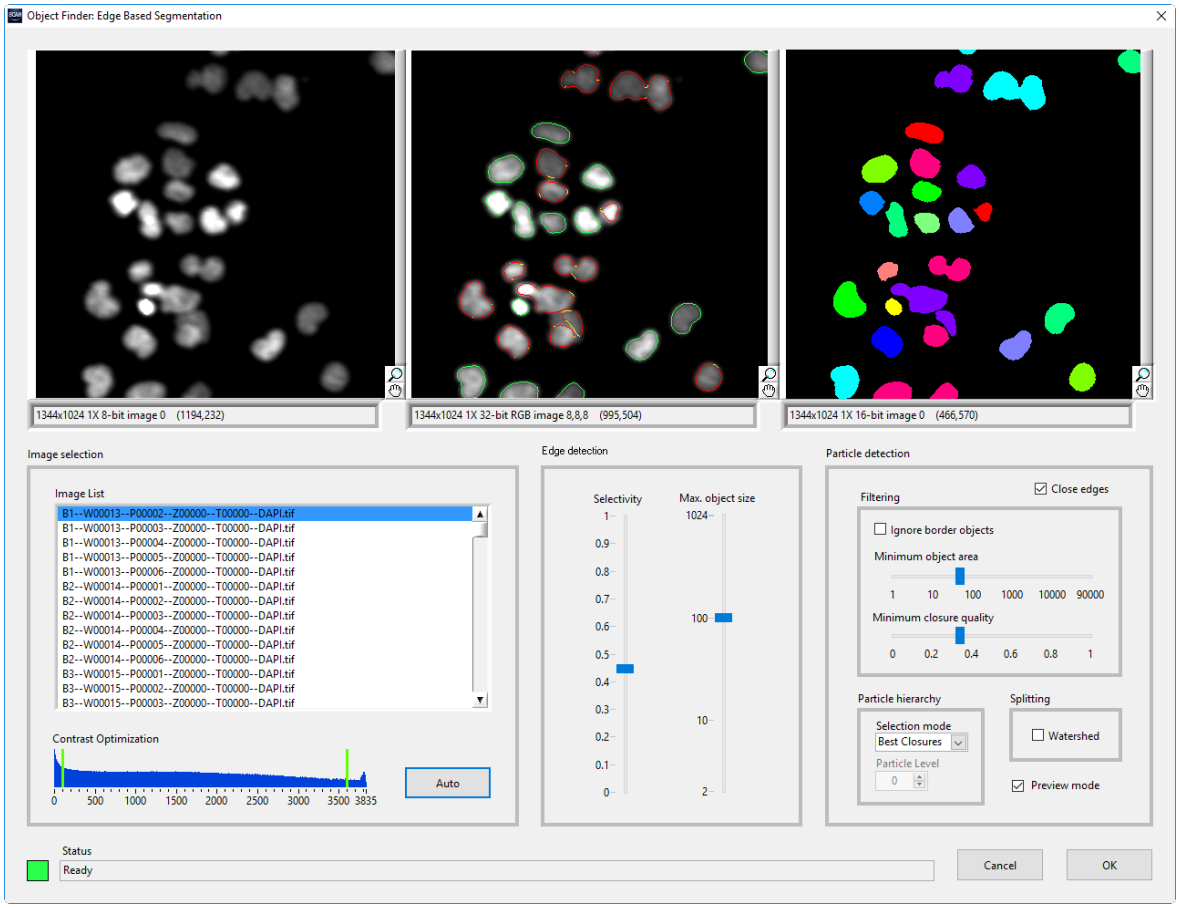
Fill holes within objects. Check this box to fill the object mask in case it contains holes.

Minimum/Maximum object size. Check these boxes and adjust the values to apply minimum and maximum size filters to the objects (in order to ignore objects that are outside these size limits).

3.5.3 Edge detection

The **Edge Based Segmentation** module is a general purpose edge based particle detector. The idea of the algorithm is to find a closed contour around each particle. First the edges of the image are extracted. For those edges which already form a closed contour, the algorithm is finished. Since the remaining open edges may be part of a closed contour around a particle, the algorithm then tries to combine these open edges so that they form a closed contour as well.

The edge detection algorithm yields better results when objects of ***strongly varying intensity*** have to be detected. In these cases the threshold detection will either lead to clusters when the threshold is set to a low value in order to detect also dim objects. If a higher value for the threshold is set, then the dim particles will be missed. Furthermore, as edge detection is intensity independent it is especially suitable for ***cell-cycle analysis***.

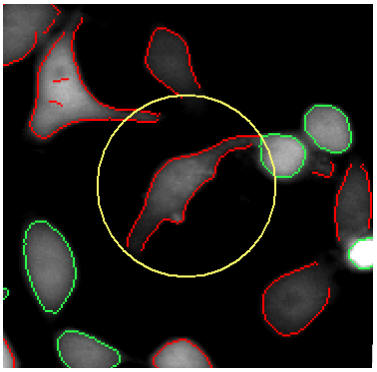


To reduce complexity, the process of finding the best settings is split up into three independent steps. The three groups of settings in the **Object Finder** window reflect these three steps. They are traversed from left to right, but you can always jump back and adjust the settings of former steps. In each step, the result of adjusting the current step's settings is shown in the image above.

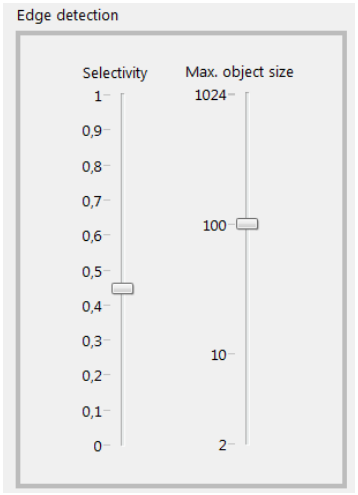
1. **Contrast Optimization.** On the left side, select an image of your choice from the list. Adjust the contrast either by pressing **Auto** or manually by moving the green bars in the histograms. If one or both bars are missing, just press **Auto** once. Try to clip away any unwanted noise or artifacts while maintaining good contrast between the particles you want to detect and the background.

---

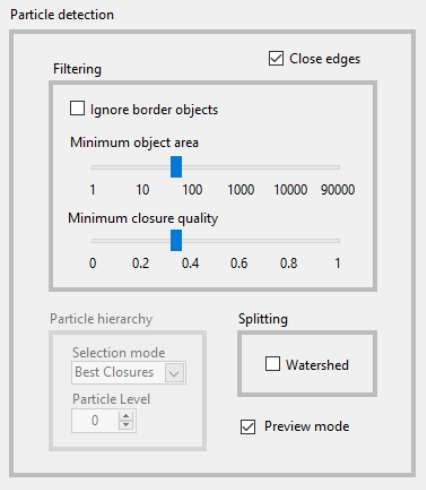
2. **Edge extraction.** Click on the image or the settings group in the middle to get to the ***second step***. In this step, the edges of the image are extracted. First grab the **Maximum object size** slider and adjust it so that the largest particles you want to detect just match inside the yellow circle appearing in the images.



3. Adjust the **Selectivity** slider to get just the strong edges (up) or also the weak edges (down). Try to increase the selectivity as much as possible, thereby removing edges due to noise and artifacts, without letting gaps in the contour of wanted particles get too large. As you can see in the image above, contours with closed edges are marked green while contours with open edges are red.



4. **Edge Closing.** Click on the image or the settings cluster on the right to get to the ***third step***. In this final step, the open edges extracted in step two are now closed by combining them with other open edges. You can then filter particles by size and closure quality, split them with the watershed algorithm or select a hierarchy.



---

5. First decide if there may have already been sufficient particles detected in step two (the green ones). If you think so you can skip the closing process by unchecking **Close edges**, thereby reducing processing time dramatically. All unclosed edges (red) are discarded then. In most cases, however, the loss of particles is too high with **Close edges** being unchecked. If **Close edges** is checked, you can see in the middle image that the open edges (red) are connected to a closed contour by yellow lines.

---

6. In the **Filtering** settings you can filter (i.e. exclude) particles by location, size and closure quality. The closure quality is a rating attributed to each detected particle, which describes the quality/reliability of the respective closure. Particles whose contour has already been closed in step 2 (the green ones) have closure quality 1. Especially when particle detection is difficult, you can at least filter out most of the wrongly detected particles by moving the **Minimum closure quality** slider towards 1.

---

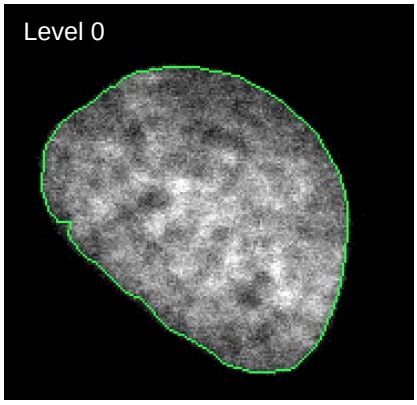
7. By checking the **Watershed** checkbox, you can split particles which have merged. The algorithm inspects the shape of each particle, splitting it at constrictions. This can be extremely useful when detecting nuclei.

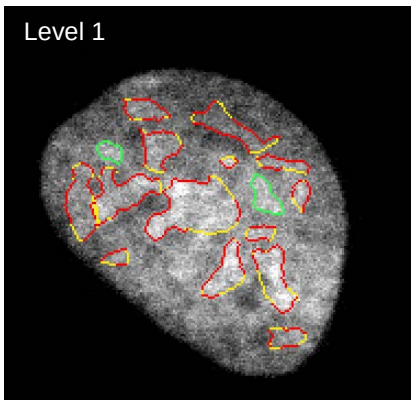
---

8. Sometimes detected particles are nested into each other. E.g. spots inside nuclei or nuclei inside the cytoplasm. Since overlapping particles are not allowed, the **Particle hierarchy** settings provide options to select the nesting or hierarchy level you are interested in. See below for the function of the **Selection mode** options.

---

9. Don’t forget to check other images of the scan to verify that your settings work with them as well. When you are satisfied with your settings click **OK** in order to complete the setting up of the image detection.





The options at the **Selection mode** dropdown list are:

Min level. Selects all level 0 particles.

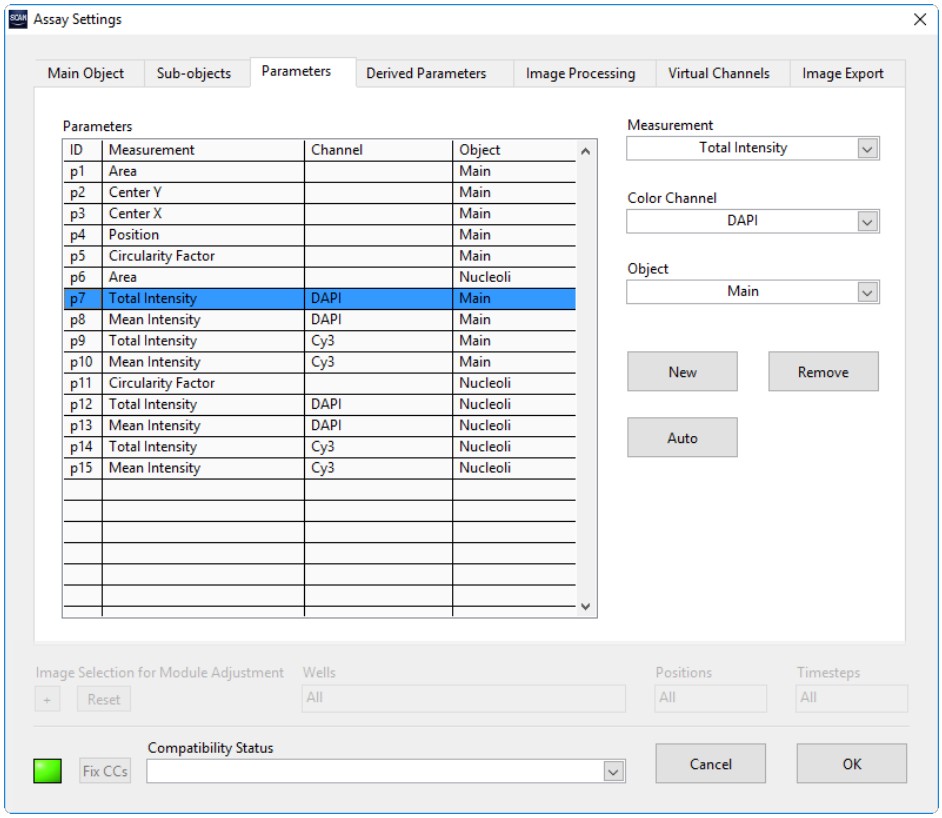
Max level. Selects all particles which do not have other particles nested inside.

Selected level. Selects all particles with the specified level.

Best closures. Selects for each nesting branch the particles which are best according to their closure quality.

3.6 Measurement parameters

The scanR **Object Analyzer** modules in the standard configuration offer a large number of individual parameters that may be measured for each recognized object. To limit memory utilization and save CPU time, only the values for the parameters listed in the **Assay Settings ▸ Parameters** list will be extracted during the analysis.



Parameters list. Each **Measurement** is labeled with an **ID (p1, p2,…)** and is assigned to the **Main Object** or a **Sub-object** and – depending on the type of measurement – may be assigned to a **Color channel**.

Measurement. Select a parameter from the drop-down list. The available parameters can be adapted through **Modules ▸ Object Analyzers** (Chapter 7.4.1).

Color channel. Assign a **Color** channel from the drop-down list to the newly added parameter.

Object. Assign an **Object** type from the drop-down list of available object types. The image mask of this object defines the parameter measurement area.

New. This allows inserting a new parameter into the list.

Remove. This deletes the selected parameter from the list.

Auto. Press this button to add a number of standard parameters for the main objects and all available sub objects to the list.

> A special parameter is available in the **Measurement** list when Sub-objects are used: **Subobject 1 counts** (if the default name for sub-objects is used, otherwise it would be ***subobjectsname*** **counts**). This parameter gives the number of sub-objects detected for each main object and is a parameter of the main object.

Measurements available by default:

- **Total Intensity.** Sum of the intensity values of all pixels of an object.
- **Mean intensity.** Average of the intensity values of all pixels of an object, i.e. the **Total Intensity** divided by the **Area**.

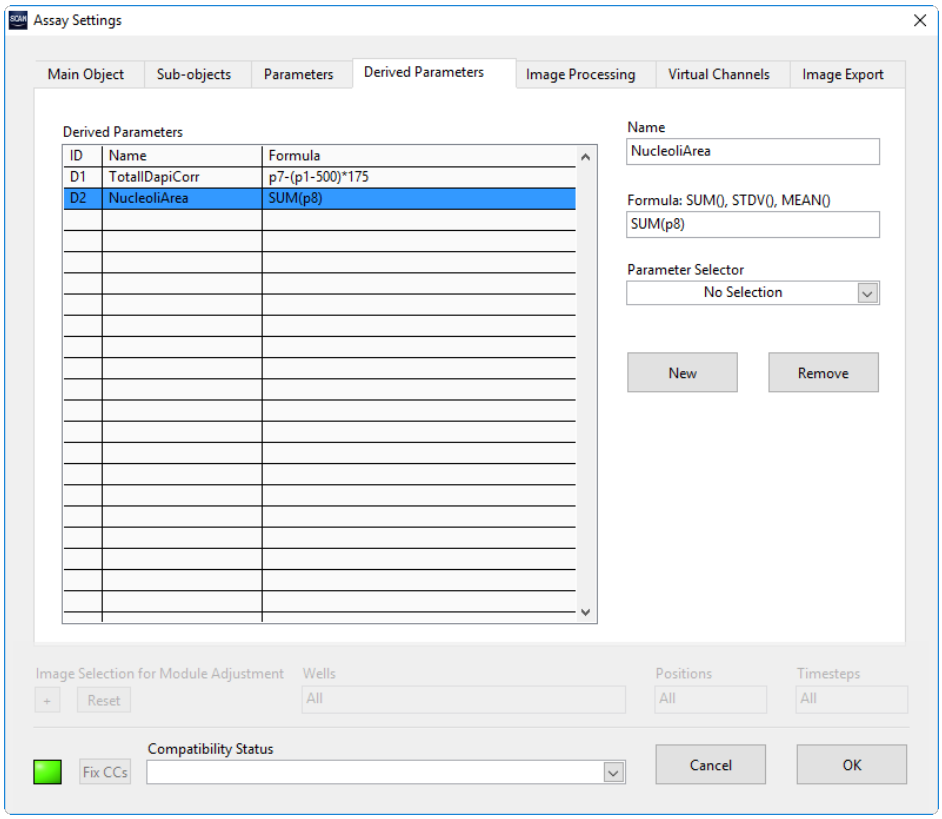
- **Max Intensity.** Intensity value of the pixel of an object which has the highest value.
- **Min Intensity**. Intensity value of the pixel of an object which has the lowest value.
- **Area**. The count of pixels of an object.
- **Circularity factor.** The **Perimeter** divided by the circumference of a circle which has an area equal to the area of the object (Heywood Circularity Factor).

$$\text{Circularity Factor} = \frac{P}{2\sqrt{\pi A}}$$

- **Perimeter.** Pixel length of the boundary of an object. Since the object is represented by rectangular pixels, the corners of the boundary pixels are subsampled to increase the accuracy of this parameter.
- **Max Feret Diameter.** The longest distance that can be measured between 2 points of the object perimeter.
- **Elongation Factor.** The ratio of the long and the short side of a rectangle with an area equal to the object area. The long side of the rectangle is defined by the **Max Feret Diameter.**
- **Most Frequent Class**. An advanced parameter used for neural network classification. Neural networks can output a new Virtual Channel where each pixel's values represent the classification result. The **Most Frequent Class** gives the dominant classification for the whole object when used on the Virtual Channel (See Chapter 6.5 *Standard classification workflow*)..
- **Entropy**. An advanced parameter used for neural network classification. ***Entropy*** describes the distribution of different classifications for different pixels within an object and is a measure of classification quality (See Chapter 6.5 *Standard classification workflow*).
- **Well.** Well ID of the well in which the object has been detected.
- **Position.** Well position ID of the image in which the object has been detected.
- **Time.** If a time-lapse has been acquired, this value returns the ID of the time-step in which the object has been detected.
- **z-slide**. If a z-stack has been acquired, this value returns the ID of the z-layer in which the object has been detected.
- **Center X.** Position of the object in X within the image.
- **Center Y**. Position of the object in Y within the image.
- **X.** Physical position of the object in X based on stage coordinates (µm).
- **Y.** Physical position of the object in Y based on stage coordinates (µm).
- **Z.** Physical position of the object in Z (µm).
- **UserDef_1/2**. User-defined values based on reclassification hotkey assignment (See Chapter 2.2 *Preferences* "**Reclassification Hotkey Assignment**").

3.7 Derived parameters

The **Assay Settings ▸ Derived Parameters** tab allows performing calculations on the **Parameters** defined in the **Measurements** list by the use of algebraic expressions (+, -, ×, ÷, sqrt, ^ etc. For a complete list see Appendix 6.4). This is especially useful for parameters assigned to different Color Channels.



Derived Parameters list. It lists the **Derived Measurements** to be carried out. Each **Derived Measurement** is labeled with an **ID (D1, D2,…)** and requires a formula that entangles parameter **ID**s from the **Measurements List**.

Name. This is the input field for setting the new **Derived Measurement's** name.

Formula. This is the input field for the algebraic expression that connects the desired parameter **ID**s from the **Derived Measurements List**.

Parameter Selector. The parameters of the measurements list can be selected from this drop-down list and their corresponding ID is entered in the **Formula**-field. Vice-versa if you select a parameter in the formula field you get the corresponding parameter name displayed in the **Parameter Selector** field.

Special Operators

It is possible that a number of individual Sub-objects of the same type can be identified within the mask of a single individual Main Object. A set of statistical operators allows evaluating individual parameters of the entire sub-object population of this type for the individual Main Objects.

Statistical Operators:

- **SUM(ID).** This operator sums up all values of the given parameter ID of the sub objects for each main object.
- **MEAN(ID).** This operator averages all values of the given parameter ID of the sub objects for each main object.
- **STDV(ID).** This operator calculates the standard deviation of all values of the given parameter ID of the sub objects for each main object.
- **MIN(ID).** This operator calculates the minimum of all values of the given parameter ID of the sub objects for each main object.
- **MAX(ID).** This operator calculates the maximum of all values of the given parameter ID of the sub objects for each main object.
- **MED(ID).** This operator calculates the median of all values of the given parameter ID of the sub objects for each main object.
- **VAR(ID).** This operator calculates the variance of all values of the given parameter ID of the sub objects for each main object.
- **IQR(ID).** This operator calculates the interquartile range of all values of the given parameter ID of the sub objects for each main object.
- **MAD(ID).** This operator calculates the median absolute deviation of all values of the given parameter ID of the sub objects for each main object.
- **AAD(ID).** This operator calculates the average absolute deviation of all values of the given parameter ID of the sub objects for each main object.

The parameter resulting from a special operator expression is assigned to the **Main Object**.

New. This allows inserting a new **Derived Parameter** definition into the list.

Remove. This deletes the selected parameter from the list.

3.8 Image processing

Images can be processed before being analyzed in an **Object Finder Module**. Thus, image quality can be enhanced and the object detection improved and facilitated. A set of predefined image processing modules is available.

A variable set of image processing steps can be assigned to each color channel. The configuration of the individual steps takes place via the **Image Processing** window accessible via **Assay ▸ Edit Assay**.

Image processing changes the image data only temporarily! The original image data will not be lost but remain stored. However, in the further course of the assay execution the analysis will always work on the processed data. In order to keep the original data accessible during the assay, virtual channels have to be created instead, see Chapter 3.9, *Virtual Channels*.

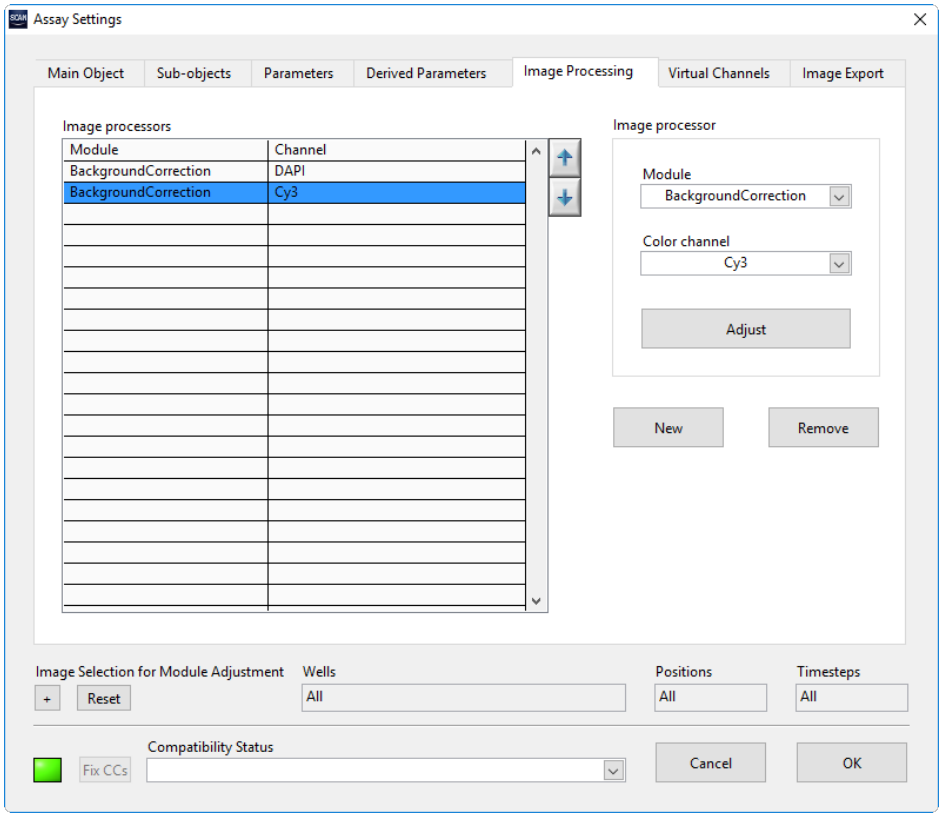


Image processors list. It lists the **Image Processor Modules**, i.e., the processing steps, to be applied on the different **Color channels** of the images. The order of appearance in this list also sets the order of execution. Moving the active entry up or down the list by using the arrow buttons changes the order of execution.

Module. Select an **Image Processor Module** from this drop-down list.

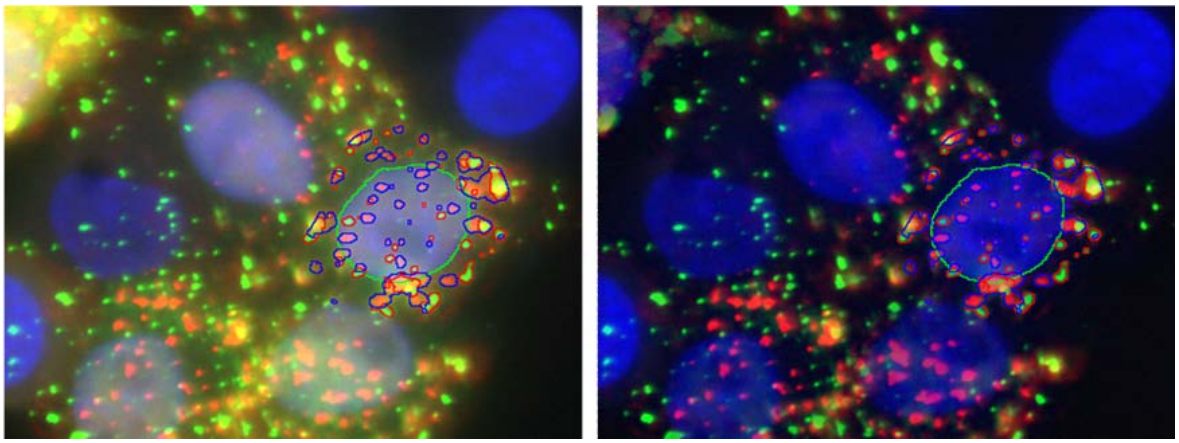
Color channel. Assign a **Color channel** from the drop-down list to the active **Image Processor Module**.

Adjust. Click here to open the **Image Processing** window for the active **Image Processor Module**.

New. This allows inserting a new **Image Processor Module** into the **Module** list.

Remove. This deletes the active module from the **Image processors** list.

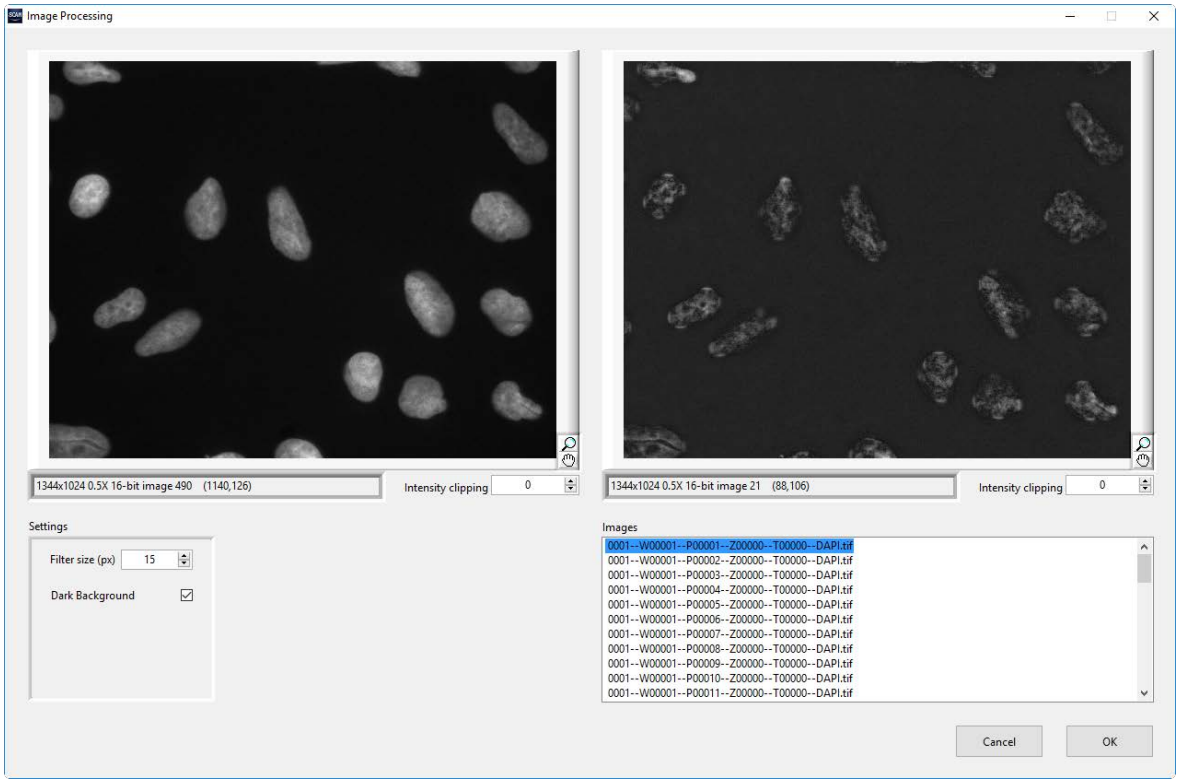
3.8.1 Background correction



For applications involving quantification of intensities and in case of inhomogeneities it is always recommended to use background correction. The algorithm implemented is an intensity-conserving algorithm which ensures that the signals remain unchanged and can therefore be quantified (left image: before background correction, right image: after background correction).

Filter size. This is the only parameter to be set.

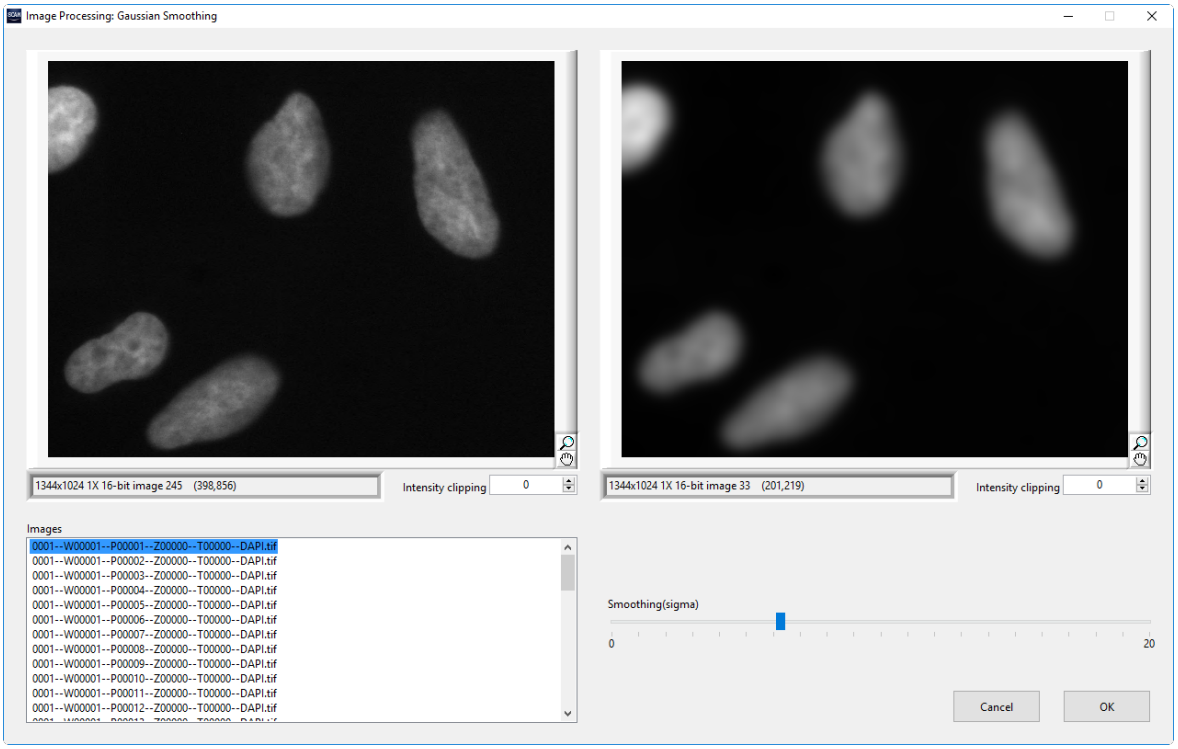
For background correction a **Filter size** of 200 is the default value. For most cases when a general background has to be removed it works fine. However, to better extract e.g. small particles in the nucleus, a background correction filter size of 15 can be suitable (cf. example below).



Example of small Filter size for background correction.

3.8.2 Smoothing

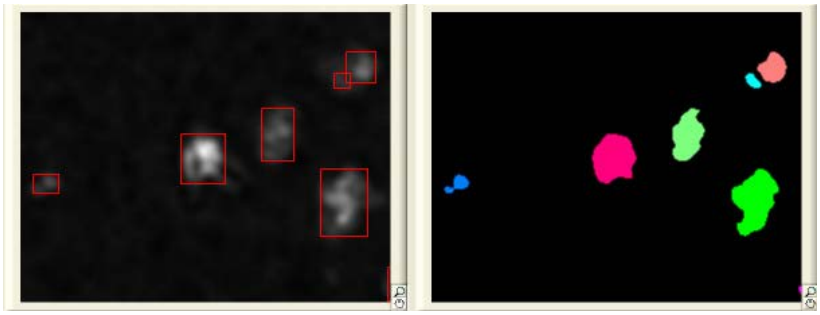
In order to use **Smoothing IP** go to the **Image Processing** tab or the **Virtual Channels** tab and press **New**. Then select **Smoothing** from the **Module** drop down list. Click on **Adjust** in order to change the settings of the IP. The **Image Processing** dialog opens. On the left the original image and on the right the processed image is displayed. In the **Images** list on the left, the acquired image that is to be shown in the displays can be selected. The **sigma** parameter allows you to adjust the smoothing.



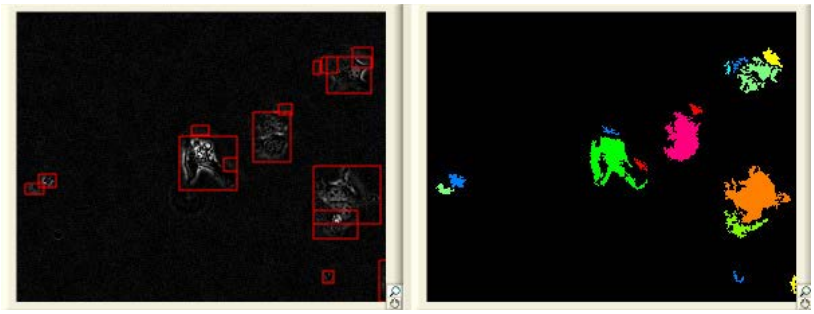
The smoothing image processor

Applications: In some cases it may be necessary to smooth the image before object detection. This is the case for

- noisy images,
- when the border of the objects is fuzzy,
- if the object detection algorithm splits one object in many small fragmented objects (see below).



Object detection without smoothing



Object detection with smoothing

3.8.3 XY Shift

In certain cases it may occur, that image channels are shifted relative to each other in the channel overlay. This is rather often the case if an observation emission filter wheel is used and images are acquired with different emission filters. This function allows correcting the shift along the X- and Y-axes.

XY Shift. Set here the number of pixels the chosen channel is to be shifted along the X- and Y-axes relative to the other channels.

You have to control the result in the main user interface by pressing the **Processed** button. It is useful to zoom into the image so that individual pixels can be detected visually.

3.8.4 Inversion

This function serves to invert the intensities of an image channel, i.e., to convert a channel into its negative image. This is necessary for example if objects are to be detected in transmission images. The object detection tools of scan^R are designed to detect bright objects on dark background. The situation is reverse in transmission images. Thus they have to be inverted prior to the object detection.

Adjust Intensity. This defines the maximum intensity in the converted image. I.e., it is the intensity that the originally darkest pixel will get in the converted image. 4096 is the default for 12 bit assays and corresponds to the maximum intensity in a raw image taken by a 12-bit camera. 65535 is the default for 16 bit assays and corresponds to the maximum intensity in a raw image taken by a 16-bit-camera.

3.8.5 Cut Image

This module allows defining regions of interest inside an image and setting all image parts outside the region to the intensity 0. Different drawing tools are available.

Rectangle

Define a rectangle by mouse drag. Mouse dragging the center changes its position. The size can be adjusted by dragging the corners.

Rotated Rectangle

This is similar to the rectangle tool. Upon mouse-over its central axes are displayed. Dragging the ends of the axes turns the rectangle.

Polygon

Standard tool to draw polygons.

Freehand

Standard tool to draw freehand regions. The region is closed automatically once the mouse button is released.

Circle

Standard tool to draw circles. Close the circle by double click.

Ring Segment

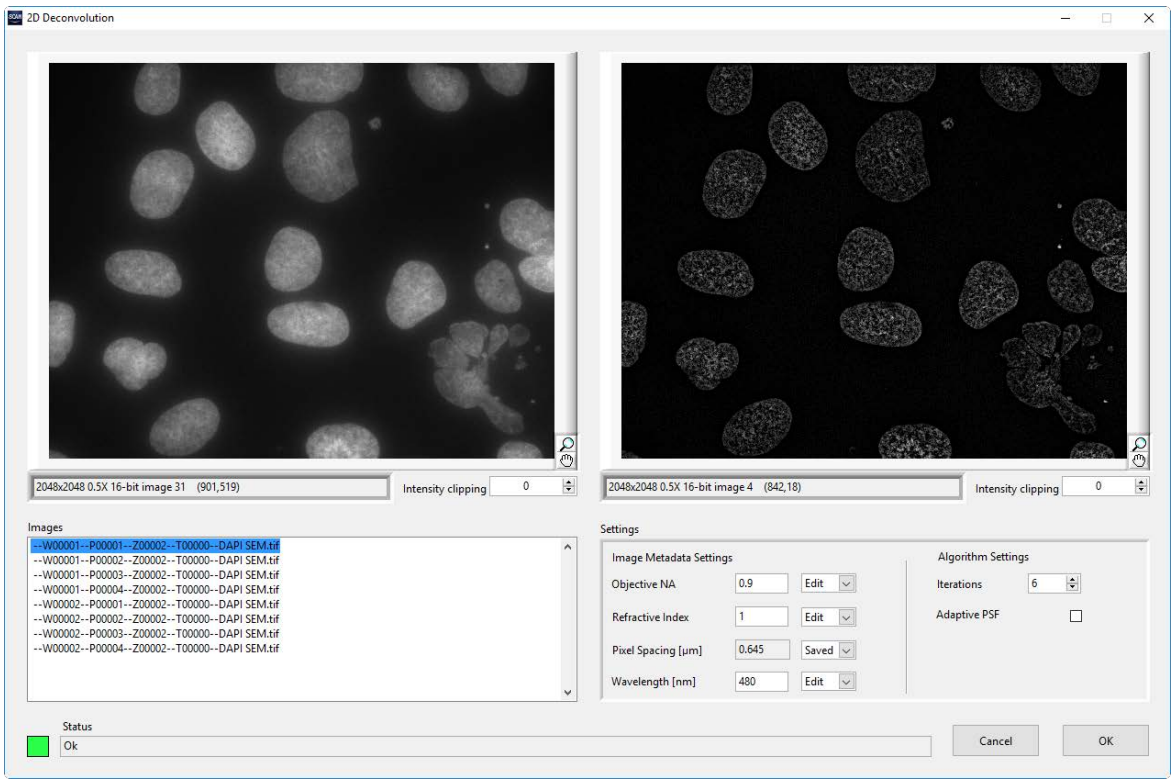
Standard tool to draw rings. Once a ring is drawn the inner and outer borders can be dragged to adjust the thickness. The cutting line can be dragged to convert the ring into a ring segment.

3.8.6 Shading Correction

scanR Acquisition has a Shading Correction Manager for the acquisition of calibration images which allow for correction of illumination inhomogeneities. If available, the relevant calibration images are always stored with the experiment data. With the Shading Correction image processor, the experimental images can be corrected before further analysis if calibration images are available.

3.8.7 2D Deconvolution

The 2D deconvolution module is an image processor which uses known experimental parameters (e.g. numerical aperture, wavelength) in order to remove distortions introduced by the optical system in an optimal way. Therefore, a physical model of the imaging system is estimated and deconvolution is applied to the image data with constraint iterative approach.



The 2D deconvolution module requires additional information about the image data, called “metadata”. The metadata usually is contained with the scan and automatically recognized by the 2D deconvolution module. For using the metadata saved with the scan, select the **Saved** option close to the relevant metadata. In cases where (part of the) metadata is not available with the scan or other metadata is supposed to be used, the **Edit** options allows using manual settings. Usually it is required to set the **Wavelength** manually because no information regarding the wavelength is saved with the scan data.

Once the metadata is complete, the deconvolution algorithm can calculate the de-convolved image and display a preview. It might be necessary to try different settings for the number of **Iterations** used internally and to select if **Adaptive PSF** (point-spread-function) should be used.

The 2D deconvolution is a powerful image filter which can also be understood as a kind of optimal sharpening filter. It can be very useful if segmentation of special features of the raw image data is difficult. Since the 2D deconvolution is working on 2D image data only, however, it is limited for fundamental reasons. More sophisticated deconvolution strategies require a z-stack of image data which contains significantly more information regarding light distribution along the optical axis.

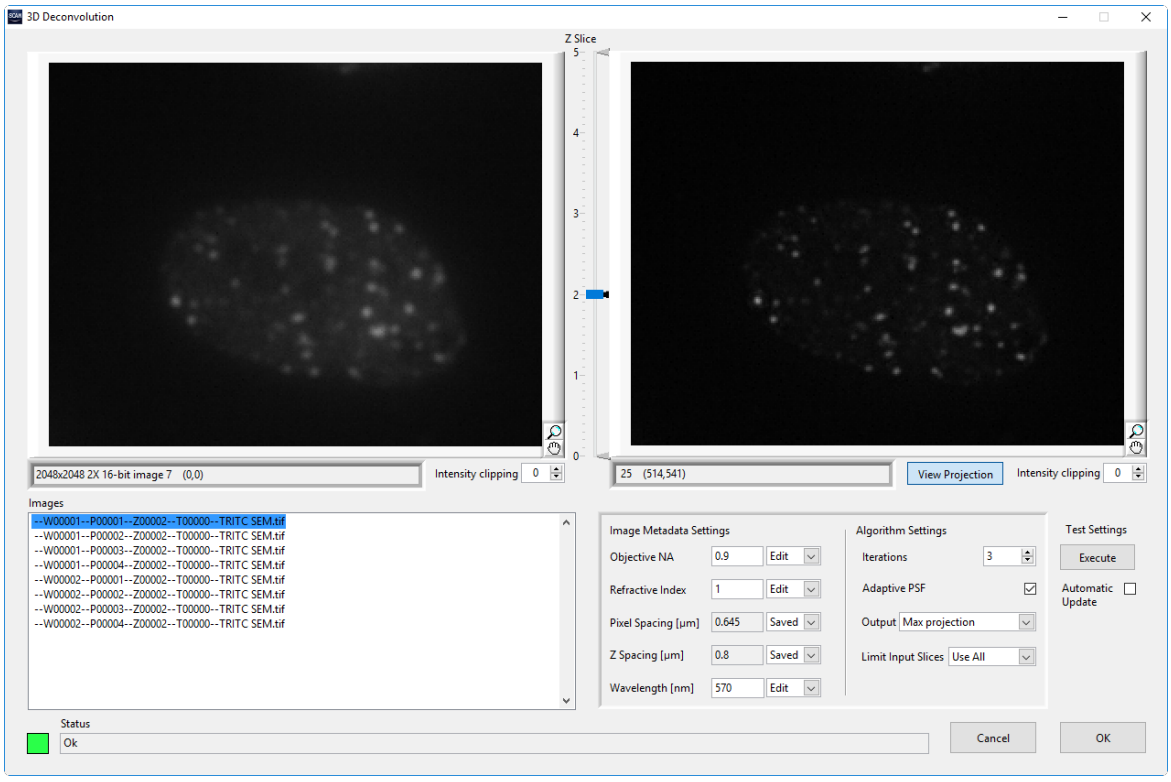
3.8.8 Z Projection

scanR Analysis is a 2D Analysis software. Many 3D applications, however, can be addressed very efficiently by condensing information from (3D) z-stack data into (2D) projections. The Z Projection module can be used to project the z-stack data only for Analysis and keep the complete 3D data for other purposes. This is an alternative workflow compared to saving only the projections and not the complete z-stacks during acquisition (see scanR Acquisition instructions manual for details).

The Z Projection module must be applied as first module to a Color Channel, i.e. no other Image Processing module can be applied to the same Color Channel before the Z Projection module but only after the Z Projection module.

3.8.9 3D Deconvolution

The 3D deconvolution module is an image processor which uses known experimental parameters (e.g. numerical aperture, wavelength) in order to remove distortions introduced by the optical system in an optimal way. Therefore, a physical model of the imaging system is estimated and deconvolution is applied to the image data with constraint iterative approach. In contrast to 2D deconvolution, 3D deconvolution works on a z-stack of image data per position. This makes it possible to correct for out-of-focus light in a much more reliable and precise way. On the other side, the calculation of the correction can take significantly more computation time. Only one 3D deconvolution module can be used per color channel and the 3D deconvolution must be the first Image Processor and Virtual Channel applied to a color channel.



The 3D deconvolution module requires additional information about the image data, called “metadata”. See the documentation for the 2D deconvolution module for details on metadata. The 3D deconvolution module requires one additional piece of metadata, the **z-spacing**. In addition to the number of **Iterations** and whether to use **Adaptive PSF** (point-spread-function), the 3D deconvolution algorithm requires the selection of the **Output** image. While the input will always be a z-stack for each position, the output will be a single image which can be either the **in-focus slice**, or a projection of the output z-stack. In order to reduce processing times, the **Limit Input Slices** functionality allows selecting a subset of the original z-stack as input.

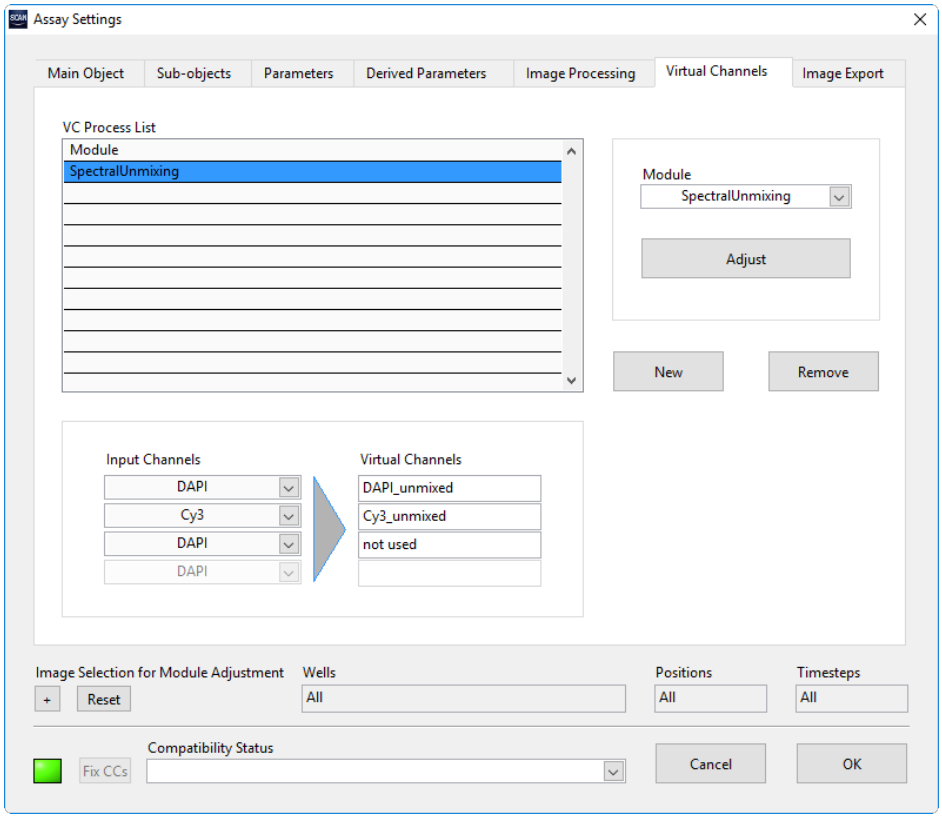
Calculation times can be in the order of many seconds to minutes, depending on the size of the input image data and the algorithm settings. For this reason the preview image will only be updated if the **Automatic Update** option is selected. If this option is not chosen, pressing the **Execute** button once will update the preview windows when the input image or the **Algorithm Settings** have changed.

The original image stack as well as the output image stack can be browsed by using the **Z Slice** slider. The slider indicated the in-focus slice as well as the used range of input slices. By setting the slider, a particular z-slice can be selected and the effect of the 3D deconvolution can be judged. The **View Projection** button toggles the output display between the processed z-stack which can be browsed with the **Z Slice** slider and the output of the 3D deconvolution module as defined by the **Output** control.

The 3D deconvolution module must be applied as first module to a Color Channel, i.e. no other Image Processing module can be applied to the same Color Channel before the 3D deconvolution module but only after the 3D deconvolution module.

3.9 Virtual Channels

Virtual channels are image channels that are not created via image acquisition during the execution of a scan. Instead they are a result of post-acquisition image processing and added as new channels to the original image data. These can then be used for further analysis steps, e.g. object detection.



New. Click here to create a new entry in the **VC Process List**.

Virtual Channels list. Default names for the virtual channels resulting from the processing are automatically created. It can be changed manually.

Remove. Click here to delete the active entry from the **VC Process List**.

Module. **Background Correction**, **XY Shift**, **Inversion**, **Cut Image,** etc. are the same functions as described in Chapter 3.7, Image Processing.

Adjust. Click here to open the dialog of the selected module.

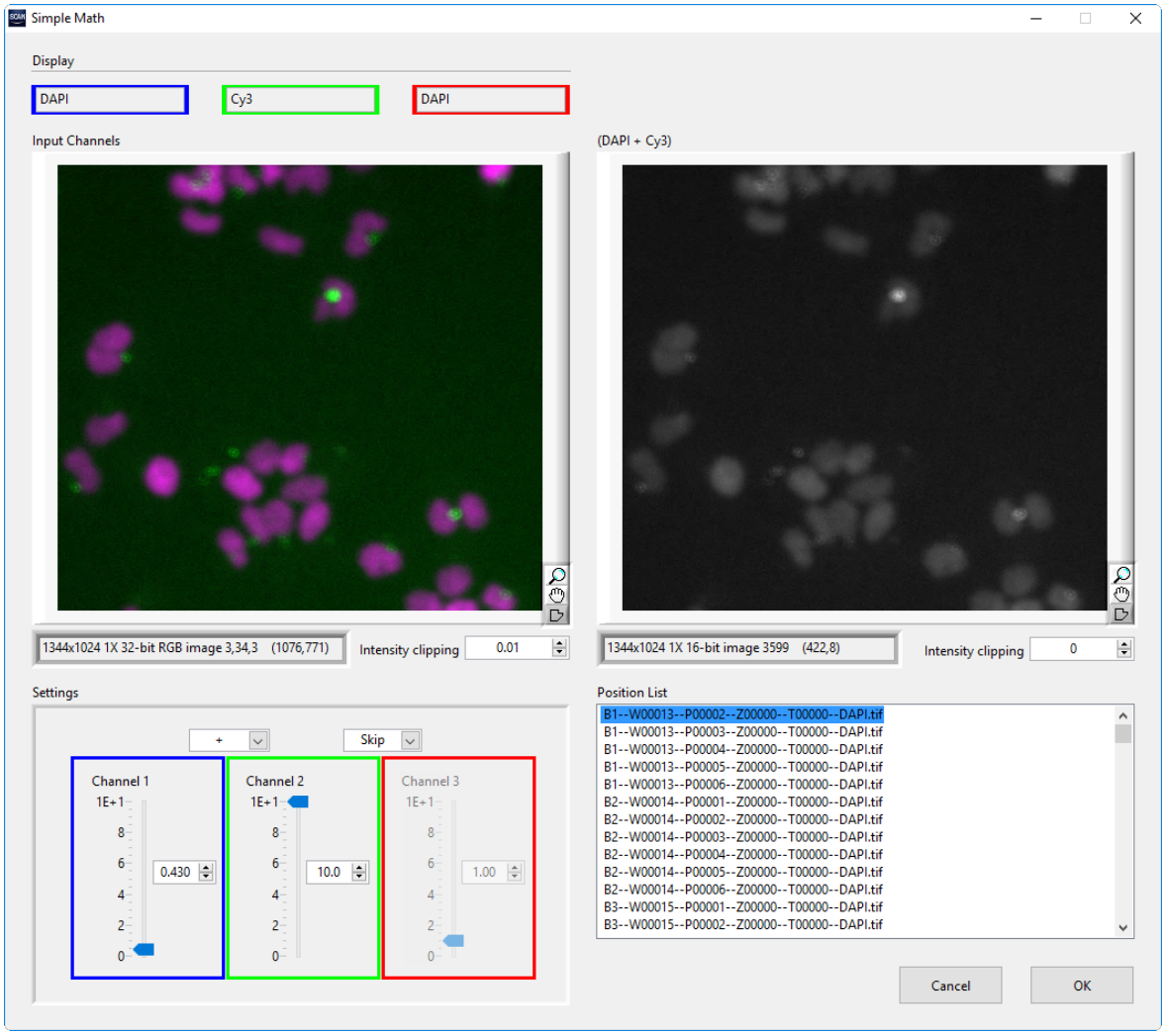
3.9.1 Simple Math

The Simple Math module serves to perform calculations on the image through basic arithmetic operations: addition, subtraction, multiplication and division.

Module. Select **Simple Math** from the **Module** pull-down list to set this module as active entry in the **VC Process List**.

Input Channels list. All color channels of the images are possible **Input Channels** for the **Simple Math** module. The channel selected as first **Input Channel** is always the first source of the arithmetic operation. The other channels are possible second sources.

Adjust. Click here to open the **Simple Math** dialog. It contains displays of the overlay of the **Input Channels** and of the **Virtual Channel** that is created by the selected arithmetic operation.

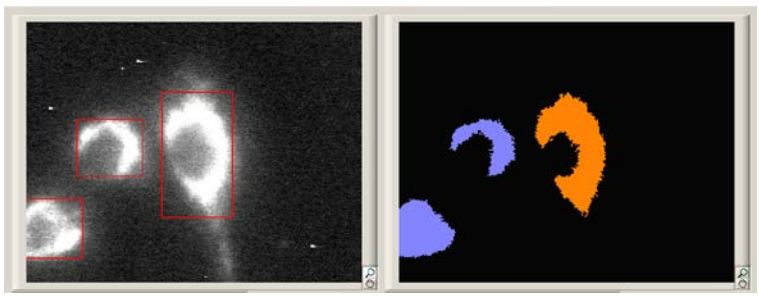


Settings

Channel 1, 2, 3. Each of the **Input Channels** can be weighed prior to the arithmetic operations by using the sliders or entering a multiplication value between 0 and 10 into the respective boxes.

Arithmetic operations selectors. Select the operation (**+**, **–**, **×**, ÷) that is to link Channel 1 and Channel 2 from the first pull-down selector (e.g. DAPI+FITC). Select the operation that is to link the result of Channel 1 and 3 with Channel 3 (e.g. (DAPI+FITC)**×**TxRed) from the second pull-down selector. Select **Skip** to deactivate Channel 2 or Channel 3, respectively.

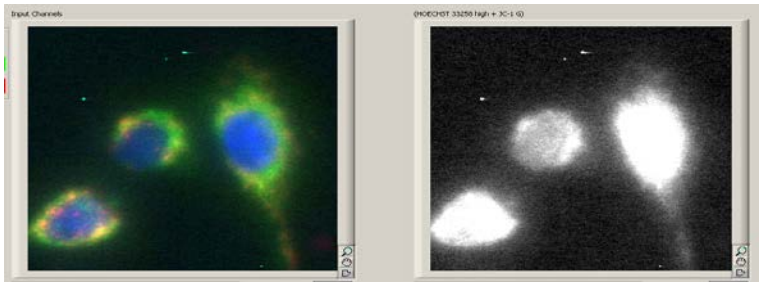
3.9.1.1 Example: Cytoplasm not detectable on a single color channel



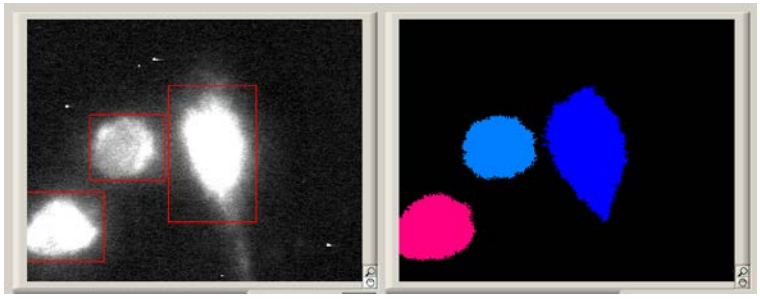
In this example it is not possible to detect the cytoplasm on a single color channel because the area of the nucleus has very little cytoplasmatic staining:

Problem. The detection is incomplete because staining is missing on the nucleus area.

Solution. The nucleus staining and the cytoplasmatic staining are added as VC.



Result. Full Cell Segmentation of the complete cell can be performed on the calculated new channel.



3.9.2 Spectral Unmixing

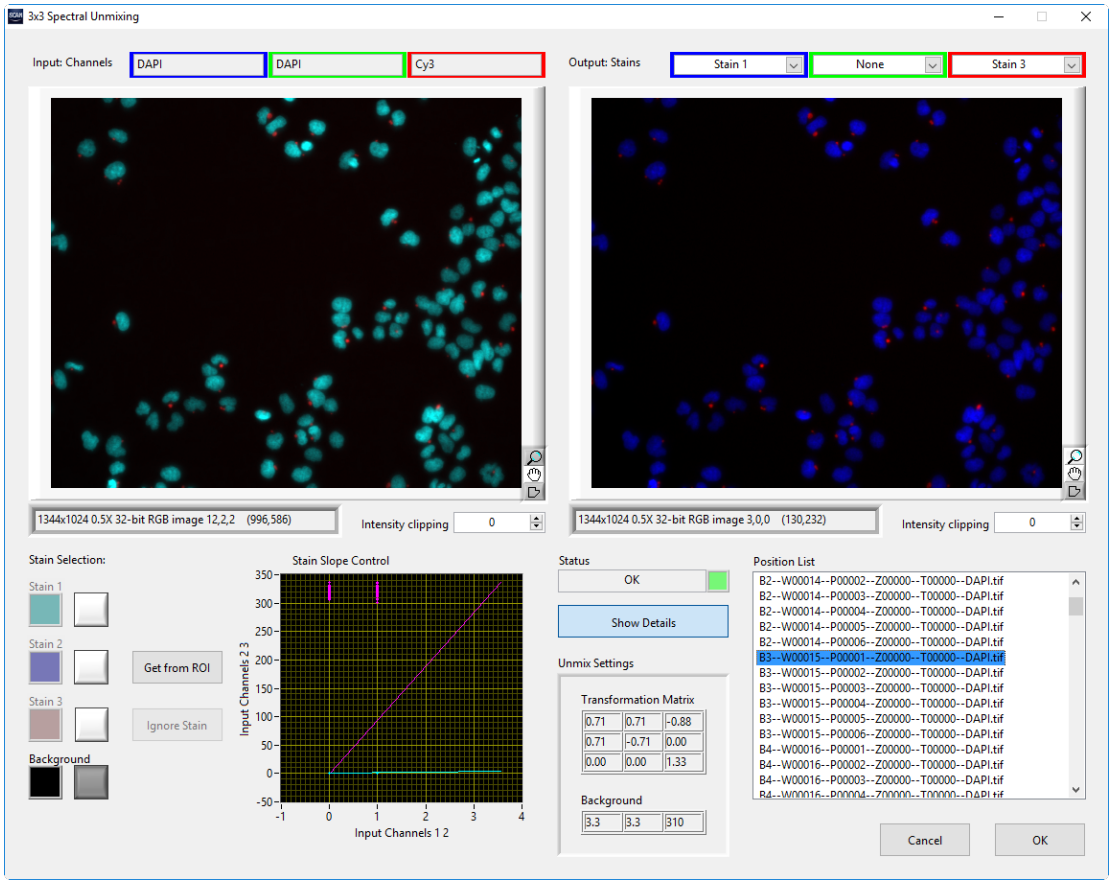
A major problem in live cell imaging arises from the use of different fluorochromes with overlapping spectra in one multi-labeled sample, impairing a number of applications. Even with the use of high quality optical filters it is not satisfactorily possible to separate the spectral information. The consequence is the excitation and imaging of structures that are labeled with one of the present fluorophores when using a filter set that is actually chosen to excite and image another fluorophore.

With the **Spectral Unmixing** module it is possible to separate and resort the contribution of different fluorochromes to the total signal in each color channel and redistribute the different color intensities. It thus improves the spectral resolution of the channels considerably and facilitates for example co-localization studies.

Get from ROI. Takes the mean value within the marked region as Stain/Background.

Ignore Stain. Click here to ignore the third channel in order to properly unmix two-channel images.

Background. Spectral Unmixing yields quantitatively meaningful data only if a background subtraction is performed prior to it.



Show details. Displays the matrix created by the selection of the stains. This matrix is used for the processing of the images. The entries of the matrix can be changed manually. A graphical representation is displayed.

Output channels. Select the stains to see the result of the spectral unmixing.

1. In the **Module** drop-down list select **Spectral Unmixing** and press **New.**

2. Press **Adjust** to start the **3x3 Spectral Unmixing** dialog.

3. Click on the Background button.

4. Mark a background area using the **drawing tool** (button on the bottom right corner of the image displays).

5. Click the **Get from ROI** button.

6. Identify a structure that contains just the first fluorophore.

7. Mark the structure using the **drawing tool** (button on the bottom right corner of the image displays).

8. Click on the **Stain 1** button.

9. Click the **Get from ROI** button.

10. Repeat steps 6 – 9 for the other fluorophores (stains).

In order to perform the spectral unmixing the software has to determine the contribution of the fluorescence of different fluorophores to the different color channels. To do so, ideally series of mono-labeled reference samples would be used. In case such samples are not available for each of the fluorophores, molecular structures have to be identified by the user that are certain to contain only one of the fluorophores and that do not spatially overlap with structures containing other fluorophores.

The result of the spectral unmixing is shown in the right display of the **3x3 Spectral Unmixing** dialog.

Unmixing dual-labeled samples. In the case of two-channel images the third input channel will be occupied by one of the two available channels. Click **Ignore stain** to ignore the third channel in order to properly unmix two-channel images.

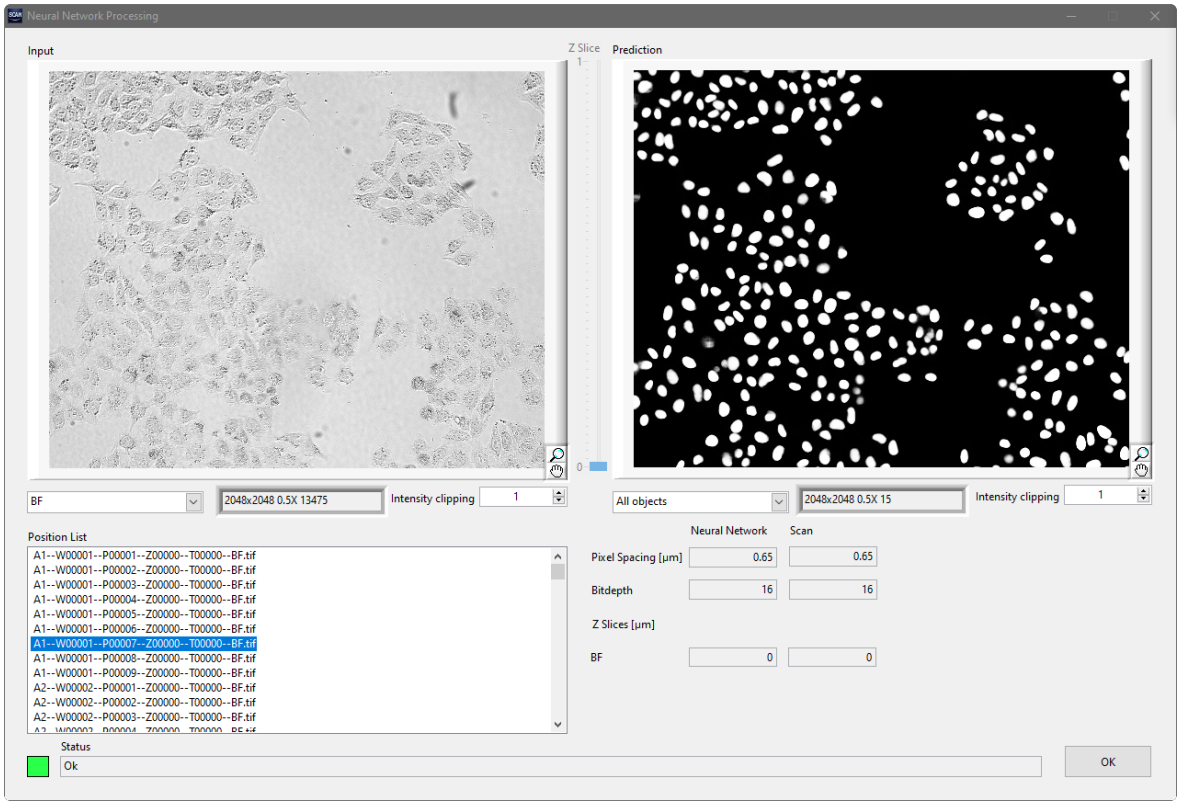
3.9.3 Neural Network Processing

The **Neural Network Processing** module is used to apply any neural network to one channel or a combination of multiple channels. The neural network is used to perform predictions of objects (for example cells or subcellular compartments or other) in the images according to its training. The processing result output is a new **Virtual Channel** that contains the probabilities that the objects to be predicted are at a particular location within the image.

When selecting the Neural Network Processing module, a new drop-down list becomes available which allows selecting a **Neural Network** by its name. Once a neural network is selected, the number of input channels (one or multiple) required for this neural network are enabled, and a **Description** of the selected neural network is shown.

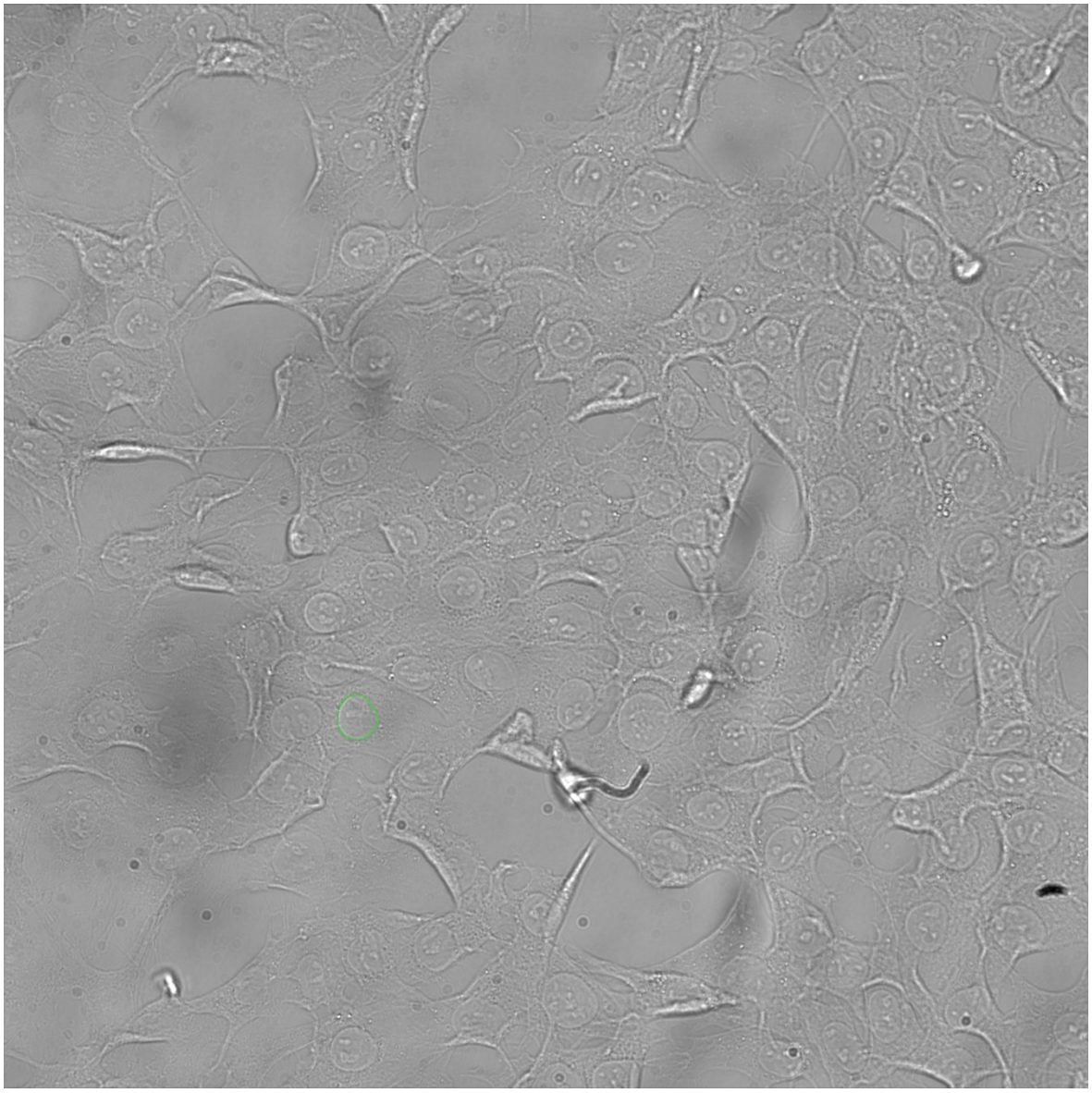
Multi-channel neural networks. Many neural networks operate on a single input channel. For more complex analysis tasks, however, it can be necessary to employ neural networks which are trained to work on multiple channels at the same time. This will be indicated in the Description, and it is required to provide the correct order of input channels according to the description.

Z-stack neural networks. Many neural networks operate on a 2D (single z-layer) input channel. For more complex analysis tasks, however, it can be necessary to employ neural networks which are trained to work on z-stack data. This will be indicated in the Description, and it is required to provide an input channel with sufficient z-stack data for correct prediction.

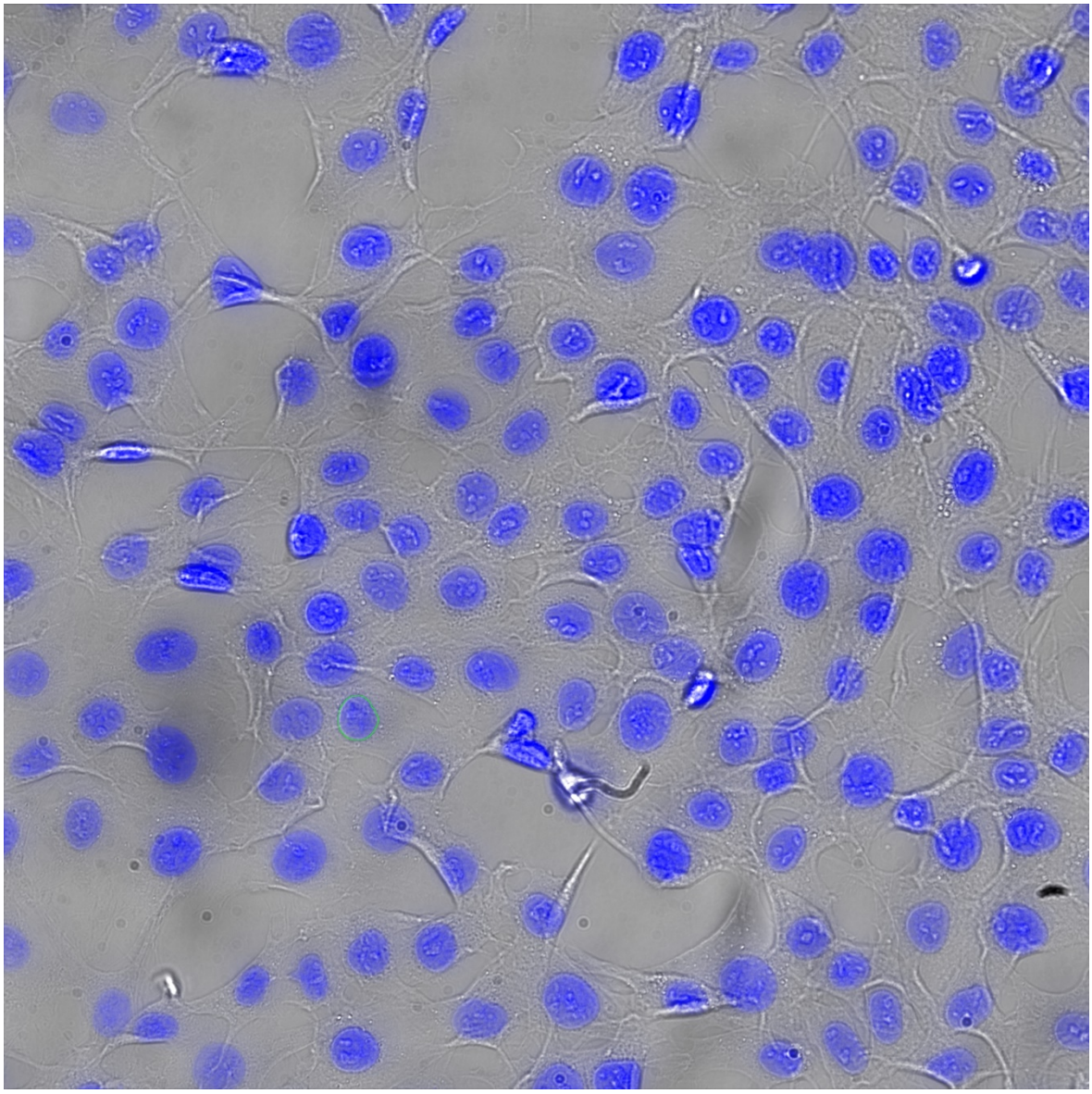


3.9.3.1 Example: Detection of nuclei in bright field channel

If, for example, it is required for an assay to detect nuclei in transmission bright field images, the best solution might be to solve this task using a neural network. A typical image of the input channel might look like shown in the following figure.



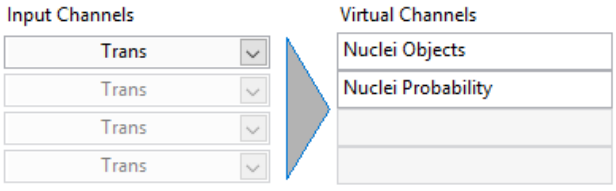
A neural network trained to find nuclei in bright field images can predict for each pixel of the image the probability that this pixel belongs to a nucleus. The probability result, a map of the size of the image, is provided as a new virtual channel and can be used for object detection. Furthermore it can be used in the main Image Viewer (see Chapter 2.5, *Using the Image Viewer*) like any other virtual channel and displayed as one channel of the combined multi-color image.



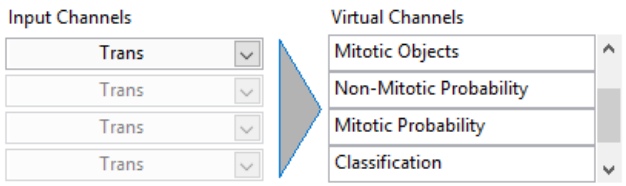
3.9.3.2 Detection and Classification

There are two different kinds of neural networks available with scanR. Depending on the training, a network can detect objects of a single class (for example, nuclei), or it can detect and differentiate multiple classes of objects (for example, nuclei undergoing mitosis and nuclei not undergoing mitosis).

Single class detection networks have a single pair of Virtual Channels as output of the Neural Network Processing. The output consist of one objects map and one one probability map which can be used for object detection as any other image type.



Classification networks have multiple Virtual Channels as output of the Neural Network Processing.



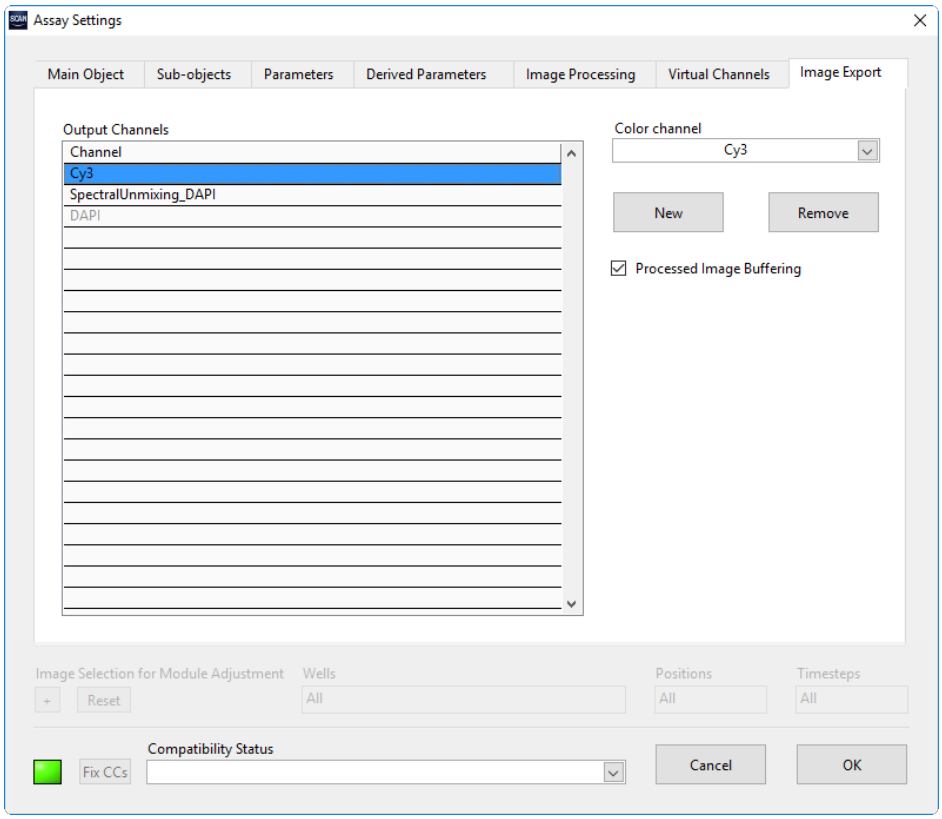
The different output channels have different functions:

- The ***All Objects*** channel enables direct detection and segmentation of all objects, independent of their class.
- One probability map for each class separately.
- The ***Classification*** channel contains all objects with their respective classification, coded in image gray values (class 1 = gray value 1, class 2 = gray value 2, ..).

For details and descriptions of the typical classification workflows see Chapter 6, *scanR AI*.

3.10 Image export

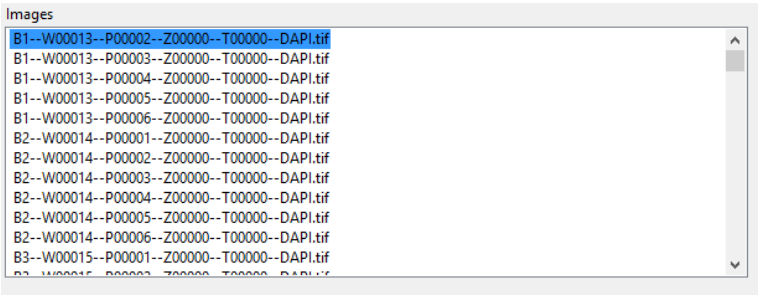
The **Image Export** tab allows selecting one or more (processed) color channels which will be exported during analysis.



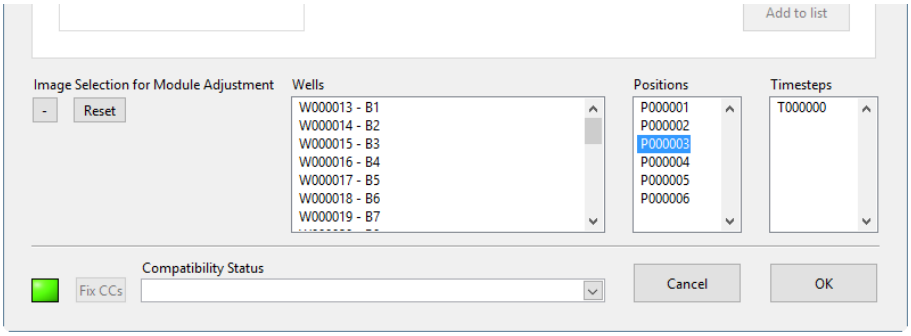
Technically, a new folder "processed" will be created next to the original "data" folder. As soon as analysis of the data has finished, the "processed" folder contains the processed images of all selected color channels. When available, the saved, processed images will be used when browsing the analysis results to optimize speed of visualization and avoid unnecessary and time-consuming re-processing. **Processed Image Buffering** automatically adds Color Channels to the list which are using with time-consuming image processors, for example 3D Deconvolution.

3.11 Image selection for module setup

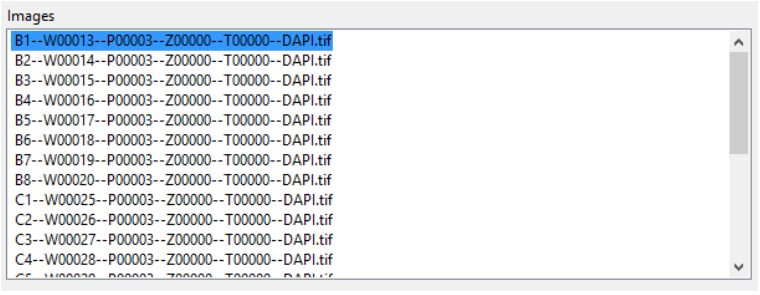
Setting up a new assay or optimizing assay settings often requires changing parameters of object detection or image processing steps using the “Adjust” buttons. The module windows then provide a list of all images at all wells, positions and time steps so that parameters can be adjusted with direct control of the effect.



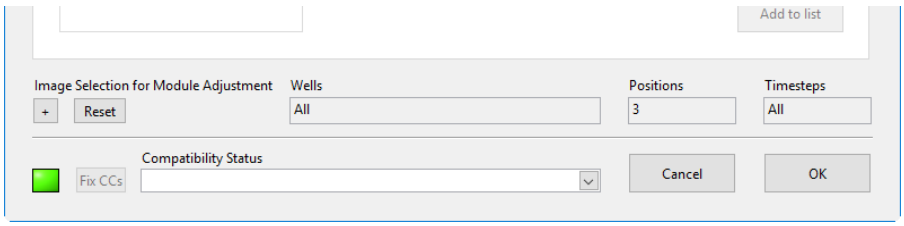
The list of all images can be long and sometimes it is useful to reduce it with the **Image Selection for Module Adjustment** filter. For example, only the positive and negative control wells or only the first time step or a few position from each well can be selected to have the possibility to set up the modules with the most relevant images. In this example, only position 3 in each well is selected.



This will result in a reduced list of images in the module windows, only listing images from the 3rd position for each well.



Once the selection has been done, the filter can be folded up such that only the summary (which well, positions, timesteps are used for module adjustment) is shown.

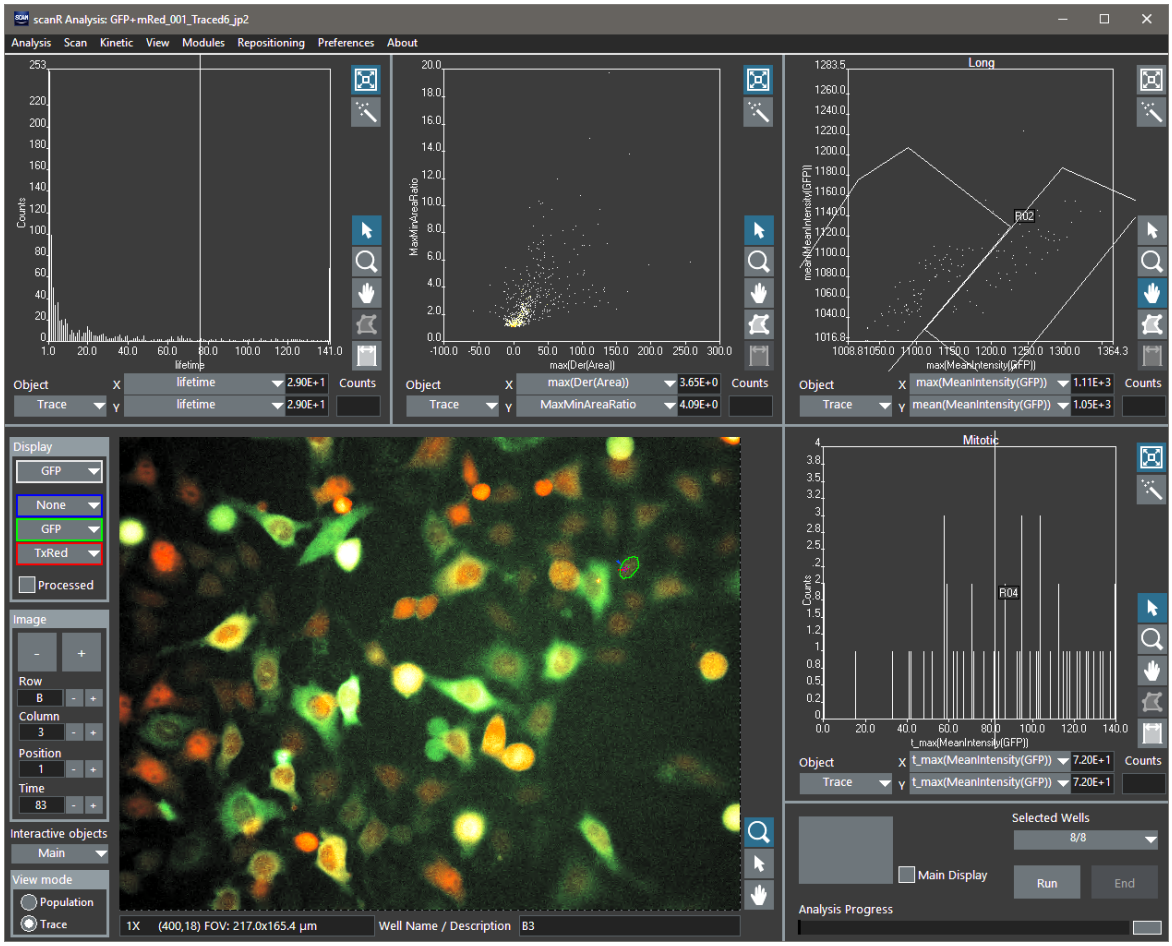


The filter is not part of the assay and hence not saved with the assay. It is only used during assay setup as convenience tool. This has no effect on the analysis itself.

3.12 Kinetic – analyzing time-lapse data

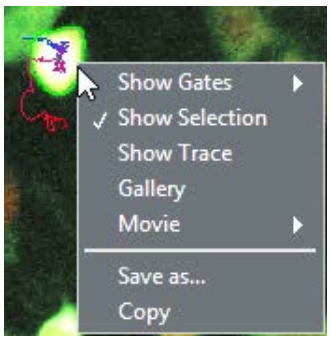
The tracking function of scanR Analysis allows the analysis of objects over the course of time, i.e., in experiments that consist of time-lapse acquisitions. It relates any object detected in an image to the same object in the previous and subsequent images in the time-lapse series acquired at the same stage position. Thus the change of the parameters that are measured according to the assay settings can be followed over the course of time.

To enter the **Trace View** select **Trace** in the lower left part of the front panel or in the menu bar, select **Kinetic ▸ Trace view.** The front panel display changes such, that the displayed objects change from **Main** to **Trace** and also the available X/Y-parameters in the histograms are changed according to the parameters that are defined in **Kinetic ▸ Define Parameters** (See Chapter 3.12.2.1, *Original parameters*).



When clicking on one tracked object in the image now not only the object is highlighted with a green border but also the trace it covered during acquisition will be displayed. With time the object moves from the blue end of the line to the red end. Like in the **Population View** the detected object and the corresponding data points in the histograms are directly linked.

Right-clicking on an object in the image yields the following context-menu:

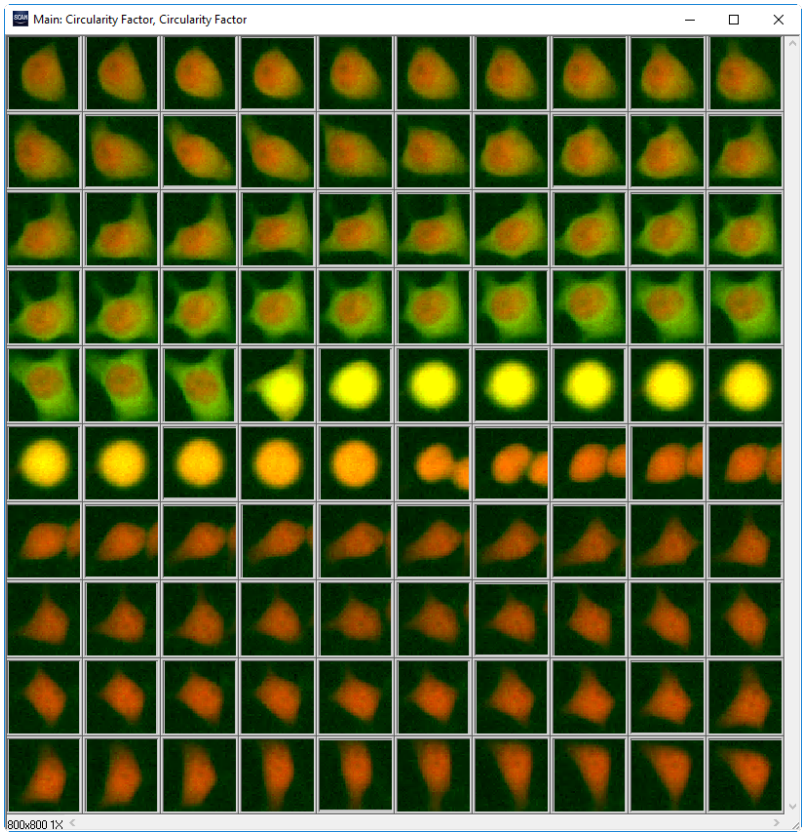


Show Gates. It has the same function as in the **population view**. When you select a gate from the list, the objects that fall into the gates are marked with a box in the front panel image.

Show Selection. Enables or disables if cell outlines and traces of selected cells are displayed.

Show Trace. This opens the Trace Viewer which displays the time curve for the selected object. (See Chapter 3.12.3, *Trace viewer*)

Gallery. This command displays a time-gallery of the selected object.

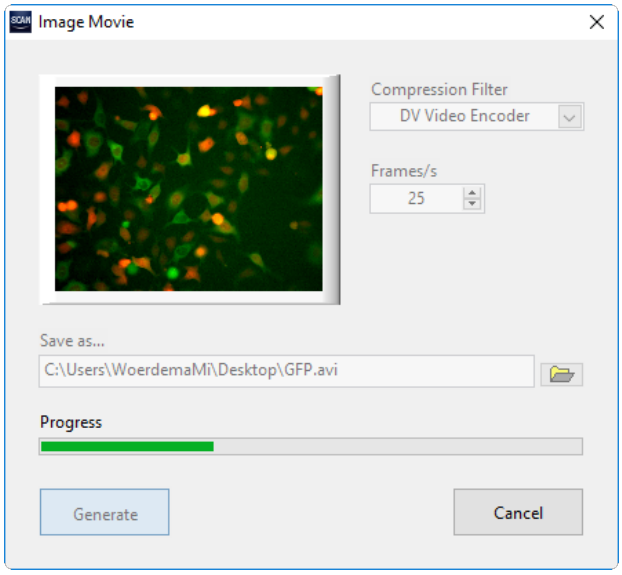


Gallery display in time-lapse mode

Movie. This opens the dialog **Trace Movie** which allows you to export a movie of a single **trace** or a complete **image** (position)**.**

Save as … This allows you to store the displayed image separately.

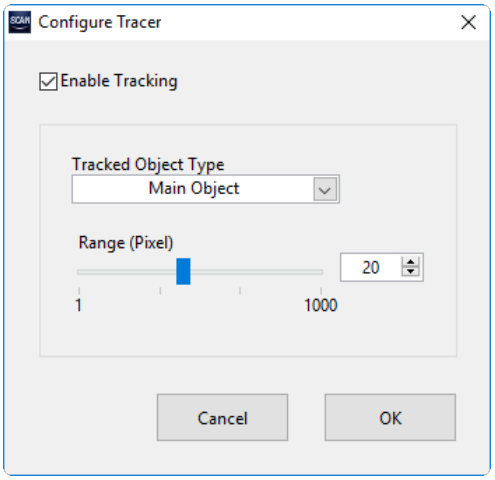
Copy … This allows you to copy the displayed image into the clipboard for pasting in other software.



3.12.1 Tracking configuration

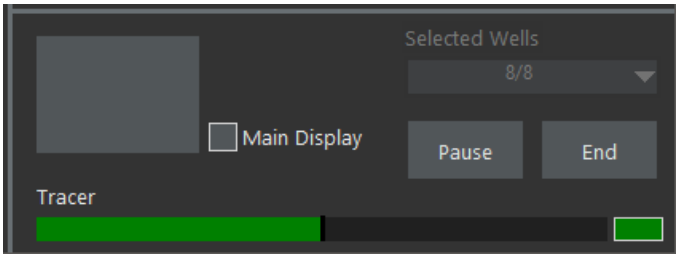
The **Kinetic ▸ Configure Tracer** command opens the **Tracker Settings** window. It also opens automatically when the **View mode: Trace** is activated without any tracking parameters being set already.

Tracked Object Type. Select here the kind of objects to be tracked, i.e. main objects or any of the sub-objects – if such are defined in **Assay Settings ▸ Sub-objects**. (See Chapter 3.4, *Sub-object finder: detecting sub-objects*)



Range (Pixel). Set here the maximum difference that is allowed to change between two frames in order to maintain a track. If the difference exceeds the **Range** an object will NOT be related to similar objects in the previous and subsequent images of the time-series.

OK. Click here to start the tracking. The progress can be followed in the status bar at the bottom right corner of the scan^R Analysis main interface.



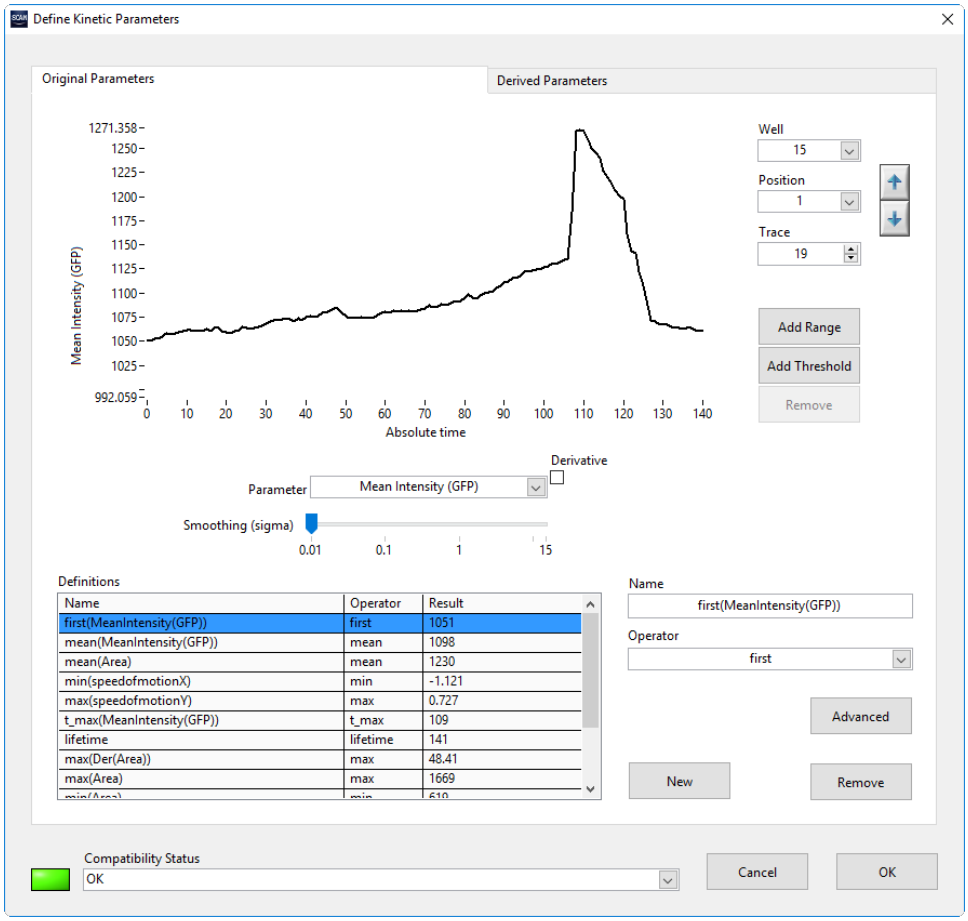
3.12.2 Track analysis parameters

During the tracking, not only the X/Y-positions of an object over time are calculated, but also the time course of the parameters previously determined in **Analysis ▸ Assay Settings… ▸ Parameters** (see Chapter 3.6, *Measurement parameters*) is extracted of the data. The curves for each individual object are displayed in **Kinetic ▸ Define Parameters… ▸ Original Parameters.** This dialog allows defining which kinetic parameters of these curves are extracted, e.g. the maximum intensity, the time of maximum intensity or the duration of an increase in signal.

3.12.2.1 Original parameters

Open the **Define Kinetic Parameters ▸ Original Parameters** window directly via **Kinetic ▸ Define Parameters…**. At startup it shows the kinetics graph of one of the measured parameters of the first track in first stage position of the first well acquired and selected for analysis. (See Chapter *2.7* Interactive plate result view.)

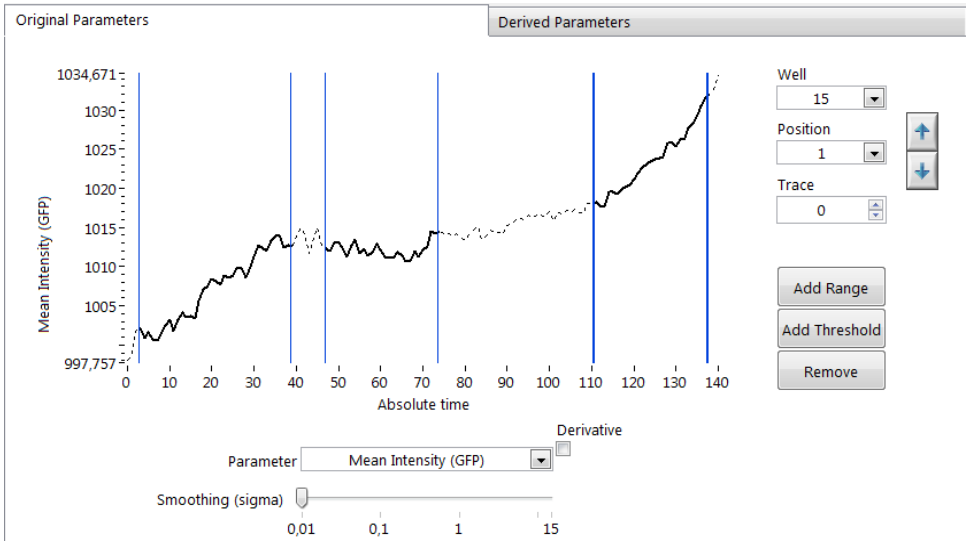
To navigate through the curves use the **Well**, **Position**, and **Trace** selectors on the right. Each **Trace** represents the object in subsequent time frames that the tracer, according to the settings in **Kinetic ▸ Configure Tracer** detected as belonging together. In the **Original Parameters** tab you can set the **Operators** that are applied to the time curves to quantify the time curves. By applying an **Operator** (min, max, std, first, etc.) the time curves again are reduced to single values per curve that in turn can be displayed in a 1-D or 2-D histogram. For example when **TotalIntensity(TxRed)** was selected in the **Parameter** tab of **Assay Settings**, the time curve of **TotalIntensity(TxRed)** can now be plotted. By applying the **Operator max** on this time curve, for all traces the maximum of **TotalIntensity(TxRed)** will be calculated.



Trace parameters.

Well, **Position**, **Trace**. You can navigate through the set of curves using these functions and their arrow buttons.

Add Range. Click here if you want to analyze just a part of the trace. Two vertical blue bars appear in the graph to define the upper and lower limit of the range. They can be moved via mouse-drag. The analyzed part of the curve is displayed solid. The remaining part is displayed as a dashed line. Click **Add Range** several times to create multiple ranges.



Three ranges applied to a trace

Add Threshold. Click here to open a dialog which allows you set an upper or lower bound for the curve. A blue bar appears that marks the upper or lower part of the curve, respectively. The bar can be moved via mouse-drag.

Remove. Click here to remove the **Range** limits.

Parameter. Select the measurement parameter of which the time curve is to be shown in the graph display.

Derivative. Check this option to calculate and display the first derivative of the kinetics time curve of the selected **Parameter**.

Smoothing (Sigma). This function applies a smoothing filter on the track data. Set the strength of the smoothing with the slider. The graph display is being updated immediately.

Definitions. This lists the names of the values to be determined from the curves as well as the operators used. In the **Result** column the values for the selected trace for all parameters are listed.

Operator. Select an operator from the drop-down list. This will be applied on the currently selected **Parameter**. For example, if *mean* is applied on the *Mean Intensity* of the tracked objects the time average of the mean intensity of each object will be calculated.

The available Operators are:

- **Lifetime.** Gives the number of contiguous time points the particle is detected
- **Sum.** Calculates the sum of the parameter value over the time
- **Mean.** Calculates the mean value of the parameter
- **Min.** Takes the minimal value the parameter reaches
- **Max.** Takes the maximal value the parameter reaches
- **Std.** Calculates the standard deviation of the values
- **First.** Takes the value of the parameter at the first point of the curve
- **Last.** Takes the value of the parameter at the last point of the curve

- **T_max.** Takes the time point when the parameter is maximal
- **T_min.** Takes the time point when the parameter is minimal
- **T_first.** Gives the time point when the trace starts
- **T_last.** Gives the time point when the trace ends
- **Num_zero_crossings.** Gives the number of zero crossings of the curve
- **Num_local_max.** Gives the number of local maxima of the curve
- **Num_local_min.** Gives the number of local minima of the curve

Name. The name of the analysis function is set automatically as <Operator>(Parameter). It can be changed manually.

New. Click here to create a new list entry.

Remove. Click here to delete the selected list entry.

Advanced. Click here to show advanced options. The **Derivative** checkbox will be replaced by a list that allows you to apply **Anti(-)/Derivatives(+)** of higher order. In the **Operator** drop-down list a new option becomes available: the **curve-fit**.

Curve-fit. This **Operato**r allows you to fit certain models to the parameter curves. Upon selecting **curve-fit** from the **Operator** drop-down list the **Define**-button becomes accessible.

Define. This opens a dialog to set the options for **curve fitting**.

As objects might move in or out of the focal plane, or the value entered for **Range** might be set too small, there might be a large number of short traces. Therefore **Lifetime** is a useful parameter as it allows setting a gate on long traces.

3.12.2.2 Curve fitting

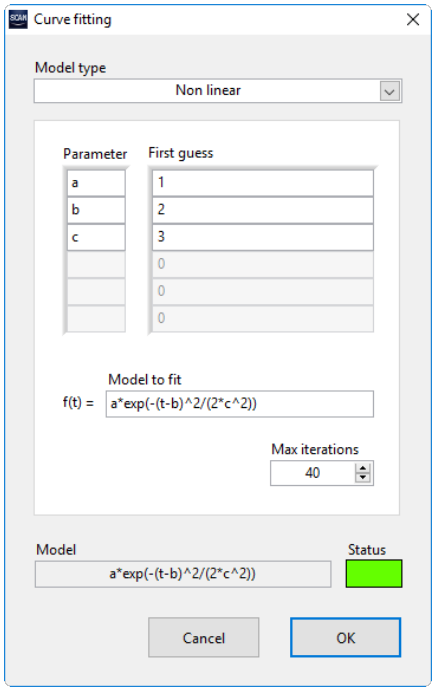
Model type. Select the type of model that is to be fitted to the curves. You can select **Polynomial**, **Exponential**, **General ls linear** and **non linear.**

Polynomial. The only parameter to be set is the **Polynomial order**. E.g., the model function is $f(t)=a_0+a_1 \cdot t+a_2 \cdot t^2$ for **Polynomial order** of 2. In this example the parameters a_0, a_1 and a_2 are fitted to the curves.

Exponential. No parameters have to be set. The model function is $f(t)=a \cdot exp(ct)$. The parameters *a* and *c* are fitted to the curves.

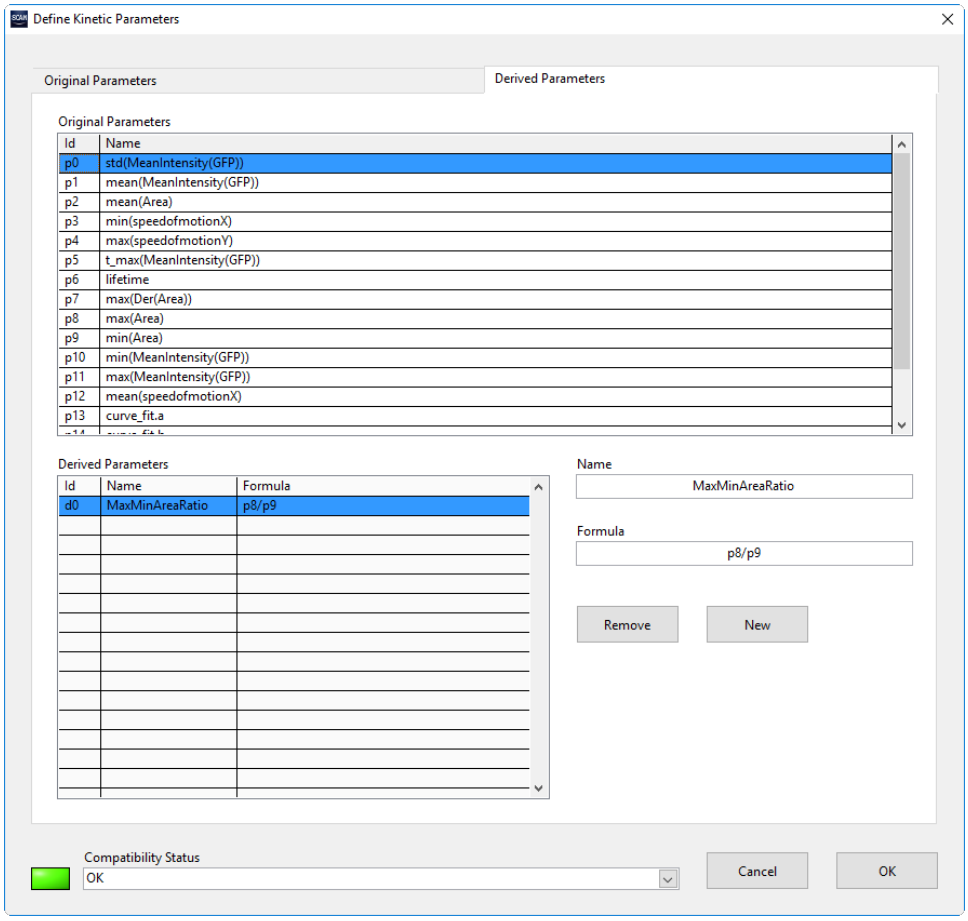
General ls linear. General linear least-squares fit. Here you can enter arbitrary **Basis functions** (linear, exponential, trigonometric,…) which will be combined linearly. E.g. set $f_1(t)=17 \cdot t+4$ and $f_2(t)=exp(-(t+2))^2$ will be combined linearly to $f(t)=a_0 \cdot f_1(t)+ a_1 \cdot f_2(t)= a_0 \cdot (17 \cdot t+4)+ a_1 \cdot exp(-(t+2)^2)$. This function represents a linear combination of a line and a Gaussian. The parameters a_0 and a_1 are fitted.

Non linear. Here you can enter an arbitrary model function to be fitted with up to 6 parameters. You have also to enter the start values as **First guess**. E.g. if you want to fit a Gaussian you set $f(t)=a \cdot exp(-(t-b)^2/(2c^2))$ and enter start values for *a, b* and *c*. You can also set the number of iterations to be run by entering a value for **Max iterations**.



The parameters that are fitted (e.g. a_0, a_1,…) and the mean squared error of the fit (mse) are listed in the **derived Parameters** tab in the following representation: curve_fit<*modelfunction*>(Parameter).a0, curve_fit<*modelfunction*>(Parameter).a1, …, curve_fit<*modelfunction*>(Parameter).mse.

3.12.2.3 Derived parameters



The **Define Kinetic Parameters ▸ Derived Parameters** tab allows to perform calculations with the parameters listed in the **Definitions** list described above by the use of basic algebraic expressions (+, -, ×, ÷, sqrt, ^ etc. For a complete list, see Appendix 7.5).

Original Parameters. This is the list as created on the **Original Parameters** tab and includes also the fitted parameters when the operator **curve_fitting** is applied.

New. Click here to create a new list entry.

Name. Set the name of the new **Derived parameter** here.

Formula. Set the formula here using the **ID** of the original parameter(s) and the algebraic expression.

Example.

Imagine you want to measure the change of cell size. The Original Parameters needed are thus the maximum (max) and the minimum (min) of the parameter Area. The formula for the Derived Parameter would thus be p8/p9 (assuming parameter definitions as shown in the figure), yielding a growing factor.

3.12.3 Trace viewer

Open the **Trace Viewer** by selecting **Kinetic ▸ Show Traces** in the main menu. Alternatively you can right-click on one detected object in the image while in the **Trace View** and select **Show Trace** to see the trace of this individual object. Set the number of displayed traces with the slider on the right. To display only objects of a certain gate that was previously defined in the **Trace View** this gate can be selected in the **Gate** drop-down list.

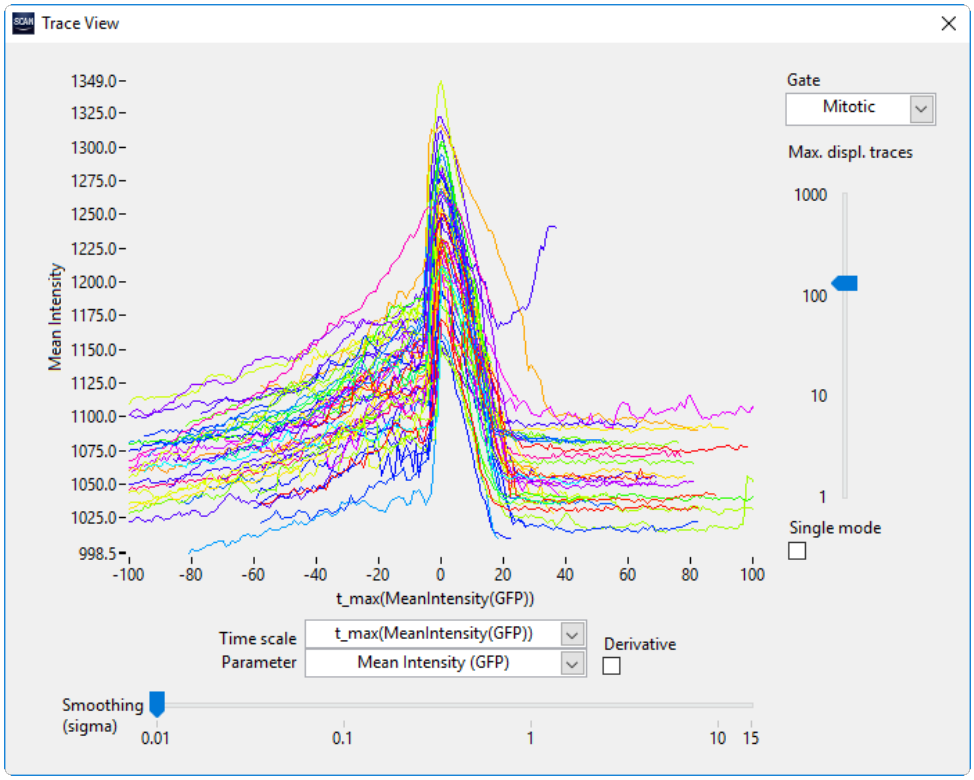
The results of the tracking are displayed in the **Trace Viewer** once the tracking has been executed.

Max displ. traces. Use the slider to adjust the number of displayed traces. The traces will be taken from a random selection of the specified gate.

Single mode. Use this option to have just one trace displayed. If a trace is selected via mouse click it will be this one that is being displayed.

Gate. Select a gate to have only the corresponding traces displayed.

Time Scale. *Absolute Time* is the default and causes that each point in time is set relative to the beginning of the time-lapse series. Each point in time is set relative to the beginning of each track when **t_first** is selected, which may be important if an object was not tracked right from the start of the time-lapse series and one is interested especially in the data at the beginning of each track. Alternatively, when applying the operator **t_max** e.g. on the parameter MeanIntensity(GFP), the time point when the mean intensity of GFP is maximal is set to be 0. This way the time curves can be displayed “synchronized”.



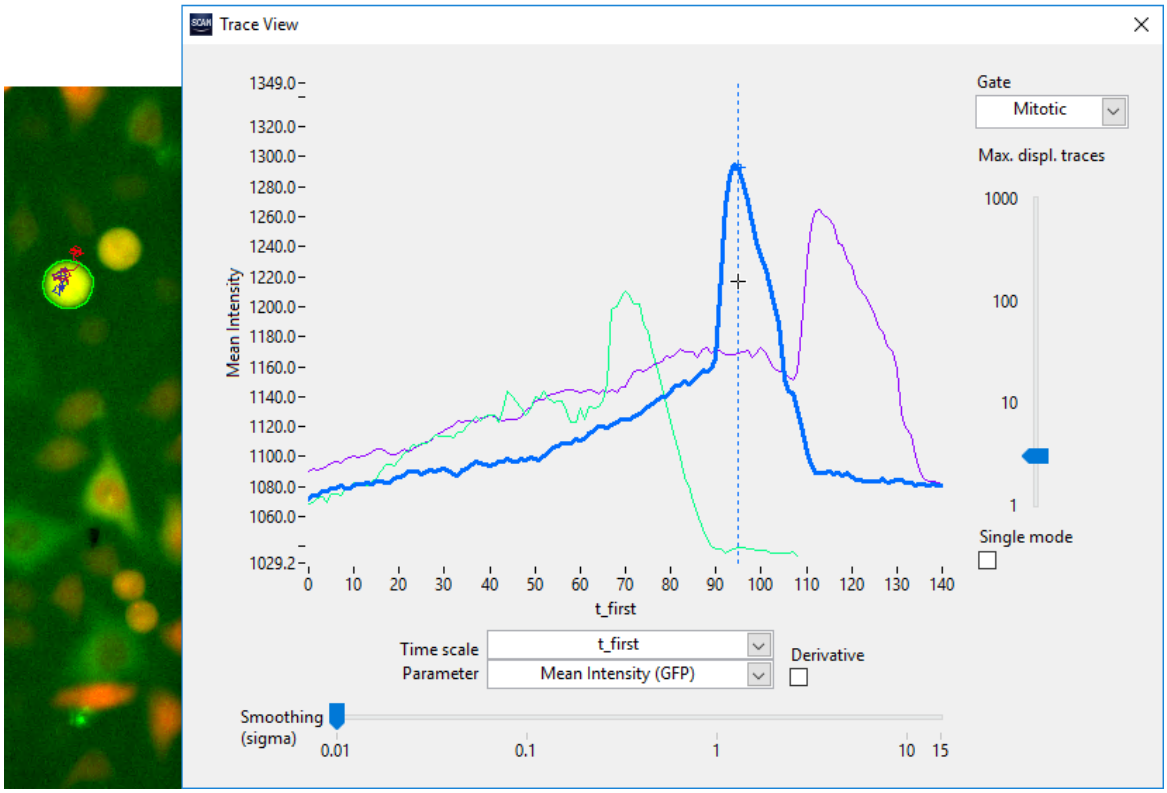
Parameter. Select parameter of which the time curve is to be shown in the graph display.

Derivative. Check this option to display the first derivative of the curve of the selected **Parameter**.

Smoothing (sigma). This function applies a smoothing filter on the .time curve. Set the strength of the smoothing with the slider. The graph display is being updated immediately.

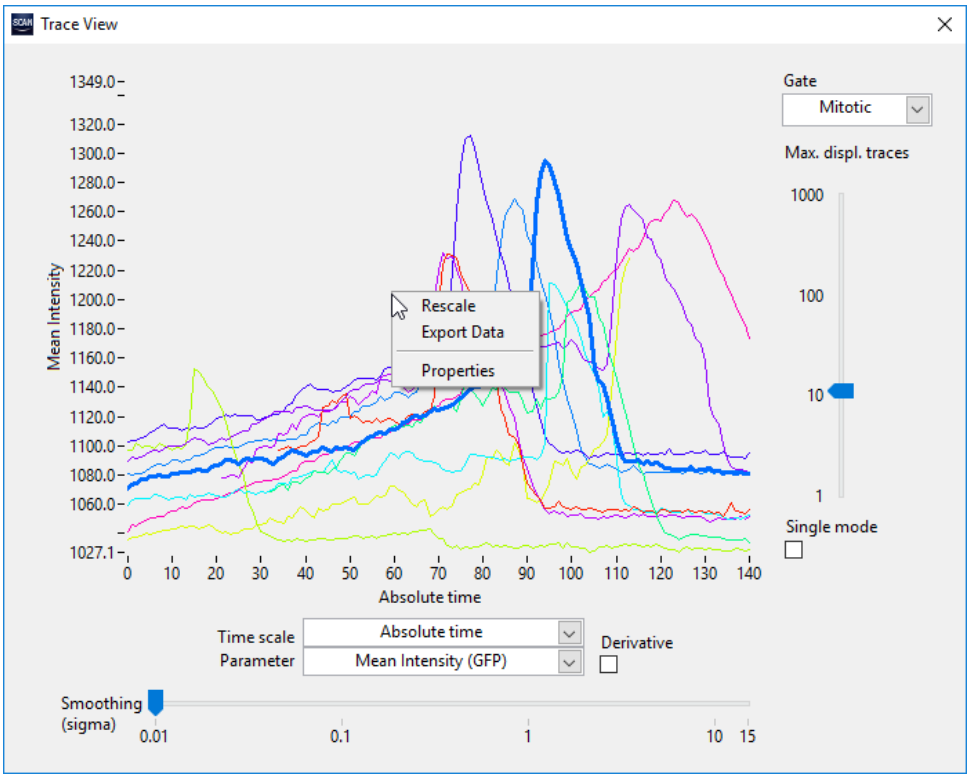
Selecting objects and navigating along tracks: Tracks in the **Trace View** and the objects in the images they derive from are directly linked. Upon clicking on a time curve the corresponding image with the marked object will be shown in the image display of the main scanR Analysis window. Also the data points marked with a red circle in the histograms are linked to the time curves. Likewise, upon clicking on an object in the image display or in a scatter plot the corresponding time curve will be shown in the Trace Viewer.

A dashed vertical line marks the current point in time in the Trace Viewer. Upon moving the cursor left and right while keeping the mouse clicked one can thus scroll back and forth through the "movie" of the time-lapse series.

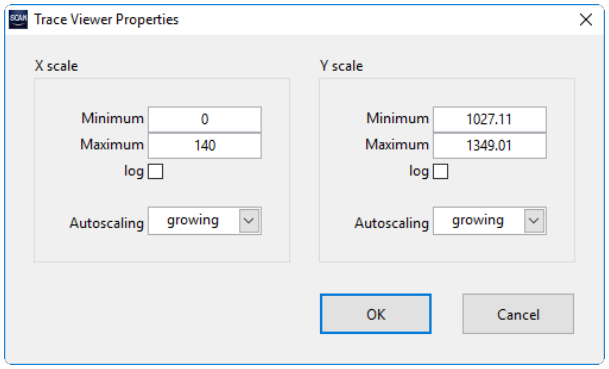


3.12.3.1 Set trace viewer scale

A right-click on the plot that is displayed in the **Trace Viewer** opens a context menu where you can select **Properties.**



The **Trace Viewer Properties** dialog opens. Here you can set the **X** and **Y scale** of the histograms.



Minimum: Sets the minimal value of the x- and y-axis, respectively.

Maximum: Sets the maximal value of the x- and y-axis, respectively.

Log: Check this box for logarithmic scaling of x- and y-axis, respectively.

Autoscaling: Choose between three autoscaling methods:

- **off**: no autoscaling. The scale remains fixed when different curves are displayed

- **growing**: the scale will increase if the current scale is not large enough to display all data points of the selected curve(s) but it will not decrease if the selected curves do not fill the complete histogram.
- **compact**: the scaling is optimized to display fit the selected curves optimal into the histogram, i.e. the display is chosen such that the max and min values of the curves are also the maximal and minimal values of the histogram.

A right-click in the **Trace Viewer** gives also the option **Rescale**. This option can be used to scale the currently selected curves optimally in the histogram without changing the settings of the autoscaling when further curves are added.

3.12.3.2 Export trace view data

Trace data can be exported by right-clicking in the Trace Viewer and selecting **Export Data**. This command opens a file explorer dialog where a location for an export file can be defined. A name for the file will be suggested which indicates the parameter for which it contains data. The file is a text file where multiple columns are separated by tabulators. While the first column contains the time index, the following columns contain the data of the trace(s) visible in the Trace Viewer.

4 Analysis Results

This chapter describes how an analysis is executed, how the results can be grouped into object populations and how these can then be analyzed and statistically evaluated.

4.1 Running an analysis

1. Execute **Scan ▸ Open…** and select the **experiment_descriptor.xml** file in the storage folder of the scan. By doing so the experimental settings are loaded and the first image of the scan (taken at the first position of the first well) will be displayed. If you want to modify or revisit an analysis execute **Analysis ▸ Open…** instead.

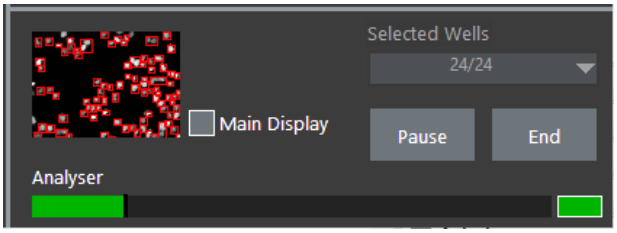
2. Execute **Analysis ▸ Load Assay…** to read in a ***.say** assay file stored in the **scan^R Analysis/Assays** folder. If you want to modify the assay file of the current analysis execute **Analysis ▸ Assay Settings…** instead.

3. Set or modify the parameters in the assay as described in Chapter 3, *Assays*

4. If you want to perform the analysis on a subset of the wells, open the **Plate** window via **Scan ▸ Plate…** and select the wells of interest.

5. In order to observe the analysis results online prepare the four histogram windows by selecting the objects and measurement parameters to be displayed.

6. Click the **Run** button (or execute **Analysis ▸ Run**) to start the analysis. You can follow the progress of the analysis in the window at the bottom right of the main scan^R Analysis interface.



The window shows the **Analysis Progress** and displays the image that is currently being analyzed in the thumbnail display. Click the **Main Display** button to have this image displayed in the main image window. You may **Pause** or **End** the analysis by clicking the respective buttons.

7. Once the analysis is completed store the results via **Analysis ▸ Save as…**

4.2 Managing gates

scanR Analysis detects objects in all images acquired during a scan and performs measurements and analyses on each object found. The data can be displayed in form of histograms where each color-coded data point represents the results of one object – or of several objects that happen to have identical results. The results – and thus the objects – can be grouped by **Gates** and **Regions**. The Gates allow classifying the objects according to their properties, i.e. the parameters that were extracted for these objects. Once these gates are defined, they can be applied to all further measurements for automatic quantification. The gates are administered in the **Gate Manager**, here you find also access to the **Well Results** dialog which contains the detailed results for all wells.

4.2.1 Gates and regions

scanR distinguishes between **Gates** and **Regions**. **Regions** define classification rules in form of polygons or ranges in histograms. **Gates** are composed of one such region or of several regions that are linked with Boolean operators (AND, OR, AND NOT). Once a **Gate** is set, only data points within its boundary are being considered in further steps.

Gates and **Regions** can be created using the drawing tool beside each scatter plot. A **Region** can be defined as **Gate** by selecting **Set Gate** in the histogram context menu or by defining a gate in the **Gate Manager**. (See Chapter 2.4, *Managing histograms and scatter plots.*)

By clicking on the borderline the Region can be displaced. By clicking one corner of the region this corner can be moved in order to include or exclude data points.

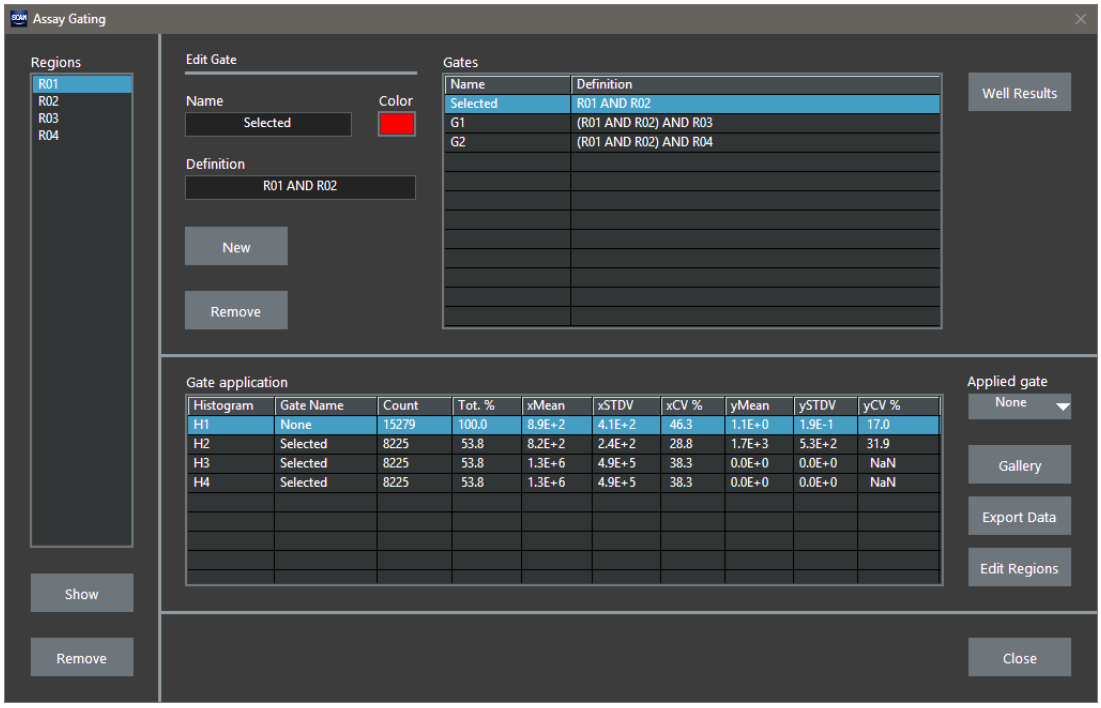
Right-clicking on the borderline of a Region opens the **Region** context menu. (See Chapter 2.4.3, *The region context menu*).

4.2.2 The gate manager

The command **Analysis ▸ Assay Gating** opens the **Gate Manager**, which allows converting **Regions** into Gates and to combine Regions by Boolean operations. The **Gate Manager** also gives an overview of the results in the histograms (H1 to H4) and the applied Gates.

Regions. This lists all regions drawn into the histograms.

Show. Click here to display the selected **Region** in the histogram selected in the **Gate application** list.



Remove (Region Box). This command removes the selected entry from the **Regions** list and deletes the defined Region.

Gates. It lists all gates with their **Names** and **Definitions**.

Name. Set the name of a Gate.

Color. Click into the box to open the dialog window to select the color of the object boundaries when displayed in the image viewer. This color is also used for color gating. (Compare Chapter 2.4, *Managing histograms and scatter plots*.)

Definition. A **Region** or a Boolean combination (via “AND”, “OR” or “AND NOT”) of **Regions** can be set to define the **Gate**.

New. Click here to add a new **Gate**. The default **Name** and default **Definition** will be that of a region in the **Regions** list. The **Name** can be modified at will; the definition must be a region or a logical combination of regions.

Remove. This command removes the selected entry from the **Gates** list.

Gate Application. This table lists all histograms with the applied **Gates** and statistical information about the gated data. (Number of Objects (Count), Percentage of total number of detected objects (% Tot.), Mean Value, STDV and CV% for the parameters plotted along the X and Y-axis)

Histogram. Number of the histogram window in the scan^{R} Analysis front panel, H1 to H4, respectively.

Gate Name. **Gate** applied to the histogram.

Count. Number of objects in the histogram.

%Tot. Amount of objects in percent of the total objects.

xMean (yMean). Mean value of the abscissa (ordinate) parameter of all objects in the histogram.

xCV (yCV). Standard deviation of the abscissa (ordinate) parameter of all objects in the histogram

xCV% (yCV%). Coefficient of variation of the abscissa (ordinate) parameter of all objects in the histogram

Applied Gate. This command allows changing the applied gate in the selected histogram. The drop-down list lists all gates from the **Gates** table.

Gallery. This command generates an image gallery of all objects in the selected histogram. The number of images is by default limited to 100. If the histogram contains more objects the 100 objects closest to the center of gravity of the region will be shown by default. (See Chapter 2.2, Preferences.)

Export Data: This command exports the data of the selected histogram as tabulator delimited table in txt format.

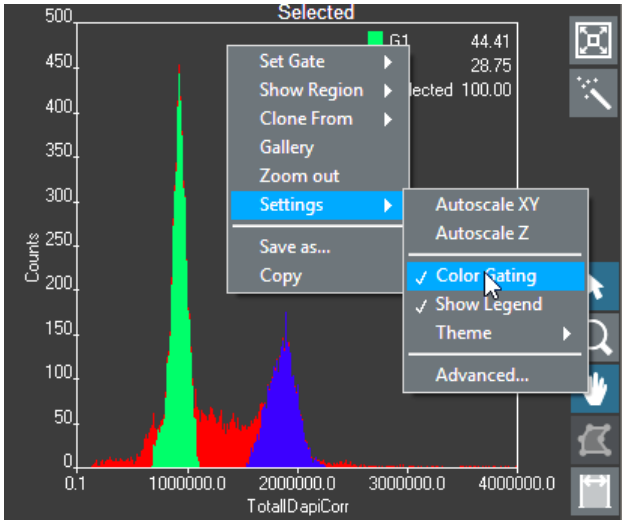
Edit Regions: You can select the region that you want to change numerically from the Region drop-down list.

Well Results. This opens the **Well Results** window, see Chapter 4.4, *Well results*.

4.2.3 Color gating

Color Gating shows different populations as defined by the gates in different colors in the one- and two-dimensional histograms. **Color Gating** is a property of each individual histogram displayed; thus, histograms with and without **Color Gating** can be displayed simultaneously in the different displays.

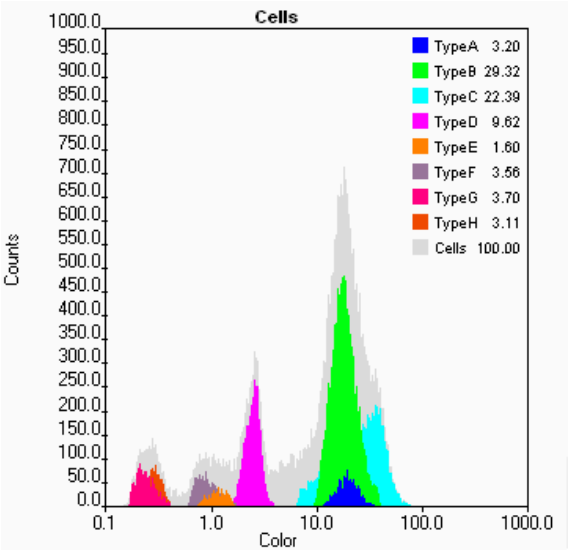
In order to activate **Color Gating** right-click in the histogram and from the context menu (right click) activate the option **Settings ▸ Color Gating.** The **Settings ▸ Show Legend** option activates the legend of the colors in the histogram. The population percentages of the displayed gates are also displayed.



The following rules apply to decide which color is shown in **Color Gating**:

- If there is only one object located on a pixel (Bin) the gating color of that object is shown.
- If there are multiple objects on a histogram pixel the gating color of the relative majority of the objects will be shown. Exception: If there is a gate applied to the histogram, that gate will be automatically in the background as 100% of the objects will belong to that Gate.

- Objects not belonging to a gate are shown in black.
- If there is no majority, the Gate Color order of the Gating list will be used (**Analysis ▸ Assay Gating…**; see Chapter 4.2.2, *The gate manager*). The "ranking" can be changed with the arrow buttons in that window.
- If a histogram is gated (context menu: **Set Gate**) the objects will be shown in the color of the sub-gate they belong to.



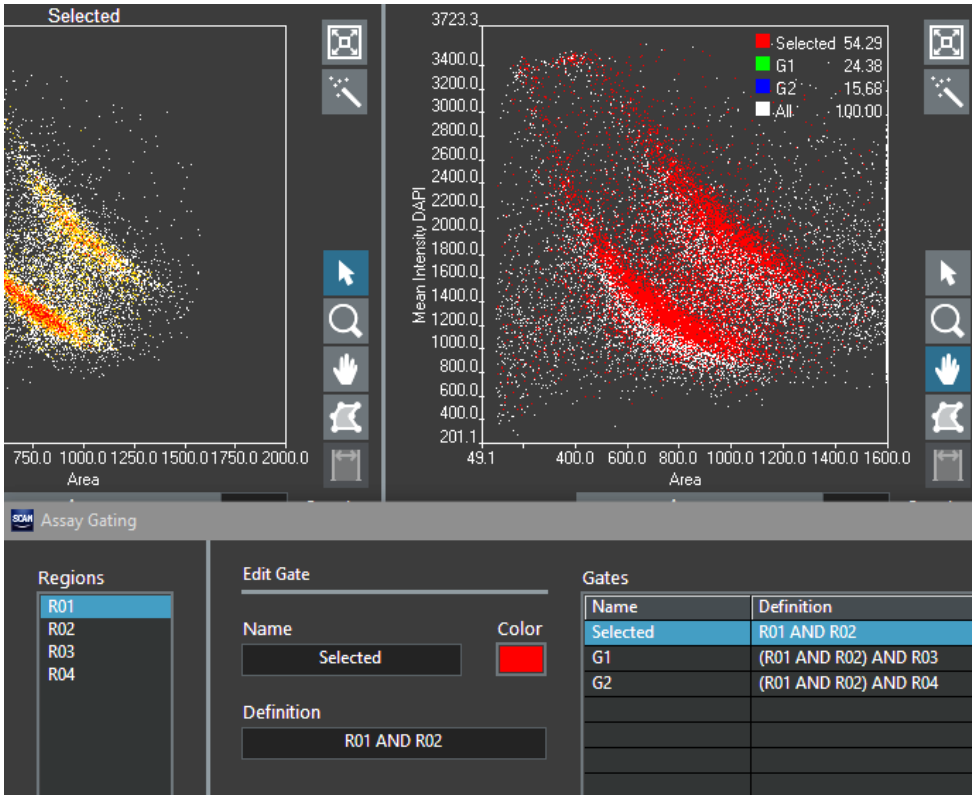
The order mechanism of Color Display (1-D):

- In 1-D histograms the applied Gate is always in the background.
- The order of display is organized by areas. If one of the Gates has less area above the other one it will be shown in the foreground.
- If **Auto Z Order** is disabled in the histogram settings (see Chapter 2.4.2, **The histogram properties** dialog), the Gate Color order of the Gating list will be used (**Analysis ▸ Assay Gating…**; see Chapter 4.2.2, *The gate manager*). The "ranking" can be changed with the arrow buttons in that window.

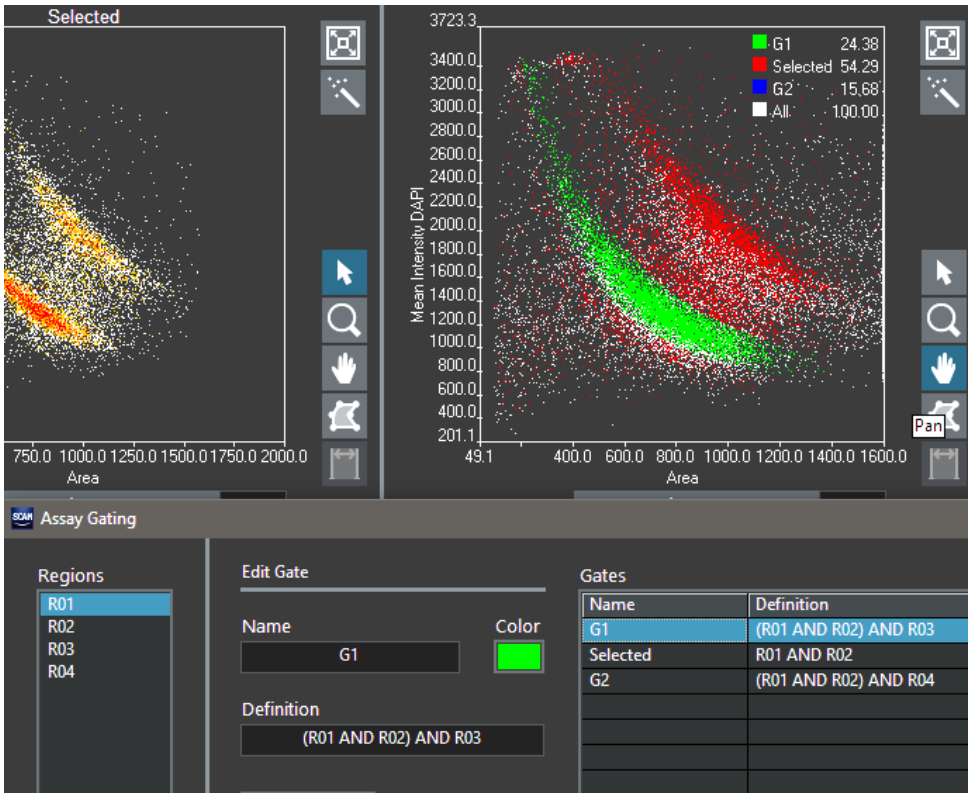
The selector line is position-sensitive in 1-D Histograms as well. Thus if you click on a green area you will get an object from the green Gate, if you click on gray you will get one from the gray Gate.

Examples:

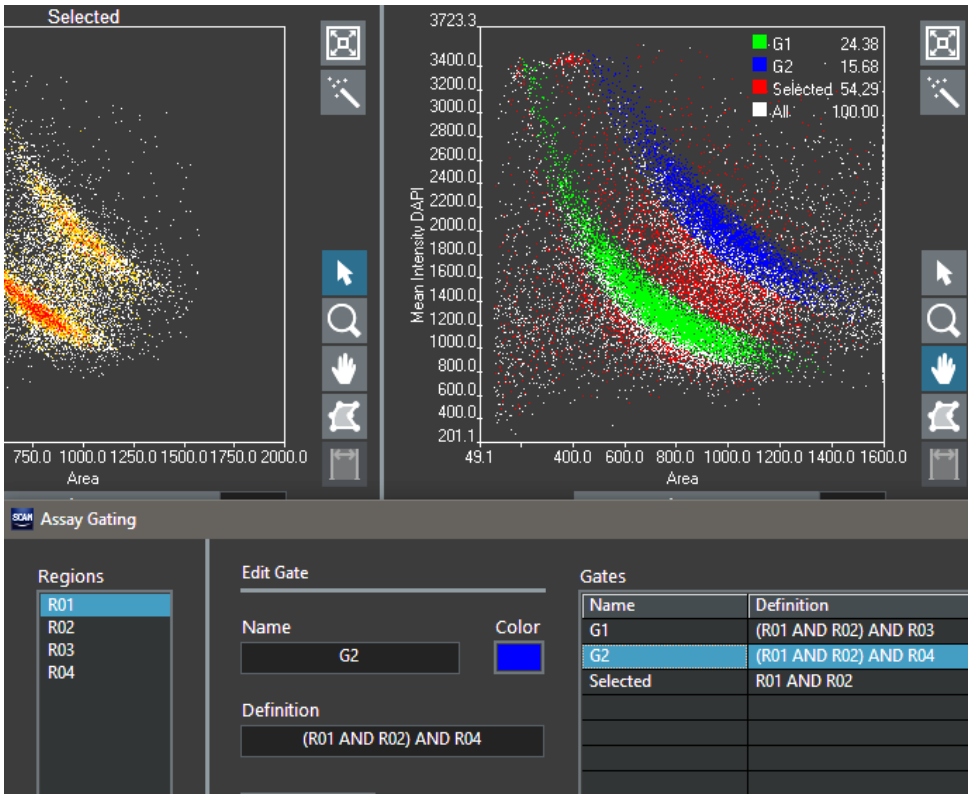
1. The example below shows a cell cycle with G1 and G2 defined as gates with different colors. The histogram is not gated, thus all detected objects are shown. Some of the pixels represent more than one object. In this case all pixels that have at least one object that belongs to the gate *Selected* are shown in red. Only the red color is shown because there is never a majority of G1 or G2 because G1 and G2 are subgroups of *Selected* and the order in the **Gate Manager** is *Selected* above G1 and G2. The other objects are shown in Black because they do not belong to any Gate.



2. In this example the situation is identical to the first example, but the order has been changed. All Pixels having a majority of G1 will now be shown in a green. All G1 objects also belong to *Selected* but when the same number of objects belongs to G1 and to *Selected* the color of the data point is determined by the list order and now G1 has a higher priority than *Selected*.



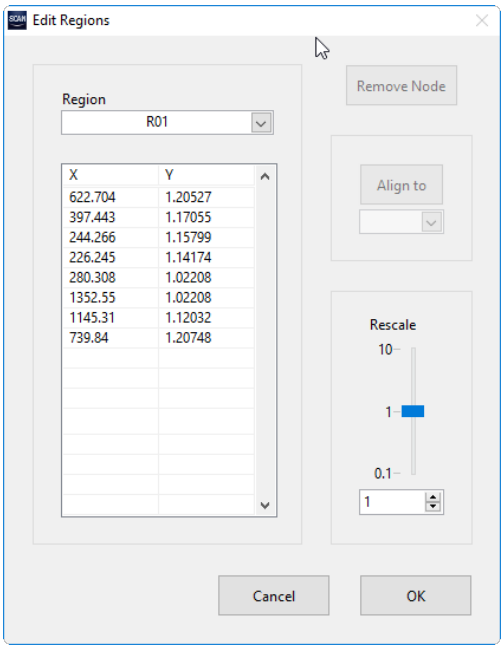
3. In the last example the order is changed again, but now also the gate *Selected* is applied to the histogram. Now the exception Rule 2) applies such that the *Selected* is not dominant.



4.3 Adjustment and alignment of regions

The regions in one histogram can be numerically adjusted and if multiple regions are created in one histogram, these regions can be aligned with respect to each other.

First draw a region in the first histogram by selecting the polygon tool. Close the region with a double-click. To numerically adjust the created polygon, go to **Analysis ▸ Assay Gating** and highlight in the **Gate application** list the histogram in which you have created the region. Then click on **Edit Regions**. The **Edit Regions** dialog opens.

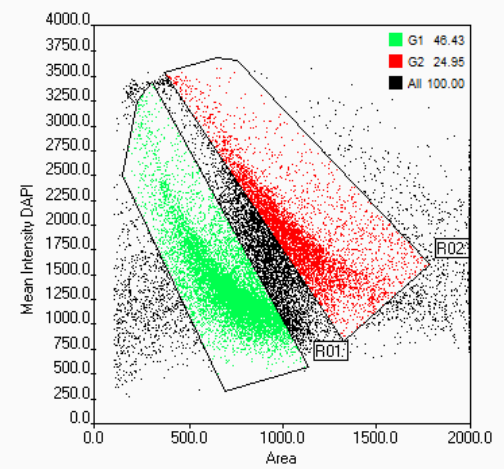


Edit Regions dialog

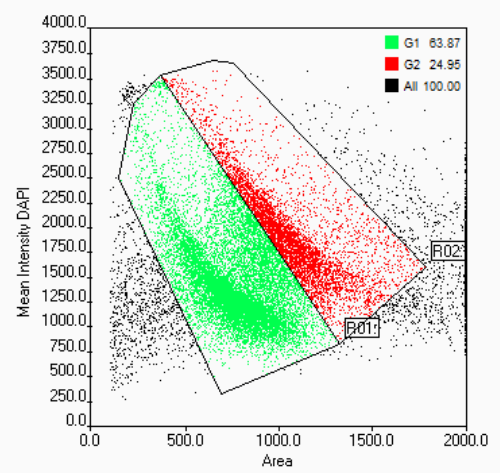
In the **Region** drop-down list you can select the region that you want to change numerically. The list will then show the x and y positions of the nodes of this region. You can change these values by entering numbers in the list. When you click on one of these values the **Remove Node** button will become accessible. With this button, single nodes of the polygon can be removed. Similarly, a new node can be created by entering the x and y values in a new line of the list.

With the **Rescale** slider or the corresponding numerical box below the slider, you can scale the polygon you have created. Values greater than 1 will increase the size of the polygon, values smaller than one will decrease the size of the polygon. The x and y values in the list will be adapted accordingly.

In the case that more than one region is created in one histogram, these Regions will become available in the **Region** drop-down list and the **Align to** button will become active. For example, the following regions are created.



Regions not aligned



Regions aligned

Now region R02 and region R03 will be aligned. In the **Edit regions** window Select R03 in the **Region** drop-down list and select **Align to** R02. Click on **Align to**. This will update the x, y coordinates of the Region R03. Click **OK** to leave the dialog. The Regions R02 and R03 will now be aligned.

> Aligning regions means that two regions will have one common, identical border. Since regions are handled as intervals excluding their end points, objects positioned exactly at the common regions' border might be contained in neither region.

4.4 Well results

The **Well Results** window shows the results of a complete scan analysis. It can be accessed from the Assay Gating window (**Analysis ▸ Assay Gating ▸ Well Results**), from the Plate window (**Scan ▸ Plate ▸ Well Results**), or using the main menu (**View ▸ Well Results)**. Statistical results of different parameters are listed for gated populations of the individual wells or of groups of wells and are displayed graphically in a histogram. Additionally, the results can be exported as txt-file for further analysis with other software.

4.4.1 Measurement results

The **Measurement Results** tab lists all the wells recorded in one measurement and displays the results (Number of Objects, Tot. %., Mean, Error, Error % and CV) of the measurement parameter which is selected in the drop-down list **Measurement**. The result for a combination of several wells is given in the bottom line when several wells are selected (press shift and click the lines you want to combine).The results can be exported by **Export Table**. The graph on the right gives a graphical representation of the results.

Well/Group results. This table gives a statistical analysis of the measurement results for each well or group of wells. It lists the mean value of a parameter for the gated population in each well or group of wells as well as its error and coefficient of variation.

Objects. Number of objects belonging to the population of the selected **Gate**

Tot %. Relative size of the **Gated** population

Mean. Mean value of the selected **Parameter**. It is shown in red in the histogram.

Error. Absolute error of **Mean**. It is shown in green in the bar plot.

Error %. Relative error

CV%. Coefficient of variation of Mean.

StdV. Standard deviation of **Mean**. It is shown in blue in the bar plot.

Object Type. Select the object type from the drop-down list (Main/Sub-objects).

Gate. Select a gate from the drop-down list in order to get the values for the corresponding gated population.

Measurement. Select the parameter of interest from the drop-down list.

Selected combined. Select a number of **Well/Group** entries (press shift and click in the lines you want to combine), their combined results will be given here.

Export Table. This function exports the data as tabulator delimited table in txt format.

Wells / Groups. Use this button to select if the values are to be given for individual wells or entire groups of wells. All wells that have the same **Name / Description** entry form a group. Wells that do not pertain to a group will be listed independently. The **Name / Description** is set in **Scan ▸ Plate** (see Chapter 2.7, *Selecting Wells for Analysis*).

4.4.2 Population results

In the **Populations** tab the number of objects (absolute and relative) in all defined gates are listed. A reference gate can be set to relate the number of objects in all other gates to the reference gate. The histogram on the right gives a graphical representation of the results.

Well/Group Results. The table gives the amount of objects in absolute numbers or as percentages that pertain to each of the **Gates** defined in the **Gates Manager.** (See Chapter 4.2.2, *The gate manager.)*

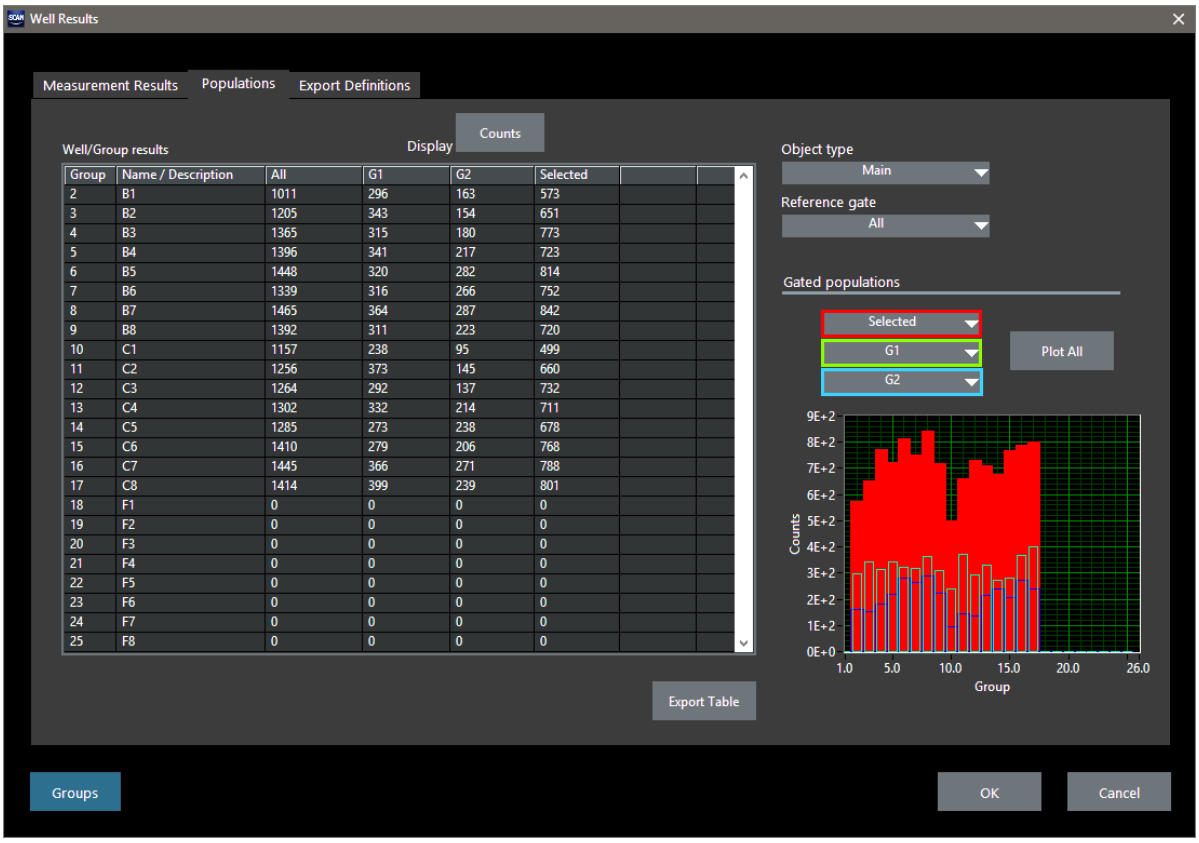
Display Counts / %. This toggle button switches between the listing of total numbers and the percentage of objects.

Object type. Select the object type from the pull-down list (Main/Sub-objects).

Reference gate. The population of the selected **Reference gate** is set relative to the populations of the other gates in the **Size comparison** table.

Export Table. This function exports the data as tabulator delimited table in txt format.

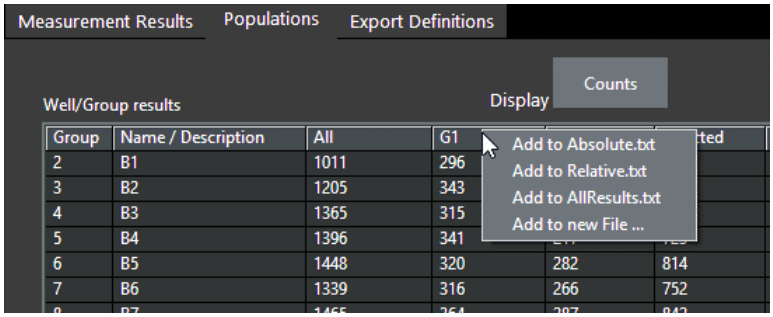
Wells / Groups. Use this button to select if the values are to be given for individual wells or entire groups of wells. All wells that have the same **Name / Description** entry form a group. Wells that do not pertain to a group will be listed independently. The **Name / Description** is set during acquisition already or after acquisition in **Scan ▸ Plate**. (See Chapter *2.7 Interactive plate result view.*)

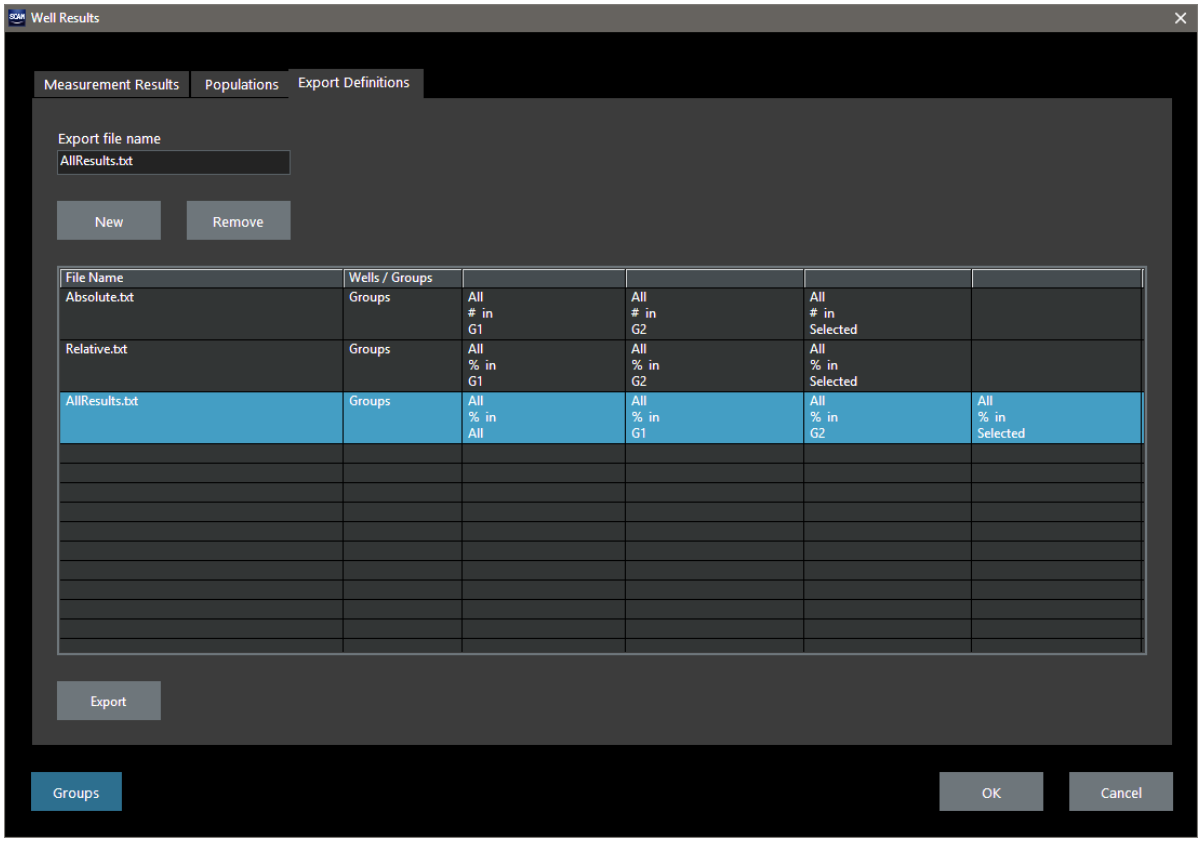


4.4.3 Export definitions

In the **Export Definitions** tab the parameters that are to be exported can be defined. This is especially useful for batch analysis and if not the complete results tables given in the **Population**s or the **Measurements** tab is to be exported.

Select **New** to create a new export file. The parameters to be exported have to be selected in the other tabs, i.e. in the P**opulations** tab or in the **Measurement Results** tab. When you right-click in these two tabs on the column names a context menu will open which allows you to export the selected column to a tab-delimited .txt-file.





Once the export files are defined, there are three different workflows for exporting the result data:

- Manual export. Press the **Export** button to select manually a folder to export the current results.
- Automatic export to scan folder. If no common **Results Export Directory** is defined in the **Preferences**, the result files will be written to a new subfolder “Population Results” at the same location where the scan is stored. This happens automatically whenever the analysis results are saved by **Analysis ▸ Save** or **Analysis ▸ Save as…**
- Automatic export to common results folder. If a common **Results Export Directory** is defined, the result files will be stored there whenever the analysis results are saved. The name of the analysis (*.sca) file will be added to the export file names in order to document the origin of the exported results.

New. Creates a new tab-delimited .txt-file for export.

Export file name. Enter the name for the txt file here. The default is Result0*.txt

Remove. Removes the selected File and its selection

Export. Allows setting set the folder where the .txt-files are stored manually. Otherwise the folder given in the **Preferences** is used.

Wells / Groups. Use this button to select if the values are to be given for individual wells or entire groups of wells. All wells that have the same **Name / Description** entry form a group. Wells that do not pertain to a group will be listed independently. The **Name / Description** is set in **Scan ▸ Plate**. (See Chapter *2.7 Interactive plate result view.)*

4.4.4 Export results of individual objects

Analysis ▸ Export Table: exports the parameter values determined for each detected object to a spread sheet. The values exported depend on the active view (**Population** view or **Trace** view). When also sub-objects are detected, not only one file is exported but for each sub-object a separate list is exported. For the “Population view” the list contains for example:

ParameterData_Main.txt

ObjectID	MeanIntensityDAPI	Subobject 1Counts	ParentObjectID	ParentTraceID	R01
0	174,52045	NaN	-1	0	0
1	295,50986	2	-1	1	1
2	220,76321	1	-1	2	1
3	161,89612	NaN	-1	3	0

ParameterData_Subobject 1.txt

ObjectID	Total Intensity GM130	Area	Parent Object ID	ParentTraceID	R01
0	201294	608	1	1	0
1	2504	12	1	1	0
2	2232	12	2	2	0
3	41191	108	4	4	0
4	4860	18	4	4	0

- Object ID. ID of the detected object
- Parameters. (e.g., MeanIntensityDAPI, Area,….)
- Derived Parameters.
- Parent Object ID. -1 if it is the main object; if not, the ID of the main object is given
- ParentTraceID. if tracking was performed the ID of the Trace the detected object belongs to is given. If no tracking was performed the Trace ID is set to -1.
- Gates. If the object belongs to the listed gate the value is set to 1, if not it is set to 0.

For the “Trace view” the list contains:

TraceID	lifetime	max(Area)	TrackedObjType	TraceLength	FirstParticleID	LastParticleID	R01
0	1	31	0	1	0	0	0
1	15	78	0	15	1	1940	0
2	8	626	0	8	2	1041	0
3	19	849	0	19	3	2640	0

- Trace ID. The ID of the Trace
- Parameters of the Trace (e.g. lifetime, max(Area),…)

- Derived Parameters.
- Traced Obj. Type. (0=main object; 1, 2, 3,…. Sub-object type)
- Trace Length. Number of objects in subsequent time frames belonging to the Trace
- First Particle ID. ID of the first object belonging to the Trace
- Last Particle ID. ID of the last object belonging to the Trace
- Gates. If the object belongs to the listed gate ate the value is set to 1, if not it is set to 0.

5 Example Assay – Step by Step

This chapter guides you step by step through the setup and execution of an assay and the analysis of the results.

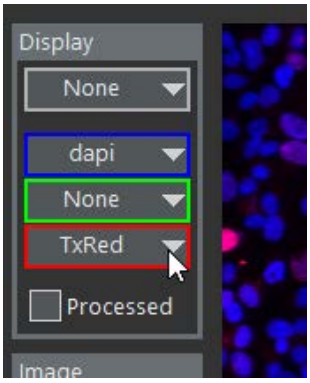
5.1 Setting up and executing an assay

The example scan which is used here to guide you through the setup and execution of an assay consists of images taken from cells labeled with DAPI to mark the nuclei and with FITC to mark the microtubules. A repair protein which is active in late G2 phase stained with TexasRed.

1. Execute **Scan ▸ Open…**, navigate to the folder where the demo data are stored, select the experiment_descriptor.xml file (scanR_Demo_Data_DVD/Slabicki_Buchholz_MPI-CBG/H2AX_PFA_002/experiment_descriptor.xml; sample data kindly provided by F. Buchholz, MPI-CBC, Dresden) and open it with **OK**.
   scan^{R} Analysis loads the first image of the scan, i.e., the image acquired at the first position of the first well is displayed in the main window of the font panel. Only the first channel of the image will be shown at first and in gray scale. Move the cursor over the image as a test and observe the status bar at the lower left. The channel intensity of each channel at the pixel positioned at the tip of the cursor arrow is shown in the status bar. The intensity values of the objects are usually much higher than the camera offset; the dark intensity of the camera.
   Memorize the intensity values of the brighter structures in the image for the subsequent display adjustment.

---

2. Set the grayscale selector to None, and load the dapi channel in the blue selector and the TxRed channel in the red selector.



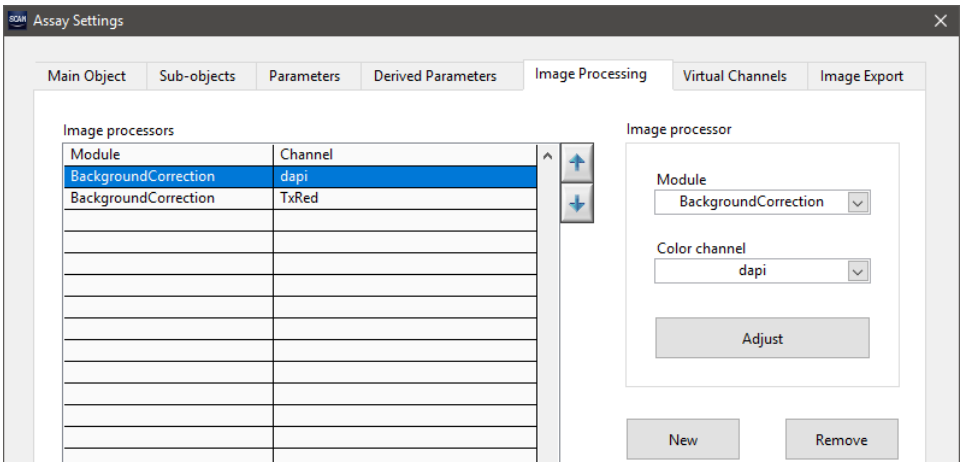
---

3. Execute **View ▸ Adjust Display** and adjust the display settings as explained in Chapter 2.5, *Using the Image Viewer*.

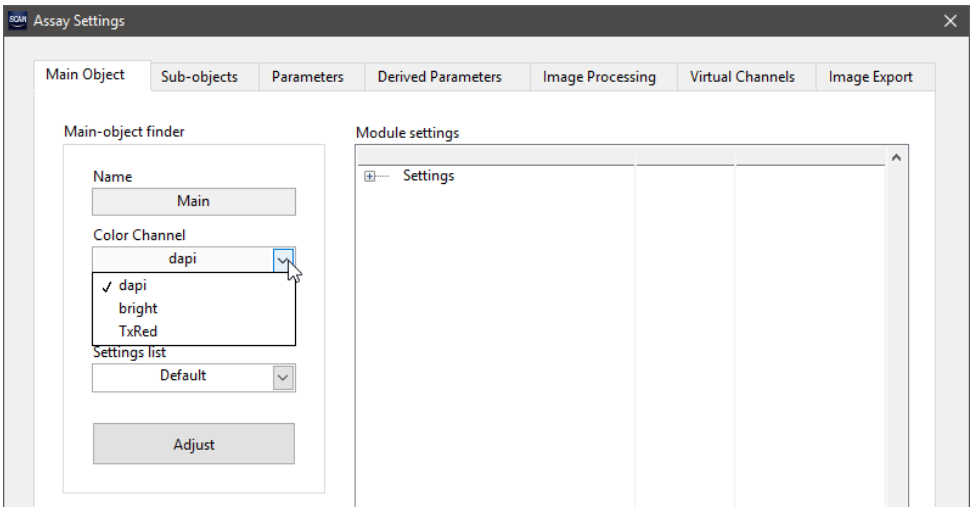
---

4. The first step in setting-up the assay is to find the best settings for the object detection. If you would have opened an old analysis file (*.sca), rather than the scan data only, the histograms would already show the data points of the analysis that was run before. In order to get these results from-scratch you first have to set up a new assay to perform a background correction, determine main (and sub-) objects, and the parameters you want to extract from the data.

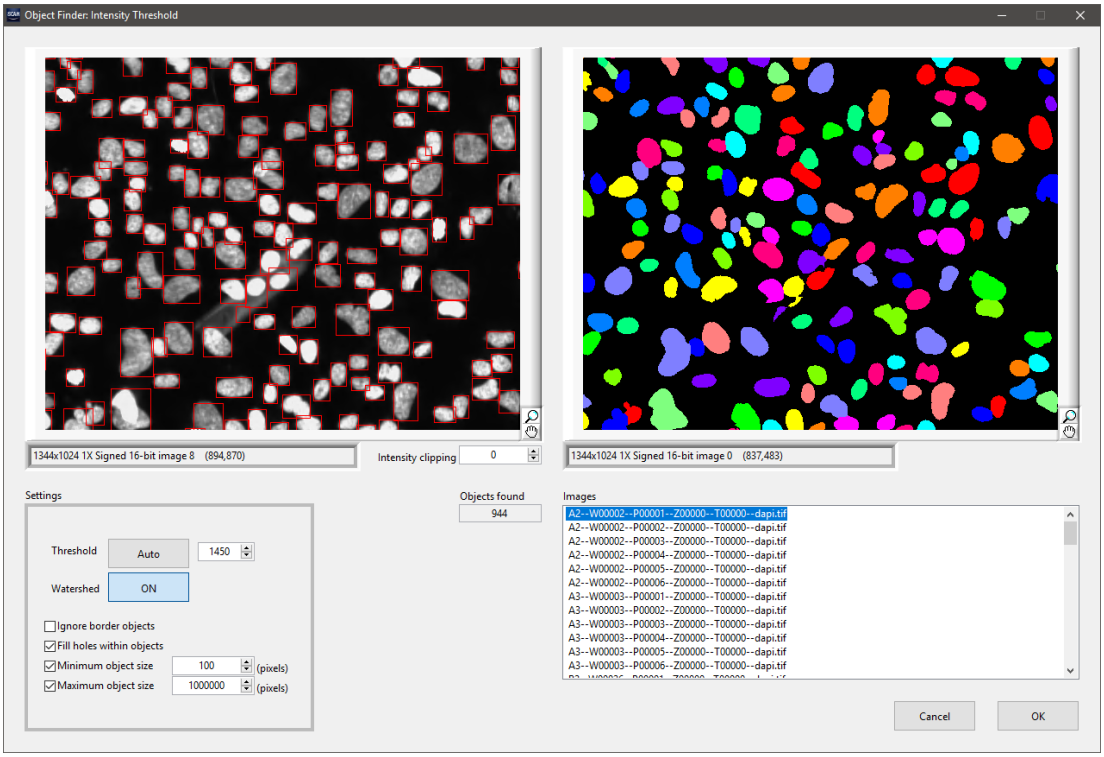
5. Execute **Analysis ▸ Assay Settings…** to open the Assay Settings window.
   The first step is to select the *Background Correction* **Image processor** Module on the **Image Processing** tab for each of the image channels. Press **Adjust** and observe in the new window the background in the image before (left) and after (right) background correction. When moving the mouse over the pixels the grey value should be reduced in the corrected images.



7. The next step is to determine the main objects. Therefore go to the **Main Object** tab in the **Assay Settings**. Here you have to select the channel in which you want to use for image segmentation. In many typical cases this will be the channel with the DAPI-labeled nuclei, the shape of which makes them rather ideal targets for object detection.



8. For image segmentation use the **Intensity** module. Both the **Edge** and the **Intensity** modules usually give good results when detecting nuclei. In other cases this is mostly a question of experience and testing. Click **Adjust** to open the **Object Finder** window. Here you can check and adjust the settings of the intensity detection algorithm. You will already get a good result by simply pressing the **Auto** button for determining the right threshold value and turn the **Watershed** algorithm **ON**. You can observe the effects of the parameters in the right panel where the result of the segmentation algorithm is shown.



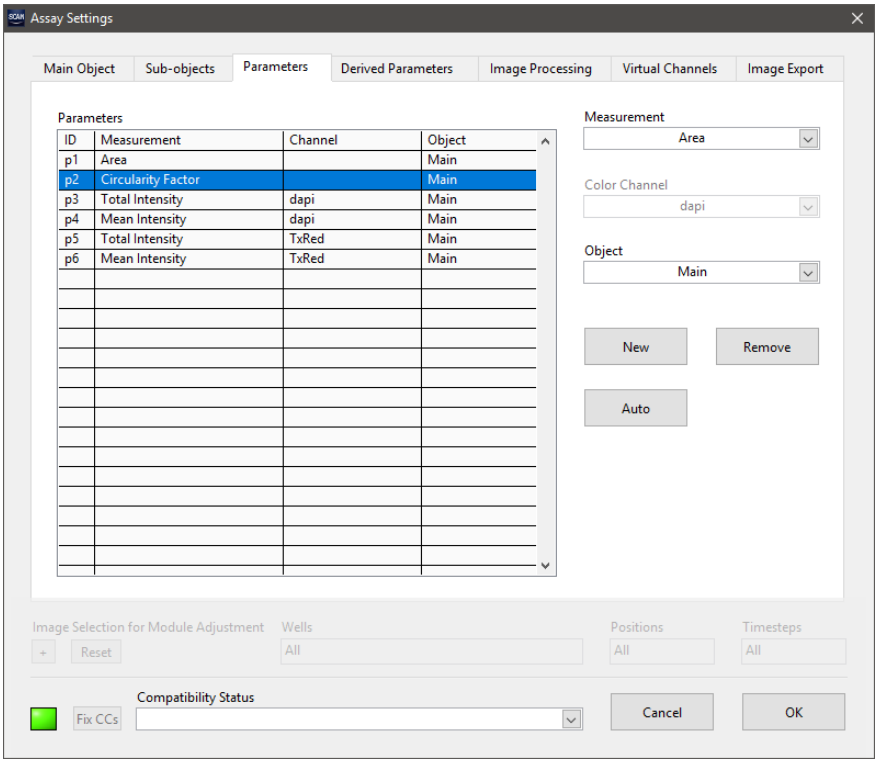
DAPI-labeled nuclei (Detected objects)

---

9. If **Sub-objects** are associated with the **Main Objects**, check and adjust their detection in the same way on the corresponding tab in the **Assay Settings** window. In this example skip the **Sub-objects** tab.

---

10. In the **Parameter** tab you can choose which parameters will be calculated during the analysis, i.e. you need to select the measurements that are to be carried out. Typical measurements are area, mean intensity and total intensity, and the circularity factor. Area (size) and circularity factor are extremely helpful as they allow to discriminate between more or less round, single nuclei and impurities or objects that consist of clustered, non-separable nuclei. The latter are usually much larger and less circular. If sub-objects are to be detected it is useful to add the parameter *Subobject. 1 Counts* to the list to determine how many sub-objects are connected to each main object. For this example, simply press the **Auto** button to add the most relevant parameters automatically to the list.



---

11. The **Derived Parameters** tab allows you to perform calculations on the previously defined parameters, e.g. here you can calculate ratios between different color channels. (For example you could evaluate the ratio “mean Intensity Dapi / mean Intensity TxRed”).

---

12. Now you have set up the analysis. Press **OK** to exit the **Assay Settings** dialog. You can save your assay with **Analysis ▸ Save Assay…**

---

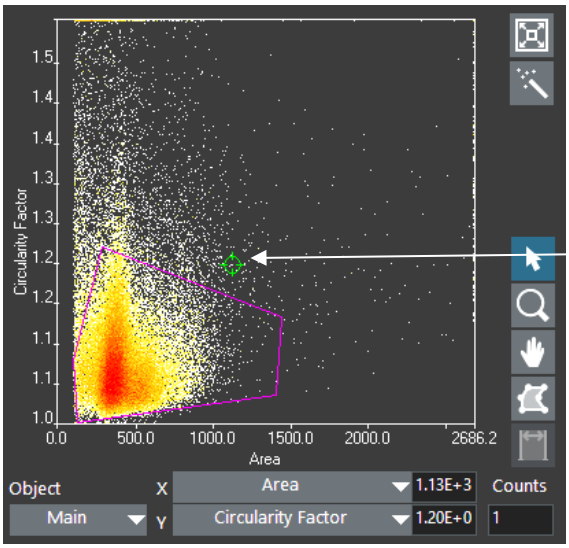
13. Start the analysis by clicking on the **Run** button.

---

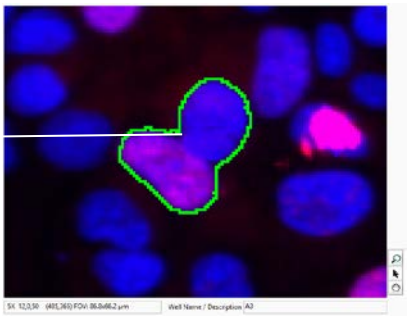
14. Prepare the histograms in the main window if you want to follow the analysis online. It is suitable to select the parameters **Area** and **Circularity Factor** in the first histogram and to display **Total intensity dapi** vs. **Mean intensity TxRed** in the second histogram. The small image window next to the main window shows the main object recognition in the image that is currently being analyzed. By clicking on the **Main Display** button you can display the current image in the large image window.

5.2 Analyzing the data

1. The first thing to do is to get an overview of the detected main objects, in our example the cell nuclei. The nuclei should all have a similar size and a more or less round shape, i.e. a Circularity Factor (CF) close to 1.0.
Set **Area** as X-axis parameter and **Circularity Factor** as Y-axis parameter and examine the size/shape distribution of object population in the scatter plots. The bulk of the population of the example has a CF of <1.1 and an area of around 300..400 pixels in images taken with 4x magnification.



Clustered cells
CF = 1.2
Area = 1130 pixels



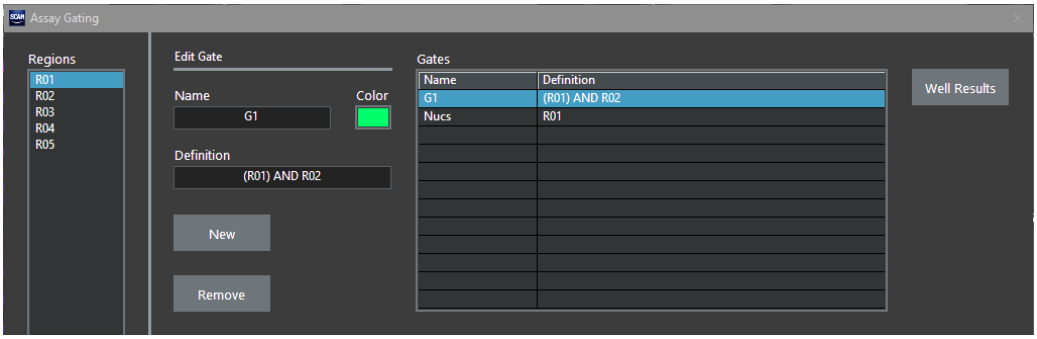
Objects that are larger and have a larger Circularity Factor may for example consist of clustered cells that could not be separated by the watershed algorithm.

---

2. For further analysis only consider those objects that do not differ significantly in size and shape from the bulk. To do so, use the **Region** tool and draw a polygon around the most densely populated part of the scatter plot. (Close the region by double clicking).
Right-click on the contour of **R01** in the scatter plot to open the context menu and select **Gate.** By doing so, the region **R01** is converted in a gate of the same name (**R01)** and you can apply this gate to all other histograms to exclude all objects outside this gate. Right-click again and select **Set Gate ▸ none** to get back all objects.
Right-click on the border of the polygon and select **Gallery** to display a selection of objects inside the gate; this allows you to roughly evaluate if your gating was set in a useful way.

---

3. In the next histogram display the **Total Intensity dapi** vs. the **Mean Intensity TxRed**. To include only the objects in **R01** right-click in the histogram and select **Set Gate ▸ R01.** You will observe a cluster around the *total Intensity dapi* of ~$9x10^5$ and another cluster around $1.8x10^6$. The left cluster consists of the nuclei in G1, whereas the cluster with twice the dapi intensity contains the nuclei in G2. (With G2 cells having twice the amount of DNA they will have twice the amount of DAPI-labeling and thus roughly twice the fluorescence intensity.) Draw a polygon around the cluster on the left with the **Region** tool and keep in mind the name (**R02**). Execute **Analysis ▸ Assay Gating** to open the **Gate Manager** window.



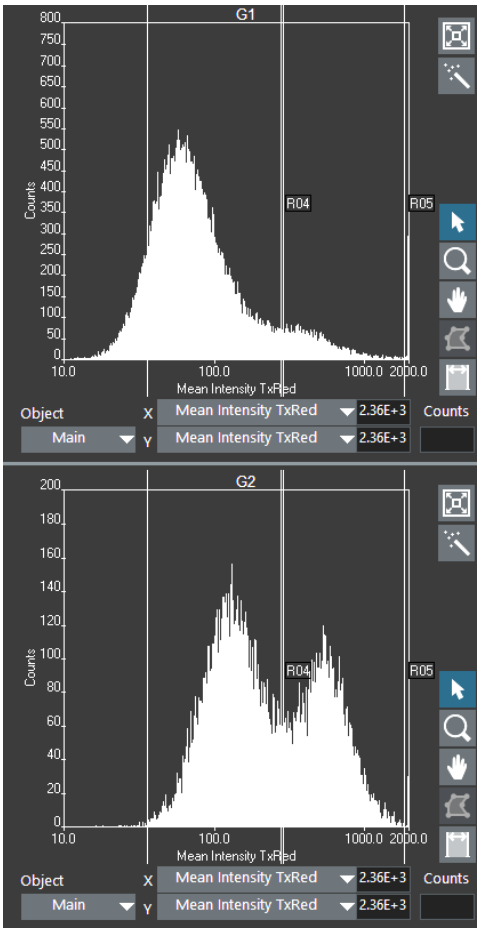
Click **new** to create a new gate. You can give the new gate a meaningful name (e.g. G1) and also select a color. In the Definition line enter **R01 AND R02** to select only the objects which are contained in both regions. Rename the Gate R01 to Nuclei; these are the objects defined in step 2. Then **Close** the Gate Manager.

---

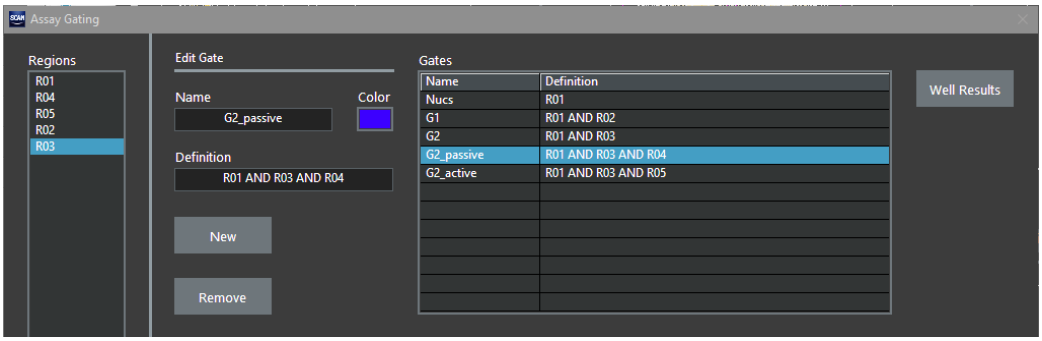
4. Next draw a region around the right cloud, i.e. the nuclei in G2 and repeat the steps described in step 3 and name this gate G2.

---

5. In the two remaining histograms choose **Mean Intensity TxRed** for the *X* and *Y* axis. This will give the distribution of objects with a certain amount of TexasRed staining in all nuclei. To find out if the distribution of TexasRed is different in the G1 and G2 phase right-click and select **Set Gate ▸ G1** in the context menu in one histogram and **Set Gate ▸ G2** in the other histogram.

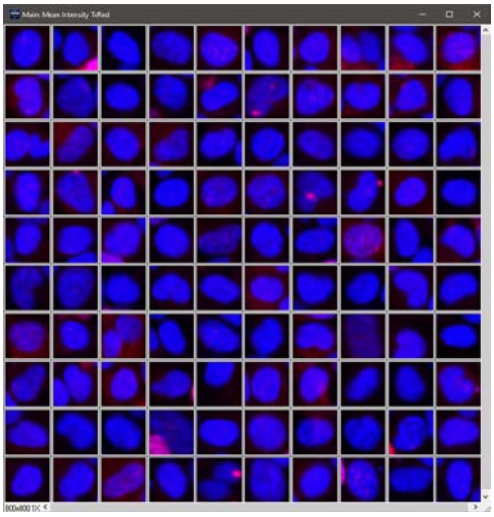


6. You will observe that the distributions in G1 and G2 look different. In G1 there are only nuclei with a low mean intensity in TexasRed, whereas in G2 there are two populations: one with a lower mean intensity in TexasRed and one with a higher intensity in TexasRed. Again select the polygon tool to define a region around the two populations in G2 and create two gates according to step 3. We name the nuclei with a high concentration of Texas Red G2_active, and the nuclei with a low concentration G2_passive. The result in the **Gate Manager** may look as follows:

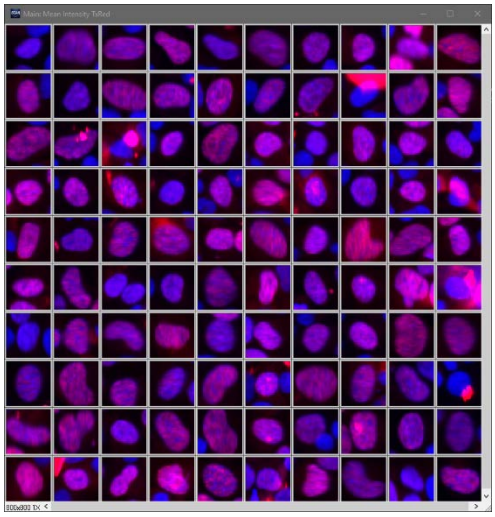


7. Leave again the **Gate Manager** window. By right-clicking on the main window you can highlight the nuclei in the different gates with boxes when you activate them (e.g. **Show Gates ▸ G2_active**). Another option to compare the results is to right-click on the border of

the gate and to choose **Region Gallery**. Plot a region gallery of G2-Active and G2-passive to compare the results of the gating.



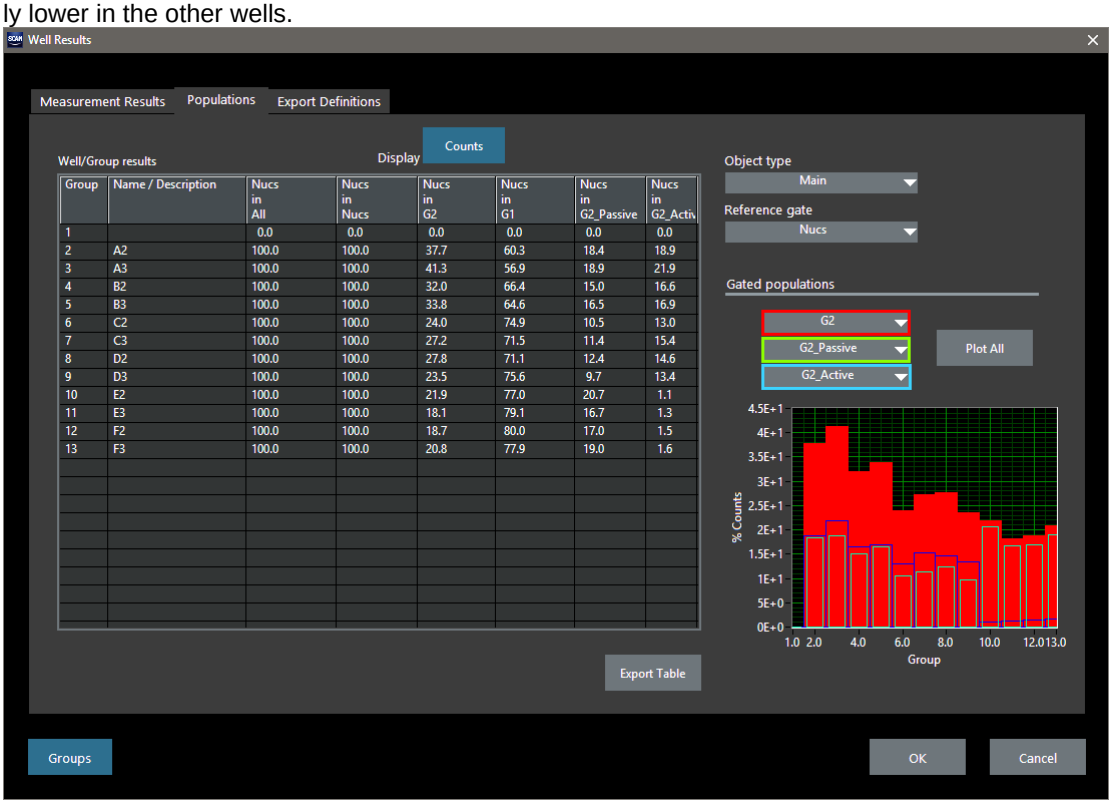
Gallery of G2-passive nuclei



Gallery of G2-active nuclei

---

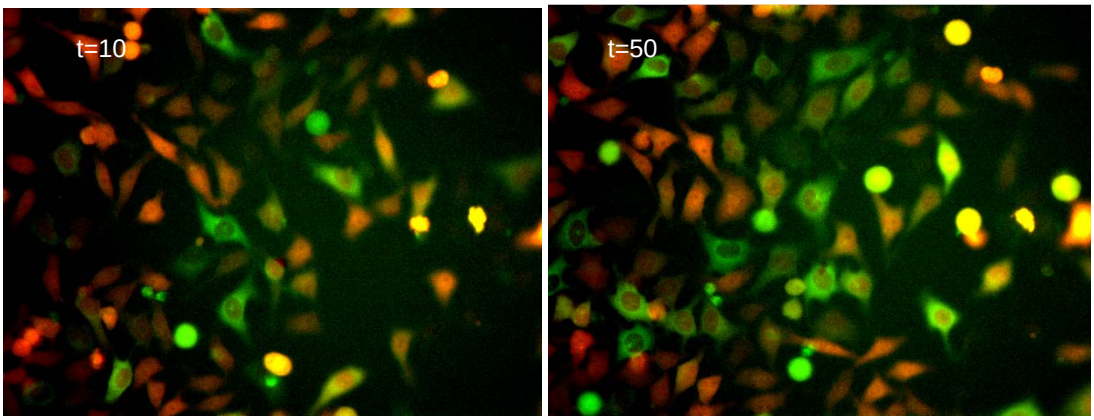
8. In the lower part of the **Gate Manager** you will also find statistics of the displayed histograms. For more detailed results click **Well Results** in the gate manager and open the tab **Populations**. Here you will find the results for all wells and all gates. You can view the display the total **counts** or the results as **percentages**. You can export these results for further analysis. A graphical display allows you to directly compare the results for different wells. Select G2 as reference gate and G2, G2_active and G2_passive as gated populations. You will notice that the fraction of active nuclei in G2 phase is very high in the central 4 wells while it is significantly lower in the other wells.



9. To export the results either select **Export Table** in the Populations or **Measurement Results** or select the results to be exported via **Export Definitions** as described in Chapter 4.4.3, *Export definitions*

5.3 Time-lapse analysis

In this example a time-lapse series is analyzed. The images of the example show cells during mitosis. The cells are marked with two stains (GFP and TxRed). The images left and right show images from the same well and position, taken at two different time points (t=10, t=50). In order to analyze time-lapse data it is necessary to segment the images properly and to track the moving cells from frame to frame.

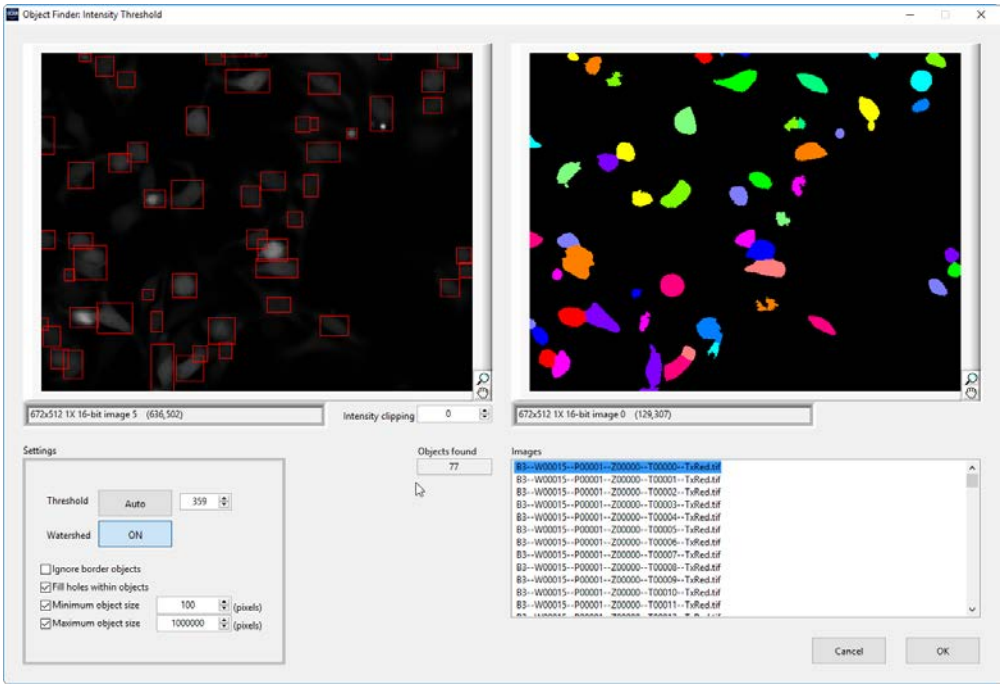


1. Execute **Scan ▸ Open…**, navigate to the folder where the demo data are stored, select the experiment_descriptor.xml file (scanR_Demo_Data_DVD/Slabicki_Buch-holz_MPI-CBG/CMV-mRed+cyclineB1GFP/GFP+mRed_001/experiment_descriptor.xml; sample data kindly provided by F. Buchholz, MPI-CBC, Dresden) and open it with **OK**.

2. Execute **Analysis ▸ Assay Settings** to open the **Assay Settings** window.

3. Go to the **Image processor** module in the **Image Processing** tab to set **Background Correction** for all channels (TxRed and GFP). Press Adjust and observe in the new window the background in the image before (left) and after (right) background correction. When moving the mouse over the pixels the grey value should be reduced in the corrected images.

4. Open the **Main Object** tab in the **Assay Settings**. Here you have to select the channel which you want to use for image segmentation. Select the TxRed channel (which has the stronger signal) for image segmentation and use the **intensity** module. Click on **Adjust**: The **Object Finder: Intensity Threshold** dialog opens. Press **Auto** to set the threshold and use the watershed algorithm.

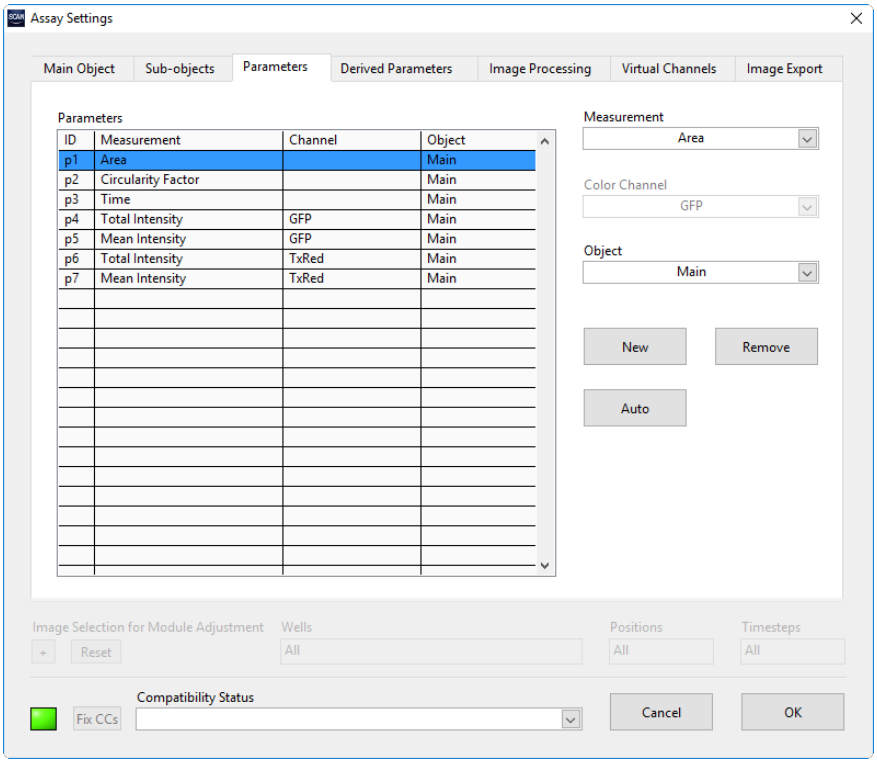


---

5. Skip the sub-object tab.

---

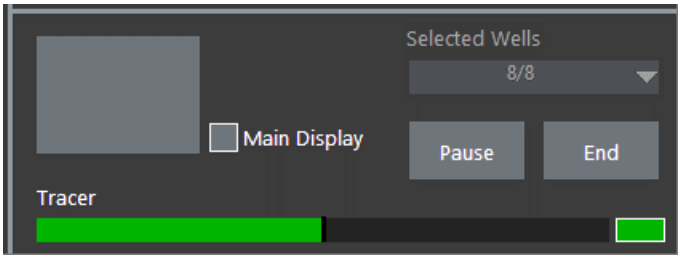
6. Go to the **Parameters** tab. Select here the parameters you want to detect for the main object. Select **Area** and **Circularity Factor**, which are determined on the channel of the main object (i.e. TxRed). For the GFP Channel select **Mean Intensity** and **Total Intensity**. Note that the intensities of the GFP channel are also detected on the main object (which was originally segmented using the TxRed channel).



7. Confirm the **Assay Settings** with **OK** and **Run** the analysis.

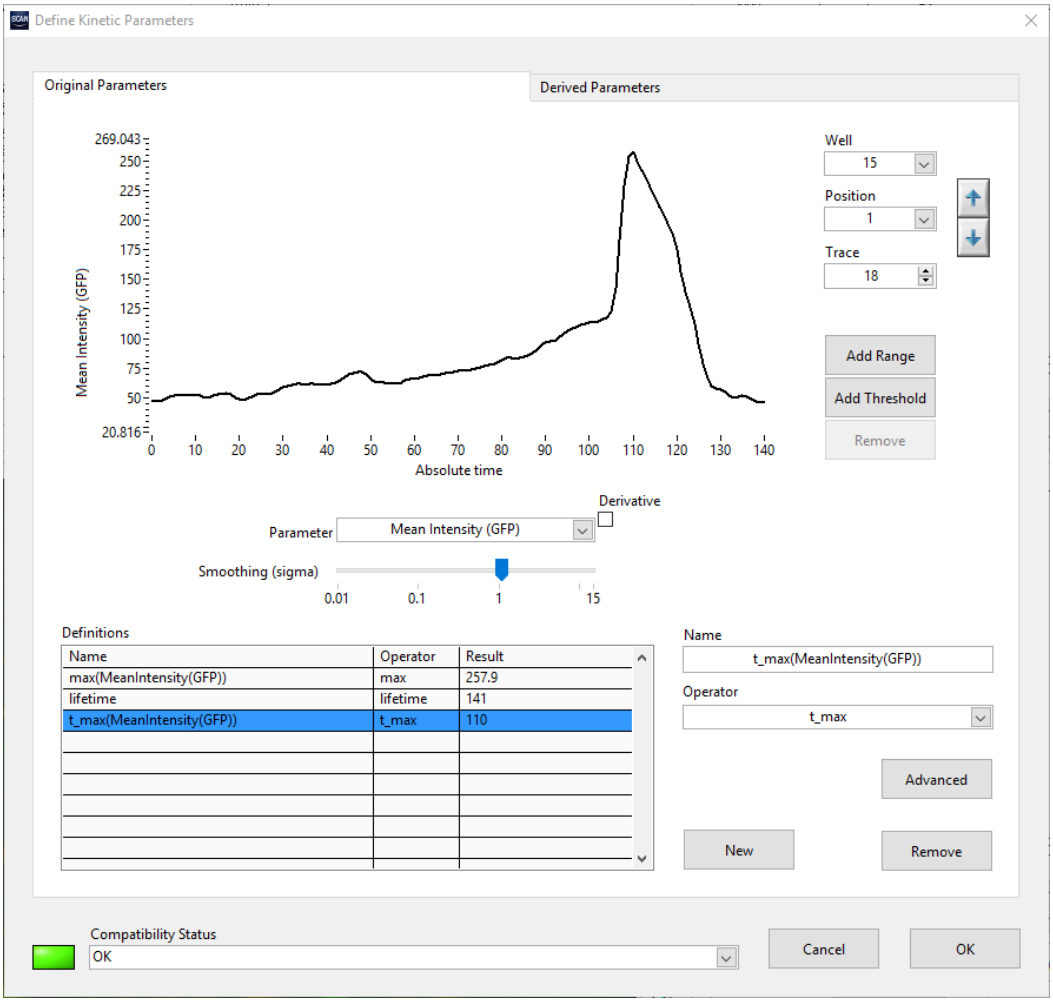
8. During and after the analysis you can plot the histograms and scatter plots as described in Chapter 5.2, *Analyzing the data*.

9. The next step after the image segmentation and the extraction of the parameters is the tracking of the objects. Go to **Kinetic** ▸ **Configure Tracer**. Check the **Enable Tracking** box and select **Main** as tracked object. Set a range of 20 pixels. Confirm with **OK**. When leaving this dialog the tracking will start. The progress of the tracking is shown in the status bar.



10. After the tracking is finished, the **Define Parameters** dialog is enabled. Here you can browse through the traces and determine the kinetic parameters that will be extracted from the curves. Select those **operators** which will be applied to the kinetic curves of the selected Parameters. In this example we want to find all the cells that undergo mitosis during the measurement. During the process of mitosis, the cells show an increase in the mean intensity of GFP. First set the smoothing to ~1 to obtain smoothed curves. In order to find out the mitotic cells in a simple approach set the following definitions: We need the **max(MeanIntensity(GFP))** to find out the maximum intensity of the curve. Then we need the

lifetime to gate on the cells that are tracked over all images. We are also interested in the time point when the maximum mean intensity is reached. Therefore select also **t_max(MeanIntensity(GFP))**

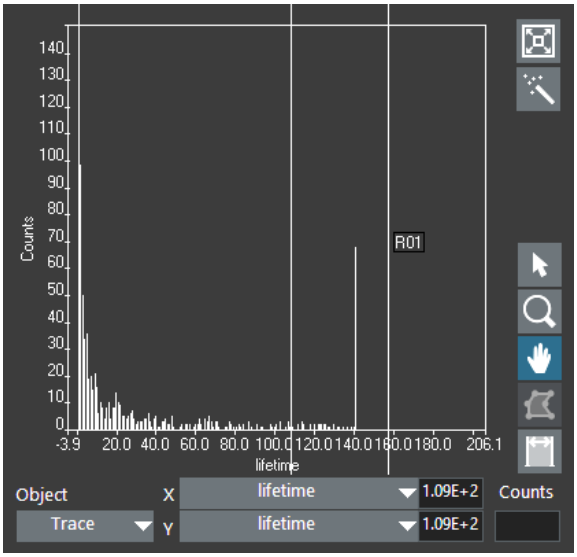


---

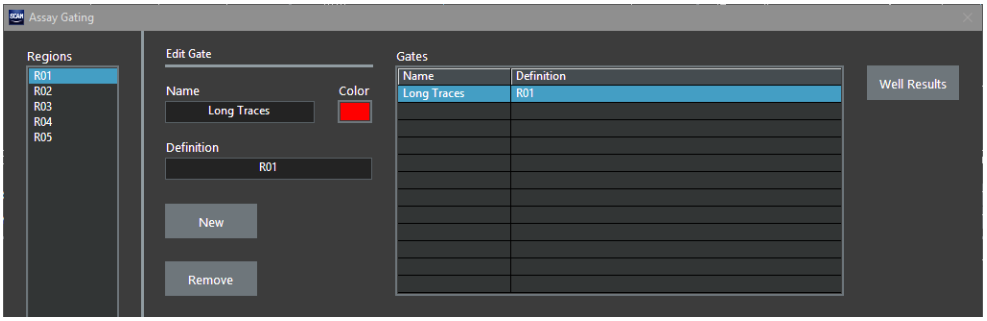
11. Close the dialog with **OK**. The analysis of the curves is performed. In the lower right of the front panel the status bar displays the progress of the analysis.

---

12. Switch to Trace View. When the analysis is finished you can plot a lifetime histogram in the first histogram panel. In order to select the cells that were detected in all timeframes draw a rectangle around the data points at 140. Right-click on the region border and select **Gate** to convert the region into a gate.

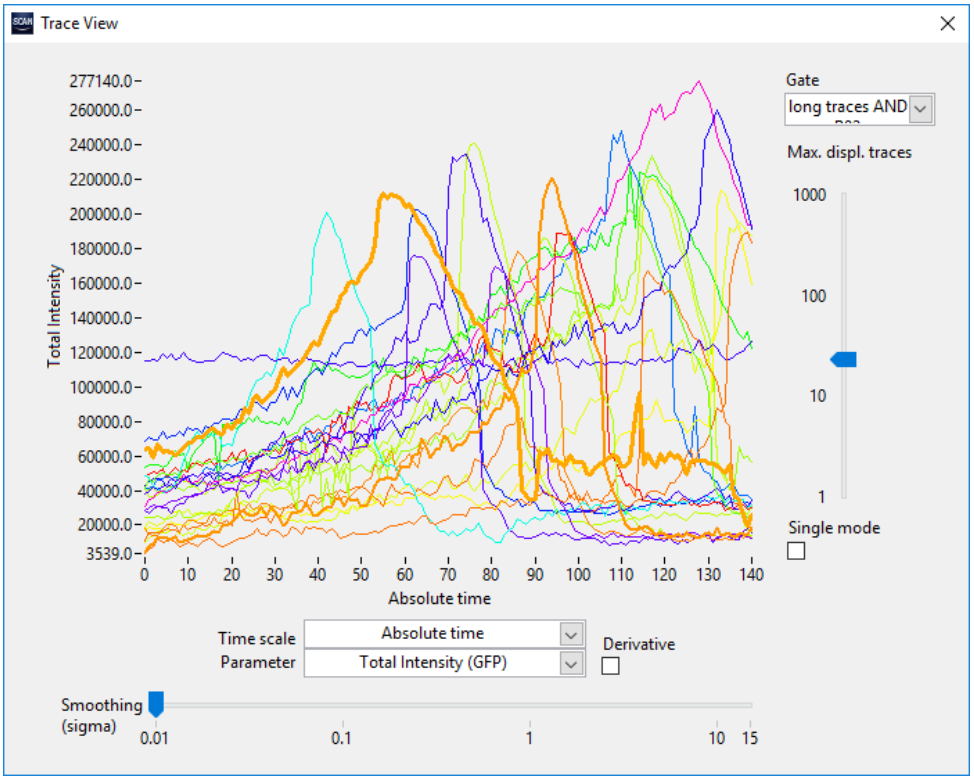


13. Go to **Analysis ▸ Assay Gating…** and rename the Gate **R01** to **longtraces**. Leave the dialog with **OK**.



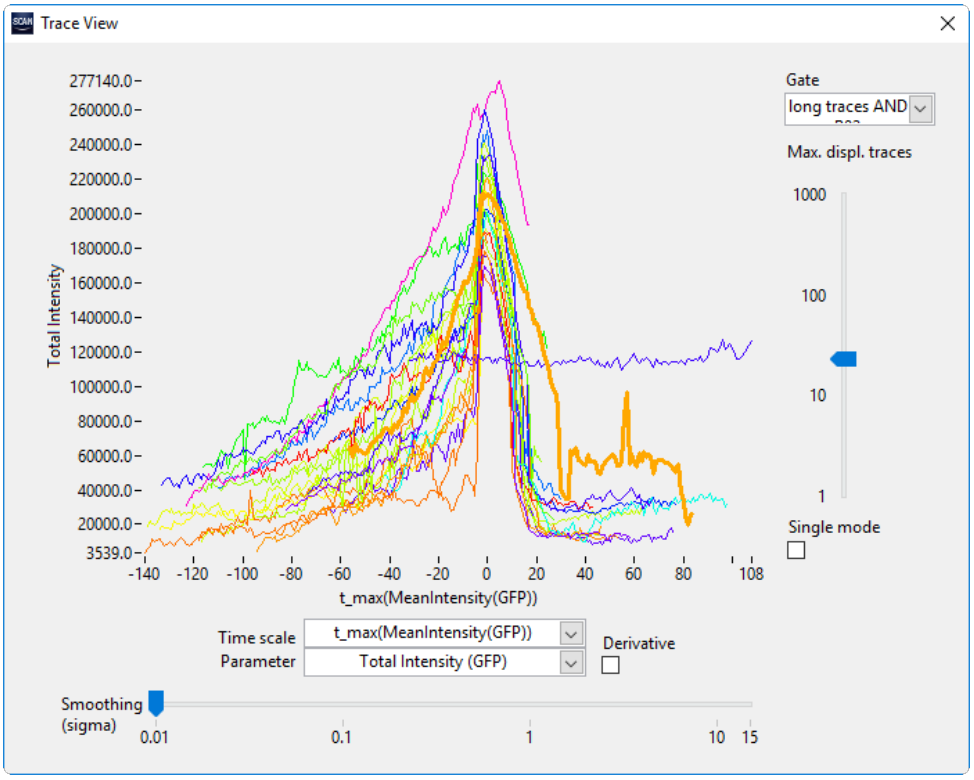
14. In the second histogram display, plot **max(MeanIntensity(GFP))**. Right-click in the histogram and select **Set Gate ▸ Longtraces** in order to include only the cells that were detected in all images into the analysis. Draw a rectangle around the datapoints on the right. These are the cells which show a high mean intensity of GFP. Right-click on the border of the rectangle and select **Gate**. The new gate **longtraces AND R02** is created.

15. Select one of the data points and the corresponding cell is shown in the image display with a green outline. Right-click on the image of the cell and select **show trace** from the context menu. The **Trace Viewer** opens and displays the curve of the selected cell. In order to show all the traces of the gated cells, deactivate **single mode** and for Gate select **longtraces AND R02**. The curves show all the same characteristics.

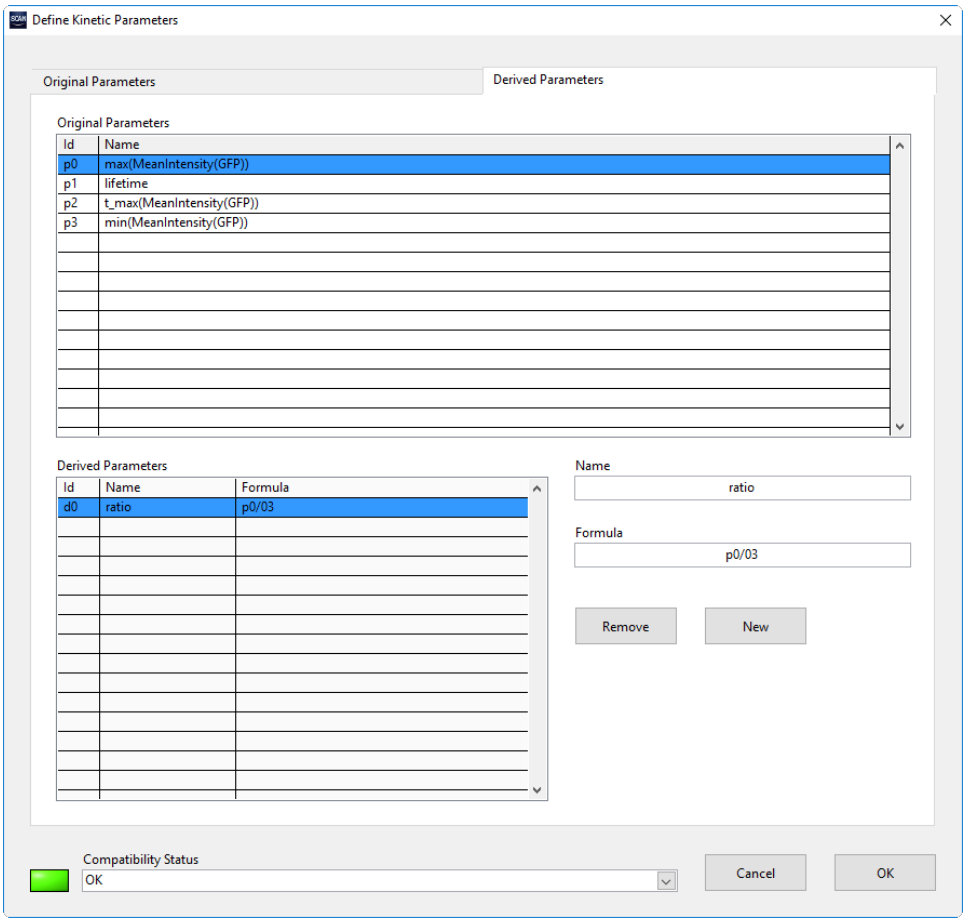


---

16. As default the **Time scale** is set to **Absolute time** but it is also possible to display and analyze the curves regarding to a certain timepoint, e.g. the time of mitosis. Therefore select **t_max**(MeanIntensity(GFP)) as **Time scale**.

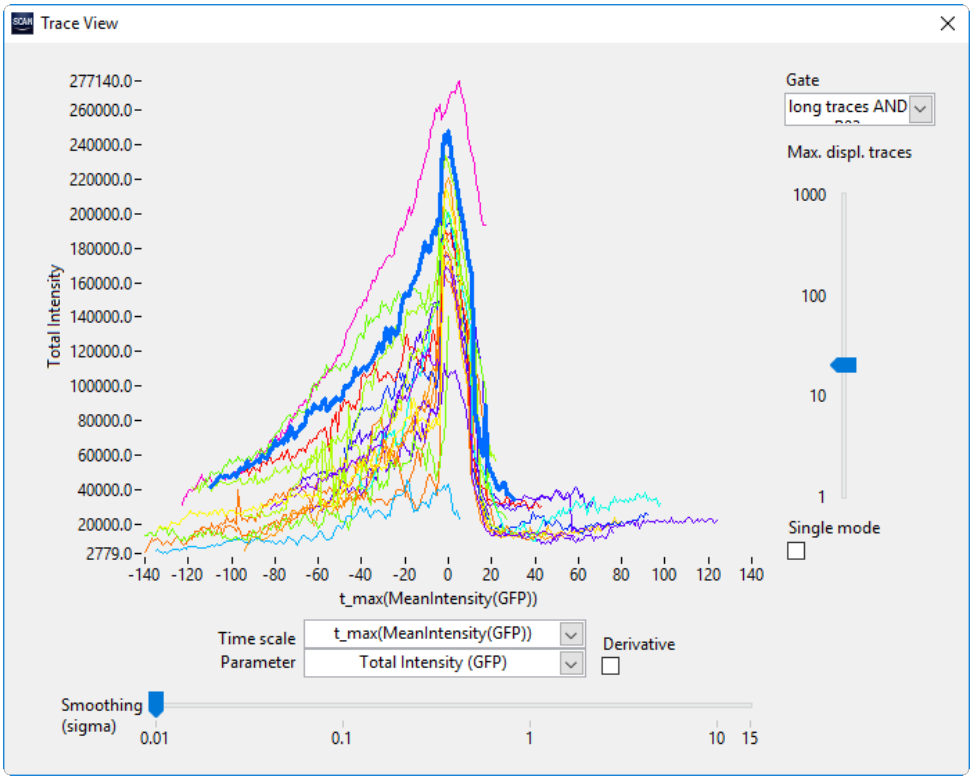


17. As you can see, there is one (blue) curve which shows no mitosis but has a high mean intensity in GFP all the time. (When you select the blue curve and observe the corresponding object in the images you see that this is probably an apoptotic cell). In order to improve the classification, you can set up a more advanced analysis, e.g. you can set the parameters **max(MeanIntensity(GFP))** and **min(MeanIntensity(GFP))** in the **Trace Parameters** dialog (therefore go back again to **Kinetic ▸ Define Parameters**). Go to the **Derived Parameters** tab In the **Trace Parameters** dialog and set as new derived parameter the ratio of **max**(MeanIntensity(GFP)) and **min**(MeanIntensity (GFP)). Name this new parameter **ratio**.



18. After running the trace analysis plot a histogram of **ratio.** Set again **longtraces** as gate. The data points around 1 show about the same minimum and maximum mean intensity of GFP. Whereas data points >1 are the ones that undergo mitosis and have a higher maximum mean intensity. Draw a rectangle around the data points with ratio >1. Right-click on the border of the rectangle and select **Gate**. The new gate **longtraces AND R03** is created.

19. Mark one of the data points and the corresponding cell is shown in the image display with a green outline. Right-click on the image of the cell and select **show trace**. In order to show all the traces of the gated cells, deactivate **single mode** and as gate select **longtraces AND R03**. You will see that this parameters lead to a better discrimination of mitotic cells than the more simple approach used before.


Trace View
277140.0
260000.0
240000.0
220000.0
200000.0
180000.0
160000.0
140000.0
120000.0
100000.0
80000.0
60000.0
40000.0
20000.0
2779.0
Total Intensity
-140 -120 -100 -80 -60 -40 -20 0 20 40 60 80 100 120 140
t_max(MeanIntensity(GFP))
Gate
long traces AND
Max. displ. traces
1000
100
10
1
Single mode
Time scale
t_max(MeanIntensity(GFP))
Parameter
Total Intensity (GFP)
Derivative
Smoothing (sigma)
0.01
0.1
1
10
15

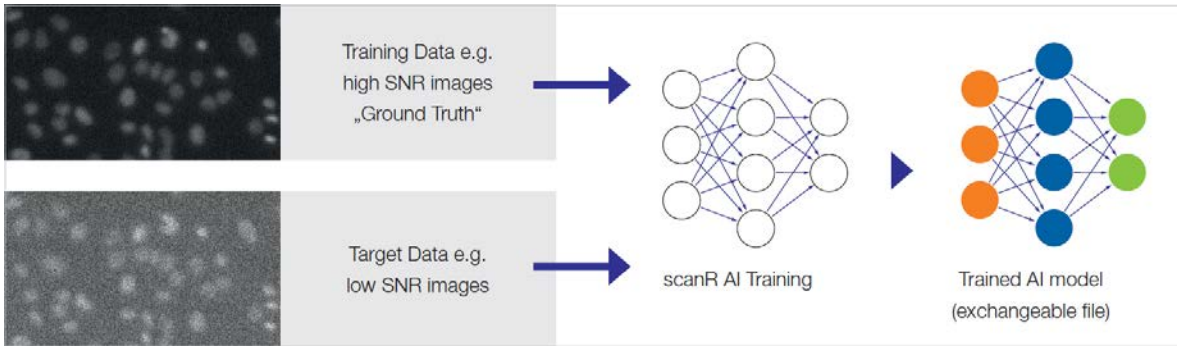
6 scan^{R} AI

This chapter introduces the concepts of self-learning microscopy. It shows how to train a deep learning AI model which can be used for advanced image analysis tasks, and how to apply trained models for image analysis and object classification.

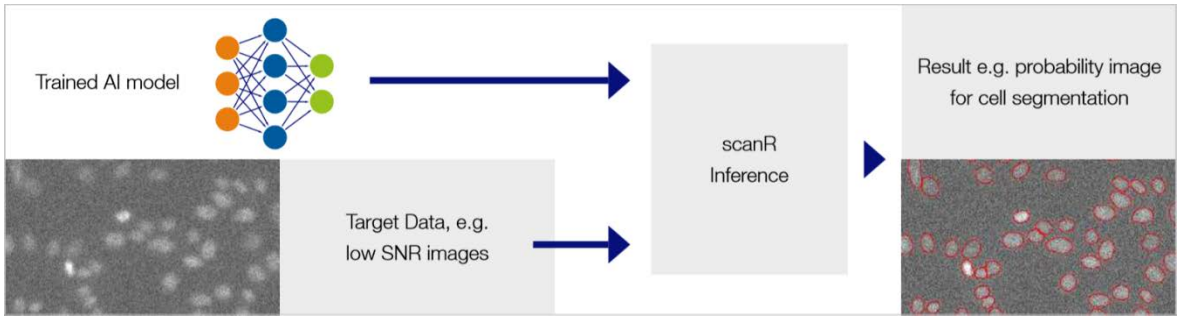
6.1 The general concept of self-learning microscopy

In self-learning microscopy, a neural network is trained without the need for manual annotations to detect particular features within images. In order to train the network, the features to be detected later by the network are once detected by other means, for example, by acquiring more image data only for the training. The extra data acquired only for the training is analyzed with conventional means, and features are detected. These detected features, or, more precisely, the segmentation masks, are used as "ground truth" to teach the neural network how to detect the same features in the actual data. Since the whole scan^R workflow is optimized for this kind of automatic data acquisition and analysis, the training phase can be executed almost without any work for the operator. Typically, large amounts of data are acquired automatically and used for training. This data amount makes it possible to train a neural network to achieve excellent detection performance and robustness even in the most challenging situations.

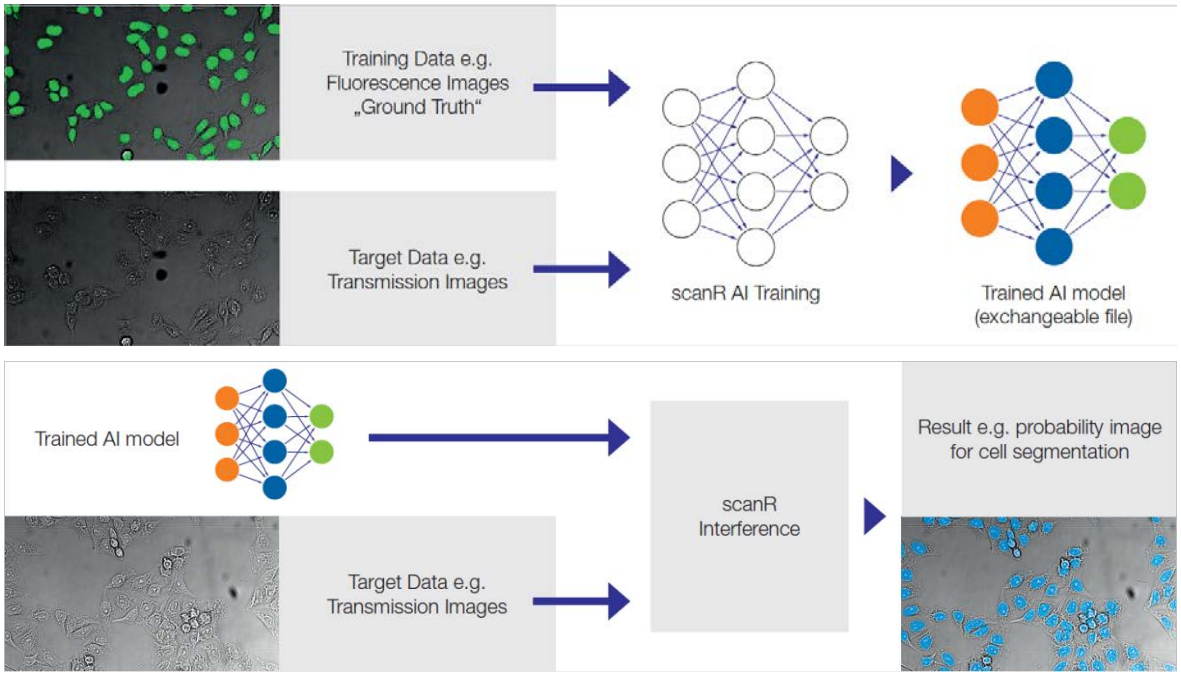
For example, it is easy to train a neural network to detect nuclei in extremely low-light conditions. This capability has important applications because it dramatically reduces the amount of light exposure to the sample without trade-offs in analysis potentials and accuracy. To train a neural network for nuclei detection in noisy image conditions, the scan^R Acquisition is set up to acquire pairs of well-exposed images and weakly exposed images of a larger part of a sample, for example, a whole well plate. This scan is analyzed by scan^R Analysis using the well-exposed channel for object detection. The entire image acquisition and object detection process takes only little time and can be performed easily using the standard scan^R workflow. The data then is fed into neural network training and results in a trained neural network that has learned to detect the nuclei directly from the weakly exposed channel.



After this one-time training phase, it is no longer necessary to acquire well-exposed images. Instead, the nuclei can directly be detected and measured from the weakly exposed channel, reducing the light exposure and phototoxicity dramatically and saving acquisition time. The trained neural network can be used on the same scan^R system or exported and distributed to as many other scan^R systems or analysis workstations as required, enabling all of them to perform the same kind of analysis.

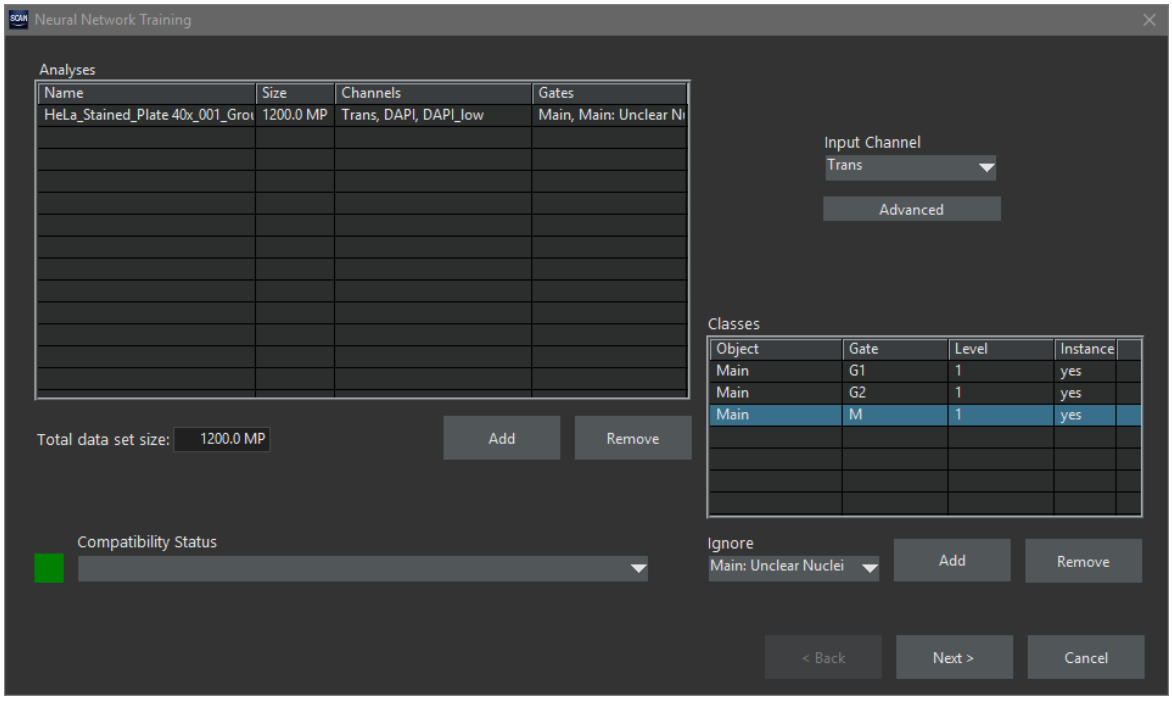


The general concept of self-learning microscopy can transfer information from one or multiple channels acquired only during the training phase, i.e., while developing an assay, to another channel. Applications range from previously impossible image segmentation tasks to quantitative analysis of shallow signal levels, simplifying staining protocols, label-free analysis, and more. A straightforward application of label-free analysis, for example, is the detection, counting, and measurement of nuclei in bright field images.



6.2 Training a neural network

A prerequisite for neural network training is a single or multiple analysis (.sca)-files containing detected (main) objects. These detected objects serve as ground truth and provide examples of what the network shall learn to detect from one or multiple color channels later. The analysis file(s) must also contain the color channel(s) where the neural network shall work on later. During the training phase, the neural network learns from these examples how to detect the objects on the target data.



A new neural model is trained with the following steps.

1. Open the **Neural Network Training** wizard by **AI ▸ Neural Network Training.**

2. In the **Analyses** section, add one or multiple analysis (.sca) files containing the training data.

3. In the **Input Channels** section, select the channel of the target data, i.e., the data the neural network shall work on later.
   Advanced:
   Multiple channels can be selected together. All channels added to this list will be used for detection later; hence all channels need to be acquired later for analysis. Often it is sufficient to add only one channel to the list so that the neural network learns to work on a single channel.
   If the input data contains z-stacks, more options are available:
   Central slice: The in focus slice of the z-stack is used for training. This means the network will not need z-stack data after training. The z-stack is only used during training to make the trained model robust to possible slight defocus of images.
   Pair of slices or **Difference of slices**: Two slices of the input z-stack are used for training .
   This means, the network will need a z-stack with at least two slices to run on after the training.
   Z-steps: Determines which two z-slices are used from a larger z-stack. The setting of Z-steps=1 means the two slices next to the in-focus slice (+/-1) are used. The setting Z-steps=2 means the slices +/- 2 are used, and so on.
   Scaling: Very large objects are better detected if the internal scaling is set to 50% or 25%.

4. As **Classes** to detect either all main objects (**Main**) or one or multiple gates can be selected to include only objects of this particular gate or these gates.
   Each class has a **level** assigned. Classes of the same level cannot overlap. If classes should overlap (like subobjects and main objects), they should be set to different levels.
   Each class can be defined as containing **instances**. Defining a class as instance makes the training to learn how to separate individual nuclei or cells. Please note that the training data

needs enough examples of well-separated objects .
Furthermore, a gate with objects to be ignored for the training can be provided. Use this **Ignore** gate for objects that are not correctly detected in the training data but are not background. For example, partly detected nuclei or double detections (nuclei not correctly separated in the training data) can be ignored.

5. On the next page of the wizard, the **Training Configuration** and the **Number of Iterations** can be changed. The default “Standard Network” and 25,000 iterations are a good start for most applications with a single class. However, networks with multiple classes or instance segmentation require longer training times.

6. Now all required information is provided, and the training can be started.

Depending on the training configuration and the number of iterations, the training can take between a couple of minutes and many hours. During this time, it is possible to follow the training progress by a **Quality** graph and a selection of validation images generated for five checkpoints during the training.

As validation data, 10% of the total input images are set aside from the training and only used to control (validate) the training results during the training already. One of the quality graphs and the preview results on the image data are only calculated on the validation images to ensure that the training is successfully generalizing rather than only improving on the training data itself.

Any **Checkpoint**, except the initial **Reference** checkpoint, can be saved as **Neural Network Model**. Judging the validation images for the different checkpoints makes it easy to select the best training result as the neural network model used in later analyses. The **Final Checkpoint** will often be the best training result because it has trained for the longest time, but in special cases, the training result can be better at an earlier training state.

6.3 Managing neural networks

Any neural network saved from any checkpoint will be added to an internal model database. It is possible to save neural network models as **private**, i.e. only visible to the current user, or as **public**, i.e. visible to all users. The neural network database can be maintained in the **AI ▸ Manage Neural Networks** dialog. Here, networks can be deleted, exported or imported. By exporting a network and importing it on another system, it is possible to deploy established neural networks to many systems and run assays based on these networks on other systems.

6.4 Standard detection workflow

A neural network model which has been trained for a single class is a detection network. It can be used to segment foreground objects and separate them from the background. Section 3.9.3, *Neural Network*

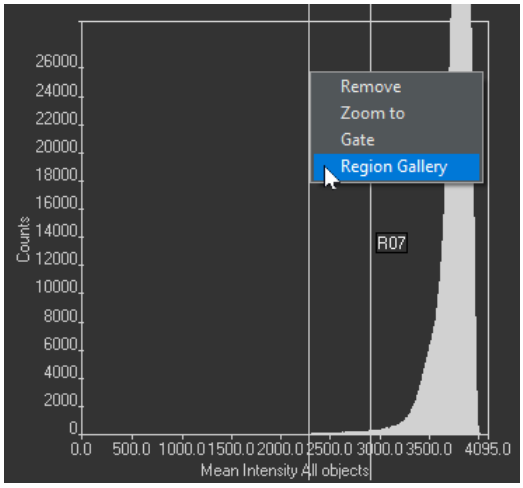
Processing, describes how to use a trained model in an assay to analyze new data. Basically, the neural network model is used to create one or multiple new Virtual Channels. They contain segmented **objects** and **probability maps**, meaning the probabilities per pixel that this pixel belongs to the foreground. The probabilities are gray values (from 0 .. MAX[1]) where 0 corresponds to a pixel where the network is very sure that it is background and MAX where the network predicts foreground (or a particular class) with maximal certainty. In other words, the probability maps give an overview of the ***confidence*** of the network.

Any Virtual Channel containing segmented objects can be used directly in the **Main Object** step of assay setup.

Alternativly, a Virtual Channel containing a probability map can be used in combination with an ***Intensity***-based Object Finder similar to the definition of objects in any other kind of channel. Typically, a value for a **threshold** of about ~50% probability is used to detect all object pixels where the network predicts higher probability of foreground than background. The value can be decreased (for example ~= 25%) to also include pixels with lower confidence or increased (for example ~= 75%) to only include pixels with very high confidence.

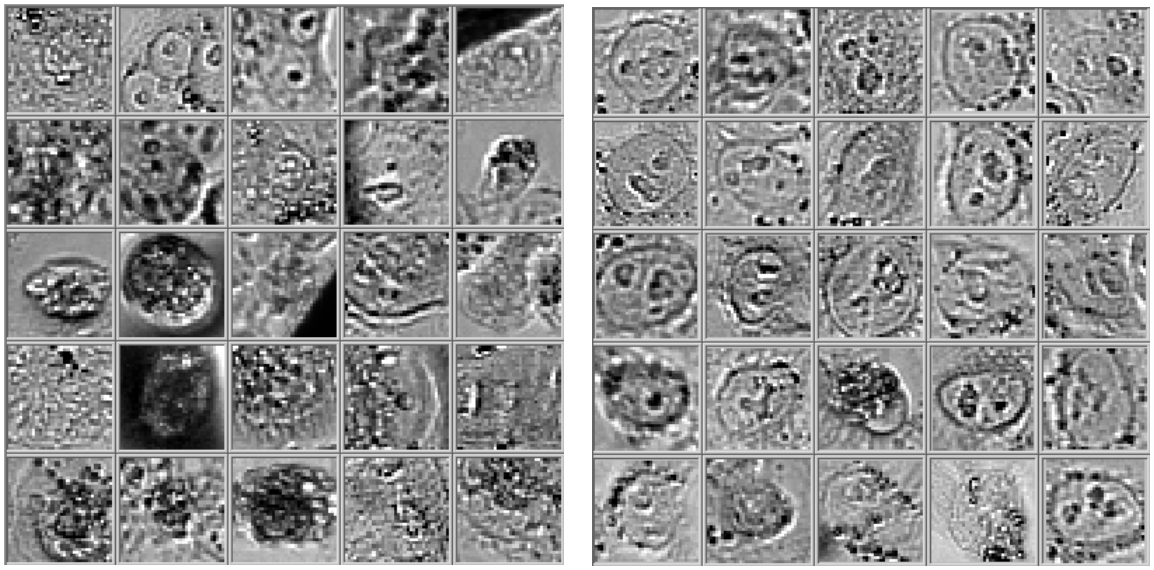
The object masks derived from the Virtual Channel can be used just like any other object masks in scan[R]. In particular, they can be used for measuring all kinds of parameters as described in Chapter 3.6, *Measurement parameters*, including geometrical parameters of the objects (like area, circularity) and intensity parameters (like mean intensity in any fluorescence channel).

Of particular interest is the parameter ***mean Intensity*** of the probability channel itself. When this parameter is defined for analysis, it can be used in histograms for directed investigation on those objects where the network has only lower confidence. If the network should have false detections, these typically happen with lower confidence, and investigation of these potentially problematic detections can strengthen trust in the network and help to understand its capabilities and limitations thoroughly.



For example, it is straightforward to inspect galleries of objects with the lowest 0.5% of confidence (left) and understand the reasons the neural network is less confident than with the other objects (right). Typically the reason is that those objects are not very typical and leave room for uncertainty for both, the human expert and the neural network.

[1] The value of MAX depends on the assay bit depth (16 bit or 12 bit). For 16-bit assays it is MAX = 65,535 = 100% and for 12-bit assays it is MAX = 4095 = 100%.



This also opens an easy way of optimizing an analysis towards a higher ***sensitivity*** (including more objects even if they are less certainly of the correct class) or towards a higher ***specificity*** (include only objects with very high confidence that they are of the correct class). For this purpose, a gate can be created on the histogram of the ***mean Intensity*** of the probability channel and set for higher or lower values.

6.5 Standard classification workflow

While a network trained for a single class detection creates only one pair of Virtual Channels, which describes all results for this class (i.e. objects and probabilities), a network trained for multiple classes creates a set of Virtual Channels with different results.

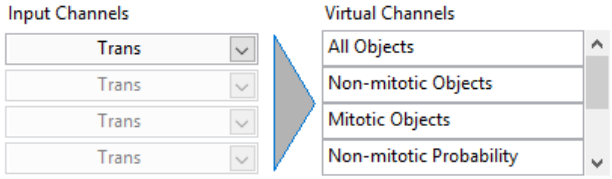
The first Virtual Channel (default name: ***All Objects***) contains all objects of all classes. This channel can be used directly for the definition of all objects as **Main Objects** or **Sub-objects**.

Then, there is one pair of Virtual Channel fors each class. These contain the objects of the class and the probability maps which give the probability for the particular class. It can be used for detection and segmentation of objects of individual classes.

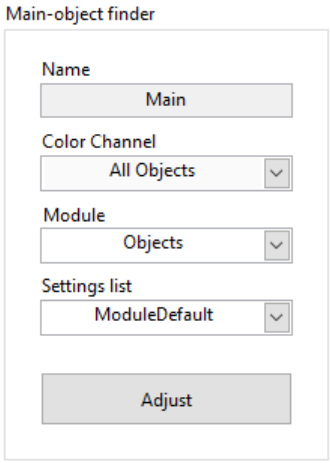
The last Virtual Channel is the ***Classification*** itself. It encodes for each pixel to which class it most likely belongs. There are special parameters available in the **Assay – Parameters** tab to work on the classification Virtual Channel and derive the object classification for the analysis.

6.5.1 Example for cell cycle analysis

In this example, a neural network has been trained to detect and classify mitotic and non-mitotic nuclei from bright-field images. The network creates six Virtual Channels:



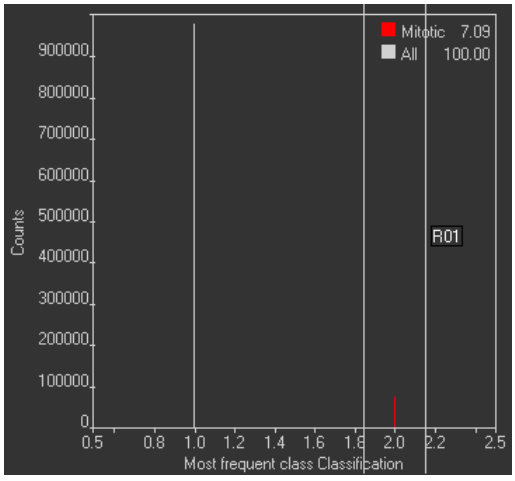
The *All Objects* channel is used to detect all objects directly.

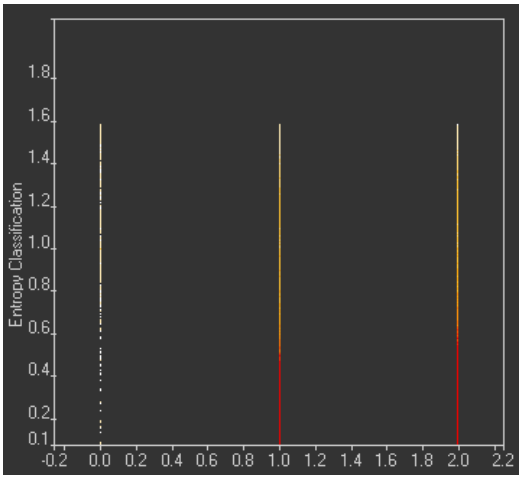


The following parameters are defined in **Assay Setting** – **Parameters**:

p7	Mean Intensity	Mitotic Probability	Main
p8	Mean Intensity	Non-Mitotic Probabilit	Main
p9	Most Frequent Class	Classification	Main
p10	Entropy	Classification	Main

The most straightforward way to create a simple classification is done by using the ***Most frequent class*** parameter on the ***Classification*** channel. The corresponding histogram (left) shows all objects with class = 1 (non-mitotic cells) and class = 2 (mitotic cells). It is now easy to gate for one or the other or both classes for further analysis.





Entropy is an advanced parameter that describes the distribution of different classifications for different pixels within an object. A value of entropy near 0 means that the vast majority of pixels within a particular object share the same class while higher values mean that there are different classes within one object.

By investigating objects with high entropy, the reasons for classification uncertainties can be better understood. Gating on objects with lower entropy increases specificity, while including objects with higher entropy increases sensitivity.

7 Appendix

7.1.1 File conversion

scan^{R} Analysis is able to analyze image data acquired with a third party imaging system as long as the data are stored as (monochromatic) 16-bit single TIFF images. The file names must allow the assignment of the images to individual positions, time stamps and color channels.

The command **Scan ▸ Custom conversion** opens the **Custom Conversion** dialog that allows setting up conversion rules. The creation of conversion rules is a four-step process, which will be described, in the following chapters.

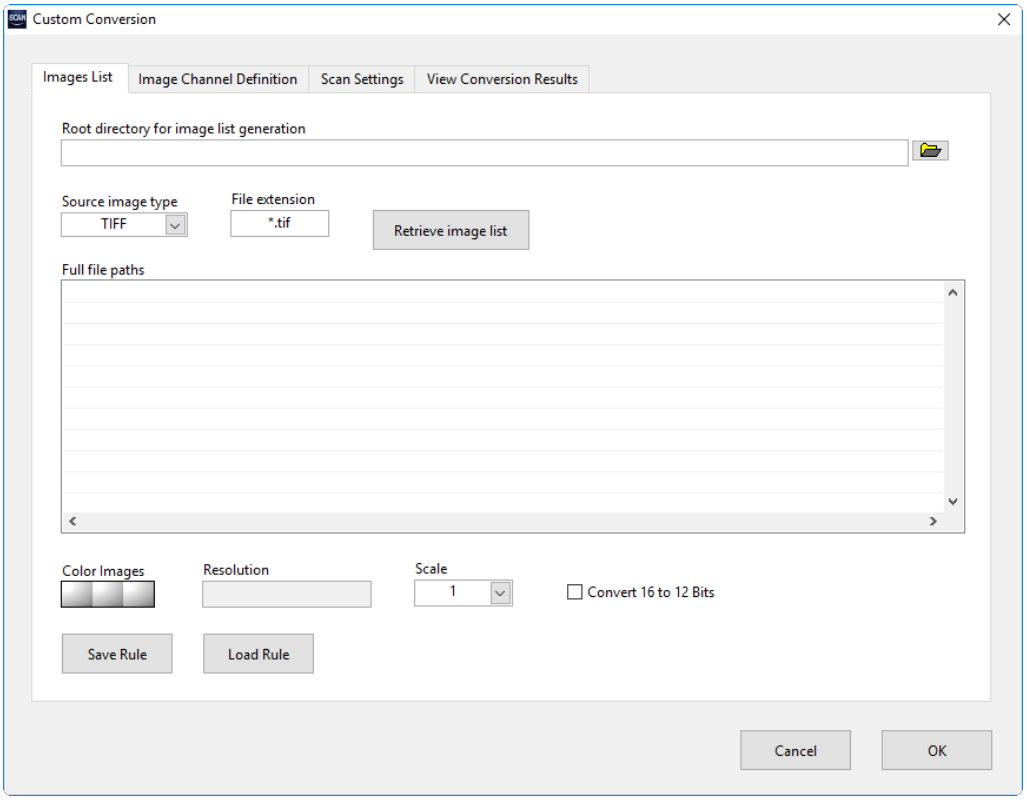
Save rule. This function saves the current conversion settings as a Custom Pattern file (*.cr).

Load rule. This function allows loading a stored set of conversion settings.

7.1.2 Images list

Root directory for image list generation. Click on the **folder** button on the right of the box to open the navigation window. Navigate into the storage directory and click **Select Curr. Dir**.

Source image type. Select the correct image type (TIFF, JPEG, PNG or BMP) from the drop-down list.



File extension. This is set automatically when the **Source image type** is chosen. The **file extension** serves to filter the content of the selected directory

Retrieve image list. This function reads the matching files into the **Full file paths** list. The arrow buttons allow easy browsing and verification of retrieved image data.

7.1.3 Image channel definition

If the file names of the third party images contain codes for Well, Position, Z-layer and Time, scanR will sort them automatically upon conversion into scanR format. The **Image Channel Definition** tab is used to define the conversion rules for this.

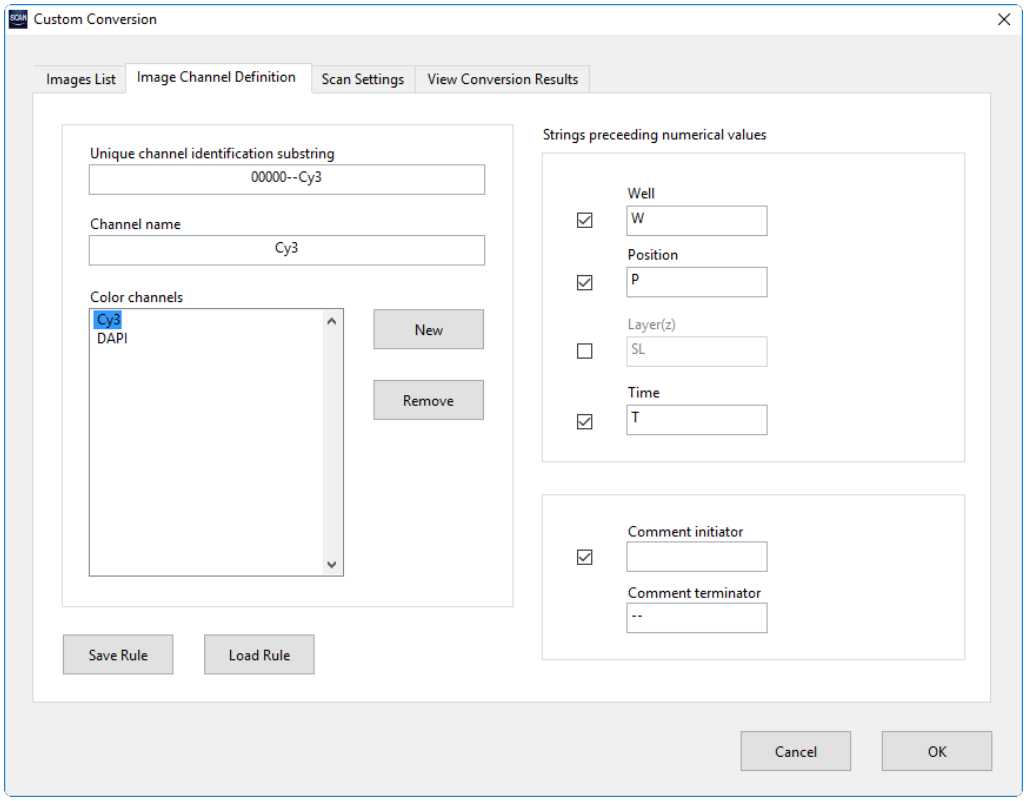
Unique Channel Identification String. If the original file names contain a certain expression – or **Identification String** – that classifies them as belonging to a certain color channel, enter it here.

Channel Name. Enter the **Channel Name** to be used in scanR here.

New. Click here to add the newly defined channel to the **Image Channel** list.

Well, **Position**, **Slide (Z)**, **Time RegExp**. If the file names contain information about Well, Position, Z-layer and Time in form of regular expressions (**RegExp**), scanR can group them when the corresponding code is entered here.

An overview of the syntax of regular expressions can be found under http://www.regular-expressions.info/tutorial.html.



Comment RegExp Initiator / Terminator. If the file names contain information about wells, scan^R can use this as **Well Description**, see for example Chapter 2.7, *Interactive plate result view*. Enter the first (**Initiator**) and last (**Terminator**) alphanumerical digit(s) in the respective boxes.

7.1.4 Scan settings

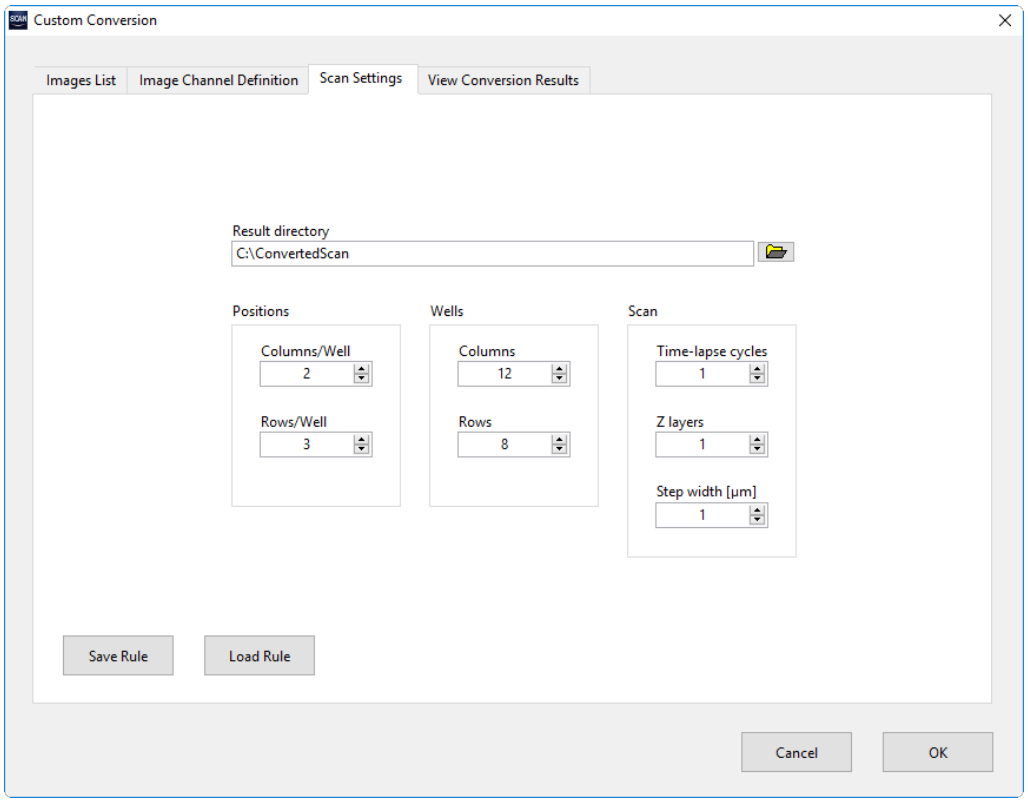
Scan settings are used to reconstruct a minimal configuration file for the data conversion. scan^R needs information about the used multi-well plate and the arrangement of the images within the single wells in order to represent the data properly in the graphical user interface

Result Dir. Click on the **folder** button on the right of the box to open the navigation window. Navigate to the storage directory and click **Select Curr. Dir**.

Positions: Columns/Well and **Rows/Well**. Set the number of image columns and rows, respectively, per well.

Wells: Columns and **Rows**. Set the number of columns and rows, respectively, of the multi well plate.

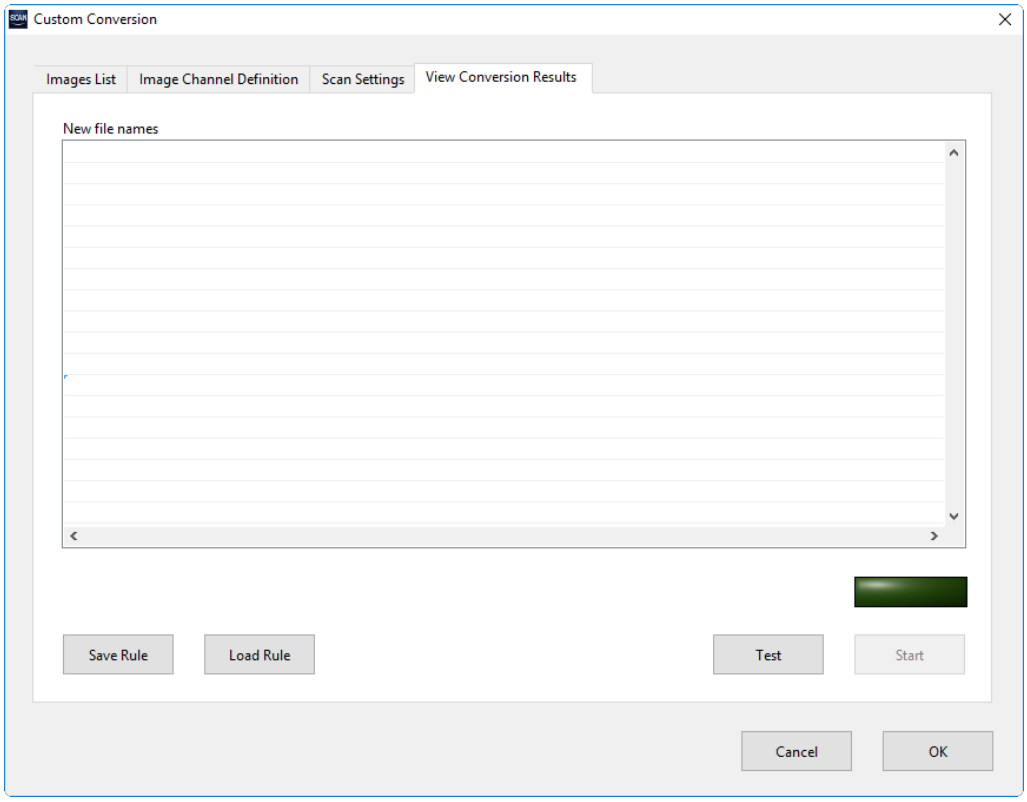
Time-lapse cycles. Set the number of time points for this scan.



7.1.5 View conversion results

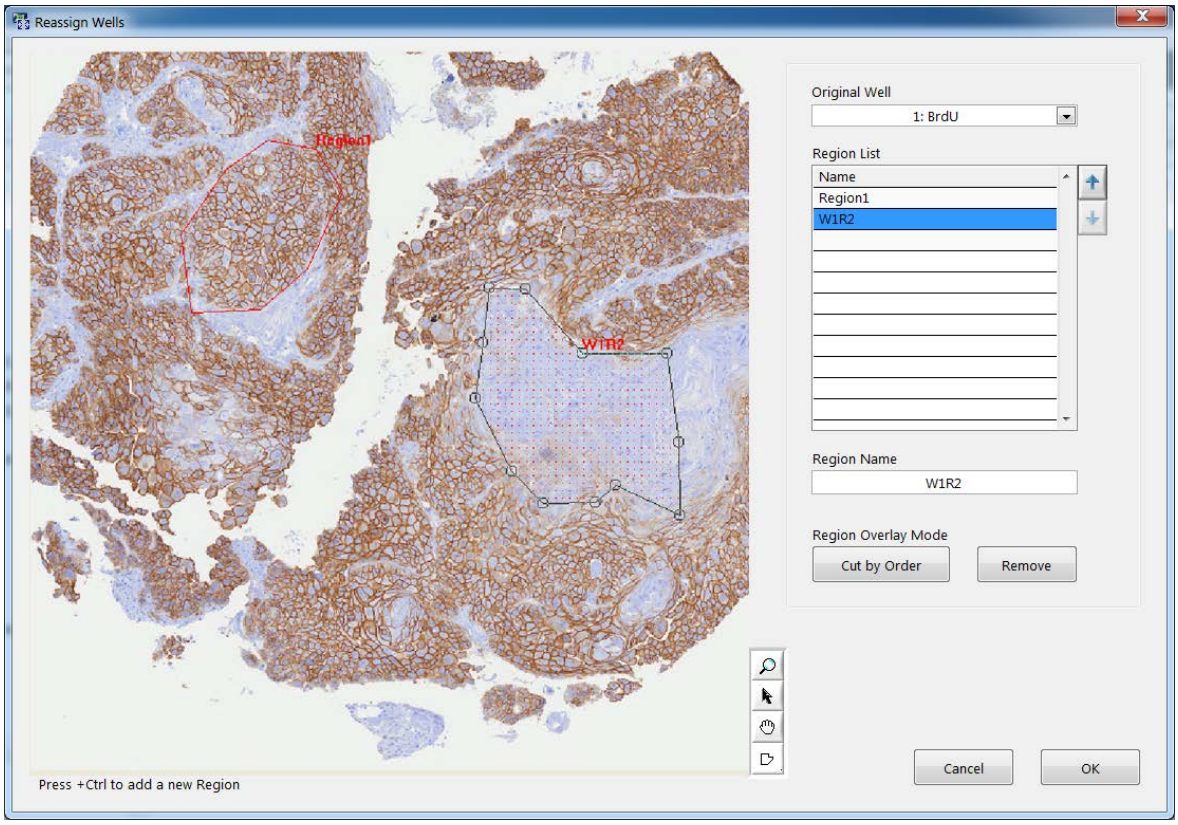
Test. Upon clicking this button, scan^R converts the file names and lists them in the **New File Name** list. This allows checking whether the conversion rules yield the desired result.

Start. Click here to start the file conversion process. The **Start** button becomes active once a conversion rule is defined or loaded. It initiates the file conversion. This may be time consuming because an entire new set of images is generated and stored.



7.2 Reassign wells

In some cases, especially for histological samples, it may be of interest to define special regions within one well (e.g. tumor and healthy tissue). The regions to be analyzed can be defined in the **Reassign Wells** window. To open this window, go to **Scan ▸ Reassign Wells….** On the left side of the dialog the image of the currently selected well will be displayed.



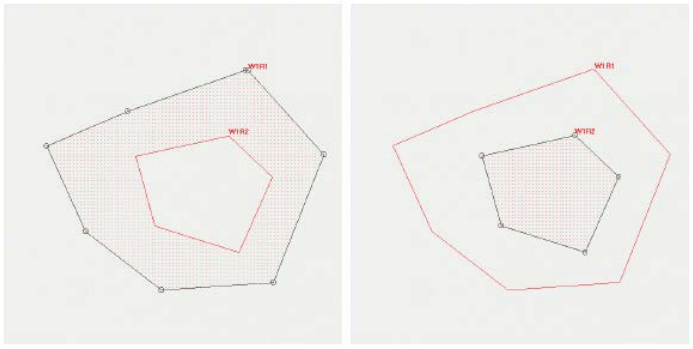
To change the well that is displayed on the left, select another well in the **Original Well** drop-down list. To navigate in the well image use the **Zoom** and **Move** mouse tools. Use the **polygon** tool to create a polygon around the region of interest, and close the polygon with a double-click. A new region will appear in the **Region list**. The default name is W1R1 (Well 1, Region 1). You can enter a meaningful name in the **Region Name** field instead. It is possible to assign the same name to two regions in order to group these regions.

To create further regions use the **polygon** tool and press the **Ctrl** key on the keyboard at the same time. Otherwise the first region will be replaced by the new region. With the **Selection** tool, the nodes of the existing regions can be displaced (indicated by a double arrow cursor) or the whole region can be moved (indicated by the crossed arrow cursor).

If more than one region is created and some of the regions are overlapping, you can choose the **Region Overlay Mode**:

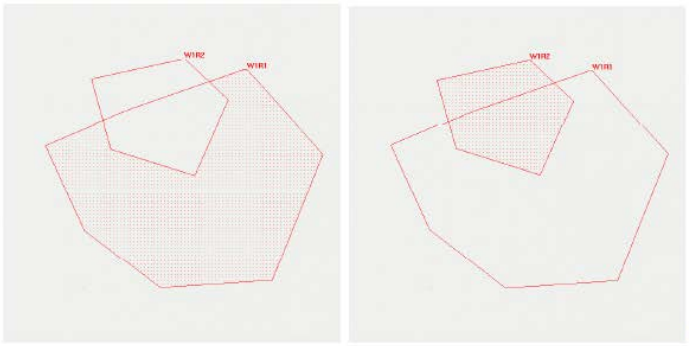
- **Order Independent**: The overlapping area will not be cut. The complete area inside a defined region belongs to this region. Overlapping areas will belong to two regions.
- **Cut by Order:** The overlapping area of a region further down the **Region List** will be cut from the higher region. By this, doughnut-like regions can be defined.

Order Independent

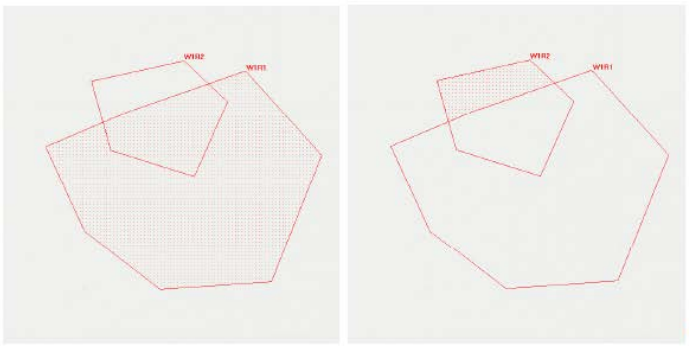


Cut by Order

The **Cut by Order** mode depends on the order of the regions in the **Region List**. To change the order of the regions, mark the region to be moved in the **Region List** and use the arrow icons to move this region up or down in the list. The region that is lower in the list will cut the overlapping area from the region that is higher in the list.



Cut by Order W1R1 above W1R2



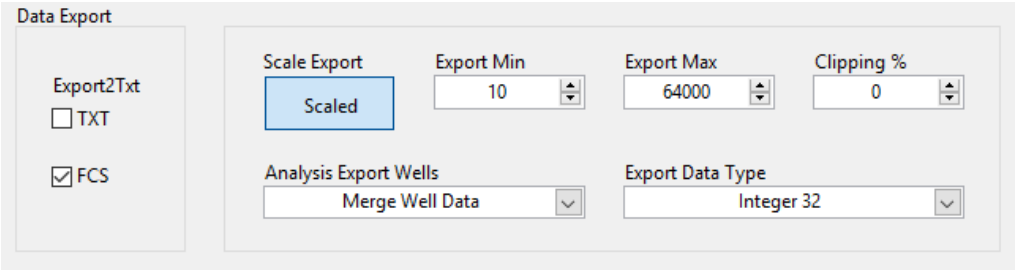
Cut by Order W1R2 above W1R1

The new “virtual” wells that are defined in the **Reassign Well** dialog will fully replace the original wells in the analysis. This means they replace the original wells in the **Assay Settings** dialogs (see Chapter 3, *Assays*), the front panel and the **Well Results** (see Chapter 4.4 *Well results*).

7.3 FCS export functionality

scan^R is able to export single cell data to ISAC FCS 3.0 Standard files. Please note, that this standard is not fully implemented by most of the programs that specify FCS 3.0 import support. Therefore some settings modifications for export might be necessary depending on the specific limitations of these programs. In practice especially the FCS 3.0 requirements for double and integer 32 bit data support as well as multi dataset FCS files might not be implemented because most flow cytometers produce integer 16 bit data in single data sets. It is therefore recommended to test the FCS 3.0 compatibility before deciding which FCS program is to be used for the analysis. Most manufactures provide demo programs for testing.

In order to export FCS data open the **Preferences** dialog from the menu bar.



Export Data Type. Here you can choose the data types exported (Integer 16 / Integer 32 / double (64-bit floating point) according to FCS 3.0 standard. It is recommended to use the highest precision that is supported by the program intended to import the fcs files.

Scale Export Scaled (On/Off). This button activates the rescaling procedures during FCS export. Switch off the scaling if you want to compare datasets from two separate export procedures quantitatively. If activated, the scaling settings (Min/Max/Clipping) will be used.

Export Min / Export Max / Clipping %. Settings used for rescaling the original data according to:

x_fcs = x*PnG + PnOffset, with:

x_fcs	exported data points
x	the original data points and
n	parameter number
PnG	automatically determined linear scaling factor (Gain)
PnOffset	automatically determined scaling offset

The resulting scaling offset and the scaling factor are stored in the FCS file header in the keywords PnG and PnOffset. If you switch on the scaling then the range of the original data will be scaled to fit into the given export range. PnG and PnOffest are calculated from the original dataset and the scale settings according to:

Export Range = Export Max - Export Min

Original Data Range = Original Data Set Max Value - Original Data Set Min Value

Clipped Range = Original Data Set Max Value (ignoring the largest clip %) - Original Data Set Min Value (ignoring the smallest clip %)

PnG = Export Range/Clipped Range

PnOffset = Export Min - PnG*(Original Data Set Min Value (ignoring the smallest clip %))

Analysis Export Wells (FCS Settings):

Merge Well Data. All Wells are taken together to a single dataset

Single File. Exports Wells into separate datasets in a single FCS file according to FCS 3.0 standard (this Multi-dataset format is not supported by all FCS programs)

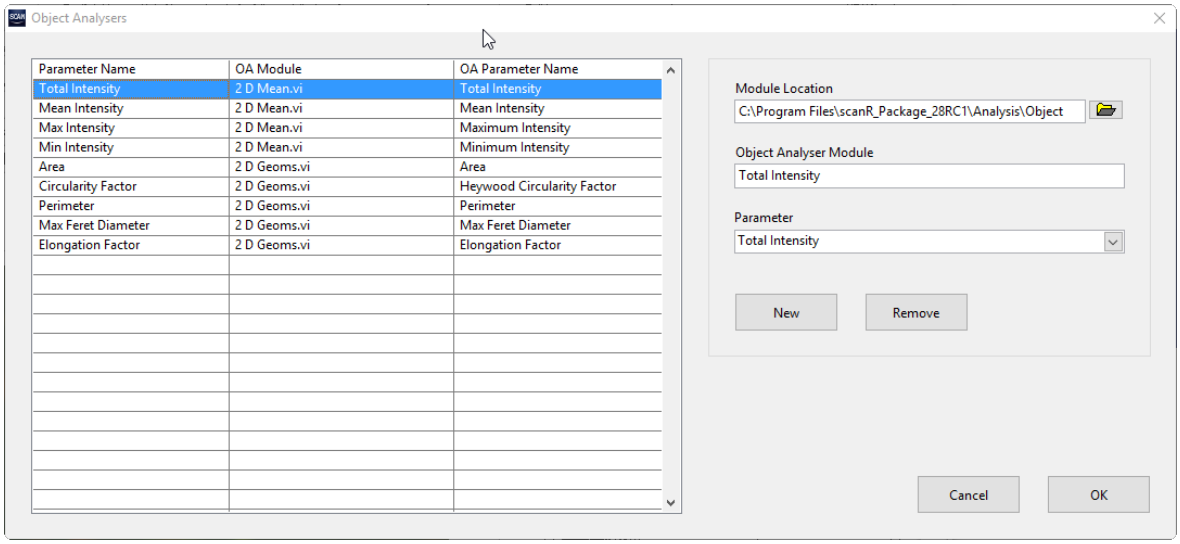
Separate Files. The export table function creates a single FCS file for every well.

> **i16 export (FlowJo™).** When using i16 export and importing with **FlowJo™**, **FlowJo™** uses the gain to rescale the data but is then not able to go back to the logscale. Therefore it is recommended to use raw mode with i16 and FlowJo™. To switch to logscale use then the FlowJo™ function: Add derived ... and convert the required parameters to log. Doubles and Multi-dataset FCS Files cannot be read by FlowJo™. Please note, that if you have data within a X,Y logscale and you rescale the X,Y data with different scaling factors (as might happen with the automated scaling during export because of range differences) there might be a shape change.

An alternative to FlowJo is FCS Express™ (http://www.denovosoftware.com).

7.4 Libraries

7.4.1 Object analyzers library (OAL)



The password-protected **Object Analyzers Library** can be accessed through **Modules ▸ Object Analyzers Library**.

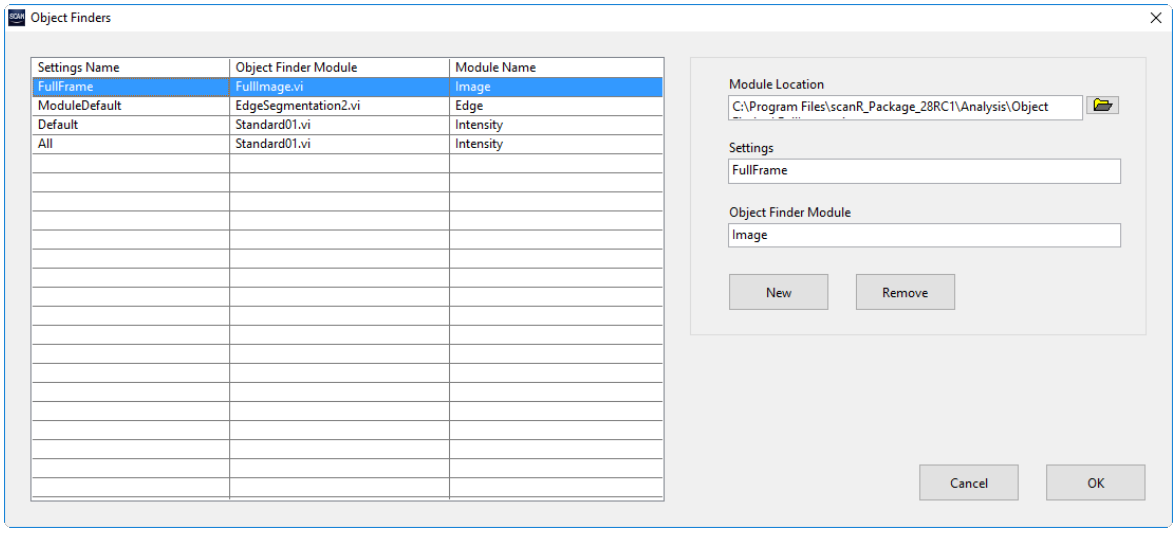
It allows managing the set of parameters selectable in the **Edit Assay** tab **Parameters**; see Chapter 4.6, *Measurement Parameters*.

Add. Click here to add a new entry to the list.

Parameter. Select the desired parameter from the drop-down list to add it to the list.

7.4.2 Object finders library (OFL)

The password protected **Object Finders Library** can be accessed through **Modules ▸ Object Finders Library.** It lists and allows administrating the available **Object Finder Modules**. Each module may have different settings lists and may thus appear several times.



Add/Remove. Click here to add or delete a new entry to the list.

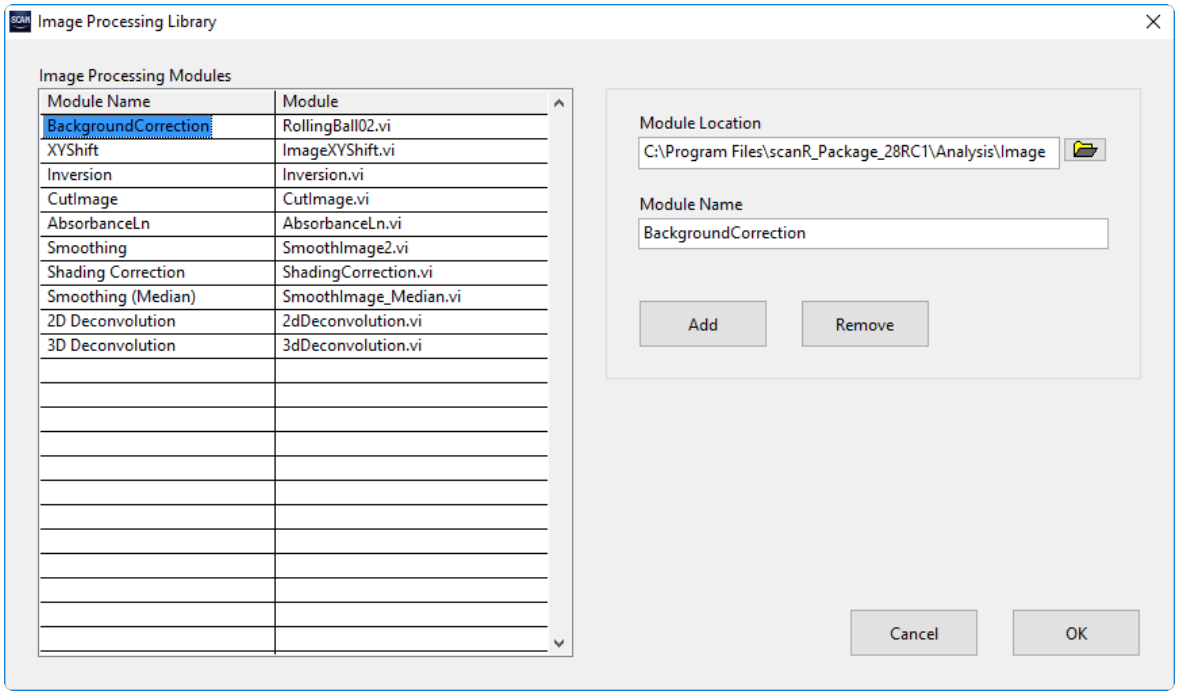
7.4.3 Image processing library

The password-protected **Image Processing Library** can be accessed through **Modules ▸ Image Processors…**

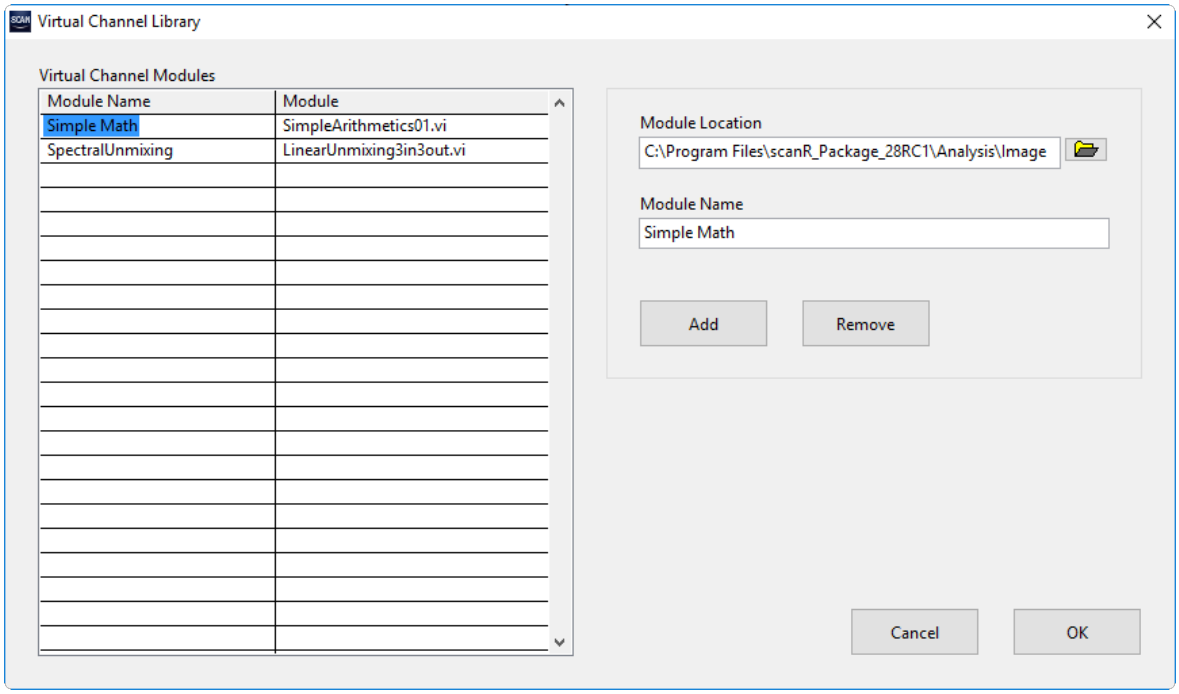
It allows managing the set of available Image Processing algorithms in the **Image Processing modules**.

Add Module. Click here to add a new entry to the list.

Remove Module. Click here remove an entry from the list.



7.4.4 Virtual channel library



The password-protected **Virtual Channel Library** can be accessed through **Modules ▸ VC Processors…**

It allows managing the set of available Virtual Channel algorithms in the **module selector on the Edit Assay ▸ Virtual Channels tab**.

Add Module. Click here to add a new entry to the list.

Remove Module. Click here remove an entry from the list.

7.5 Valid functions for derived parameters

abs(*x*)	Absolute Value	Returns the absolute value of *x*.
acos(*x*)	Inverse Cosine	Computes the inverse cosine of *x* in radians.
acosh(*x*)	Inverse Hyperbolic Cosine	Computes the inverse hyperbolic cosine of *x*.
asin(*x*)	Inverse Sine	Computes the inverse sine of *x* in radians.
asinh(*x*)	Inverse Hyperbolic Sine	Computes the inverse hyperbolic sine of *x*.
atan(*x*)	Inverse Tangent	Computes the inverse tangent of *x* in radians.
atanh(*x*)	Inverse Hyperbolic Tangent	Computes the inverse hyperbolic tangent of *x*.
ceil(*x*)	Round Toward +Infinity	Rounds *x* to the next higher integer (smallest integer >= *x*).
ci(*x*)	Cosine Integral	Evaluates the cosine integral for any real nonnegative number *x*.
cos(*x*)	Cosine	Computes the cosine of *x*, where *x* is in radians.
cosh(*x*)	Hyperbolic Cosine	Computes the hyperbolic cosine of *x*.
cot(*x*)	Cotangent	Computes the cotangent of *x* (1/tan(*x*)), where *x* is in radians.
csc(*x*)	Cosecant	Computes the cosecant of *x* (1/sin(*x*)), where *x* is in radians.
exp(*x*)	Exponential	Computes the value of *e* raised to the *x* power.
expm1(*x*)	Exponential (Arg) – 1	Computes one less than the value of *e* raised to the *x* power ((*e*^*x*) – 1).
floor(*x*)	Round To –Infinity	Truncates *x* to the next lower integer (largest integer <= *x*).
getexp(*x*)	Mantissa & Exponent	Returns the exponent of *x*.
gamma(*x*)	Gamma	Evaluates the gamma function or incomplete gamma function for *x*.
getman(*x*)	Mantissa & Exponent	Returns the mantissa of *x*.
int(*x*)	Round To Nearest	Rounds *x* to the nearest integer.
intrz(*x*)	—	Rounds *x* to the nearest integer between *x* and zero.
ln(*x*)	Natural Logarithm	Computes the natural logarithm of *x* (to the base of *e*).
lnp1(x)	Natural Logarithm (Arg +1)	Computes the natural logarithm of (x + 1).
log(x)	Logarithm Base 10	Computes the logarithm of x (to the base of 10).
log2(x)	Logarithm Base 2	Computes the logarithm of x (to the base of 2).
rand( )	Random Number (0 – 1)	Produces a floating-point number between 0 and 1 exclusively.
si(*x*)	Sine Integral	Evaluates the sine integral for any real number *x*.
sec(*x*)	Secant	Computes the secant of *x*, where *x* is in radians (1/cos(*x*)).
sign(*x*)	Sign	Returns 1 if *x* is greater than 0, returns 0 if *x* is equal to 0, and returns –1 if *x* is less than 0.
sin(*x*)	Sine	Computes the sine of *x*, where *x* is in radians.
sinc(*x*)	Sinc	Computes the sine of *x* divided by *x* (sin(*x*)/*x*), where *x* is in radians.
sinh(*x*)	Hyperbolic Sine	Computes the hyperbolic sine of *x*.
spike(*x*)	Spike	Generates the spike function for any real number *x*.
sqrt(*x*)	Square Root	Computes the square root of *x*.
step(*x*)	Step	Generates the step function for any real number *x*.
tan(*x*)	Tangent	Computes the tangent of *x*, where *x* is in radians.
tanh(*x*)	Hyperbolic Tangent	Computes the hyperbolic tangent of *x*.

7.6 Export to imaging software

In various scenarios it is desirable to have a closer look at individual cells or cells which are gated for particular properties. For this purpose, scan^R supports the export of all object positions of any defined gate into position lists that can be used in the Olympus cellSens and xcellence high-end imaging software, respectively.

7.6.1 Setting up the export to xcellence functionality

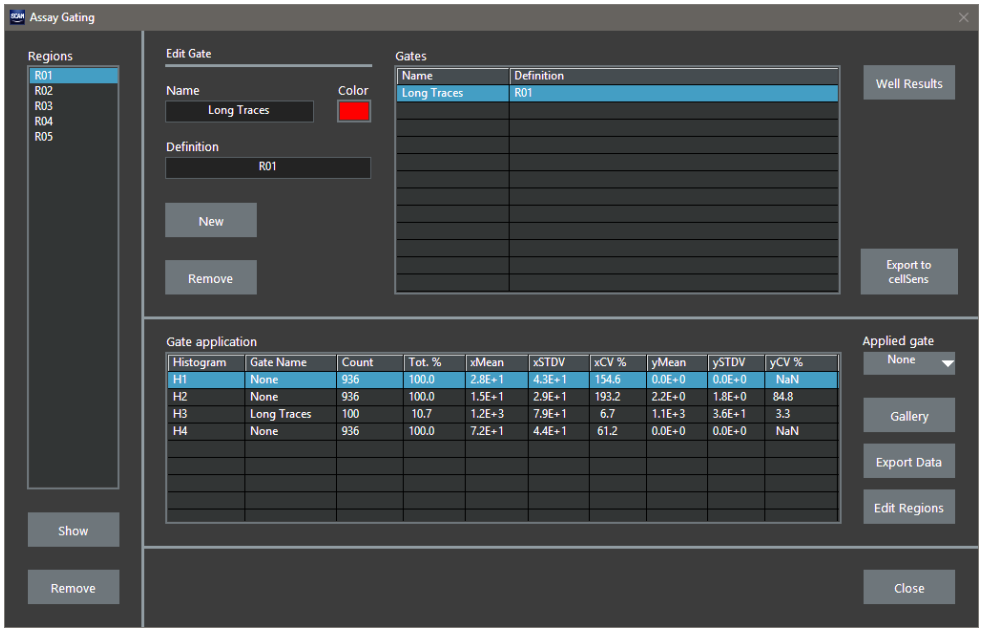
In the preferences (see Chapter 2.2 *Preferences*), the export to imaging software functionality can be enabled by checking the corresponding option. Before the export can be used for the first time, scan^R initially needs the full path to the OBSPOSLISTS.XML file, the xcellence database of position lists. For typical installations the file is located at C:\Users\All Users\Olympus\OSIS\xcellence rt.

7.6.2 Setting up the export to cellSens functionality

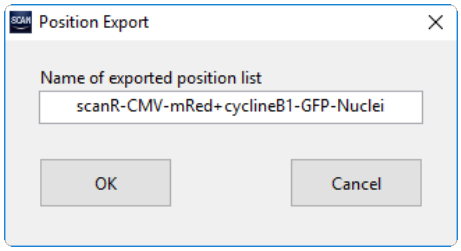
In the preferences (see Chapter 2.2 *Preferences*), the export to imaging software functionality can be enabled by checking the corresponding option. Before the export can be used for the first time, scan^R needs to be connected to cellSens by selecting “Configure”.

7.6.3 Exporting the positions of gated objects

When the export to imaging software functionality is enabled, the **Gate Manager** (see Chapter 4.2.2) reveals an additional button, labeled as **Export to xcellence** or **Export to cellSens**, respectively.



In order to export the positions of all objects within a particular gate, simply select the gate in the list left to the button and press the button. In the following dialog, a suggestion for the **name** of the position list in xcellence/cellSens is presented, which can be edited if necessary.



Pressing the **OK** button finished the export process and the newly created positions list can be used immediately in xcellence or opened in cellSens by the usual steps.

OLYMPUS

www.olympus-global.com

Manufactured by

OLYMPUS SOFT IMAGING SOLUTIONS GmbH
Johann-Krane-Weg 39, 48149 Münster, Germany

Distributed by

OLYMPUS CORPORATION
Shinjuku Monolith, 2-3-1 Nishi-Shinjuku, Shinjuku-ku, Tokyo 163-0914, Japan

OLYMPUS EUROPA SE & CO. KG,
Wendenstrasse 20, 20097 Hamburg, Germany

OLYMPUS CORPORATION OF THE AMERICAS
3500 Corporate Parkway, P.O. Box 610, Center Valley, PA 18034-0610, U.S.A.

OLYMPUS CORPORATION OF ASIA PACIFIC LIMITED
L26, Tower 1, Kowloon Commerce Centre, No.51 Kwai Cheong Road, Kwai Chung,
New Territories, Hong Kong

OLYMPUS AUSTRALIA PTY LTD
3 Acacia Place, Notting Hill, 3168, Australia